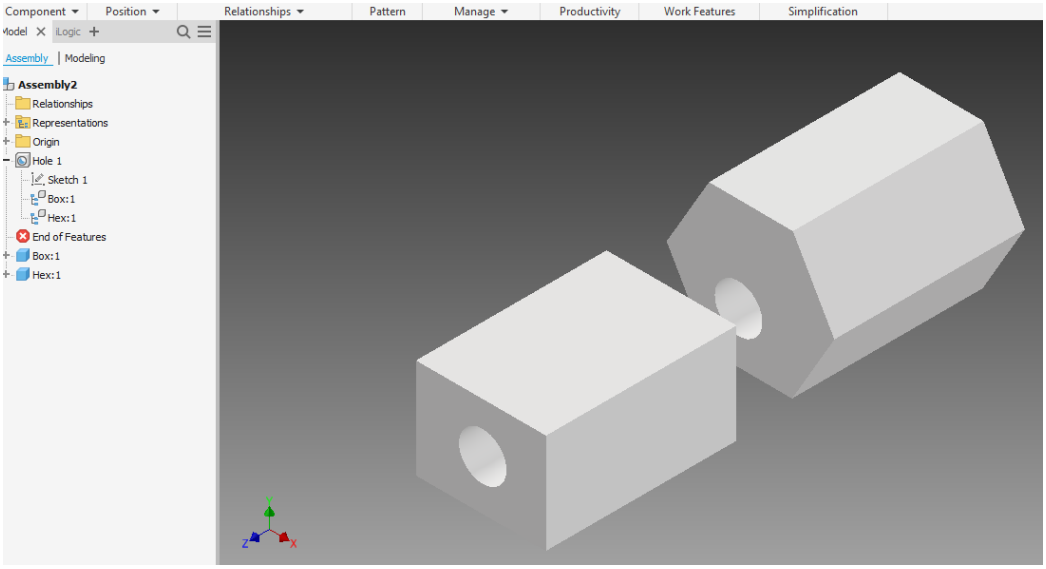
Model contains 2 Parts and an Assembly Feature Hole going through both parts

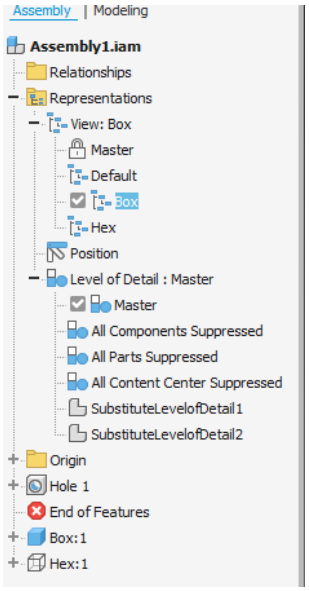


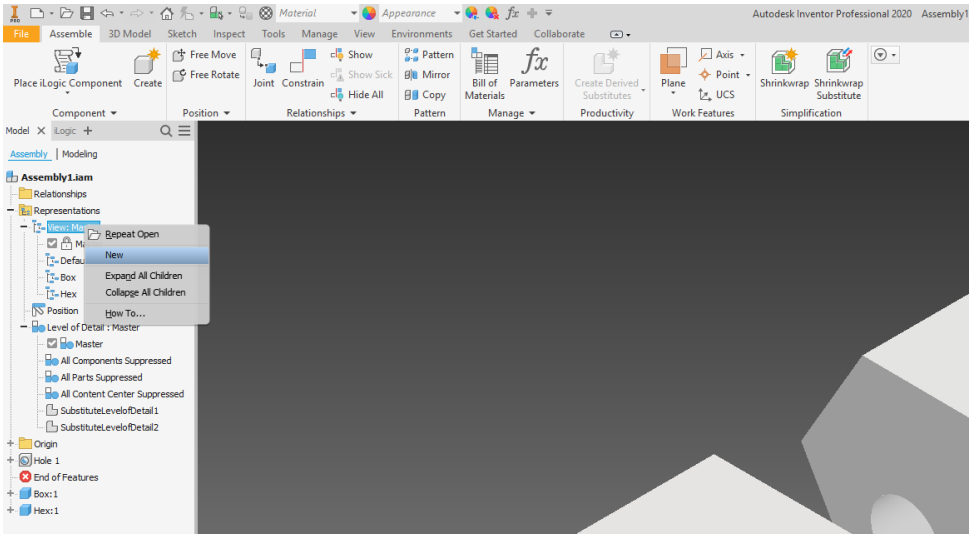
There are multiple ways to extract these solids out

1. Create view representations in the assembly and then use derived assembly to write out the individual solid bodies as separate parts
2. Create 1 single file with multiple bodies and control each solid body separate
3. Either one will work

To create a View Representation in the Assembly

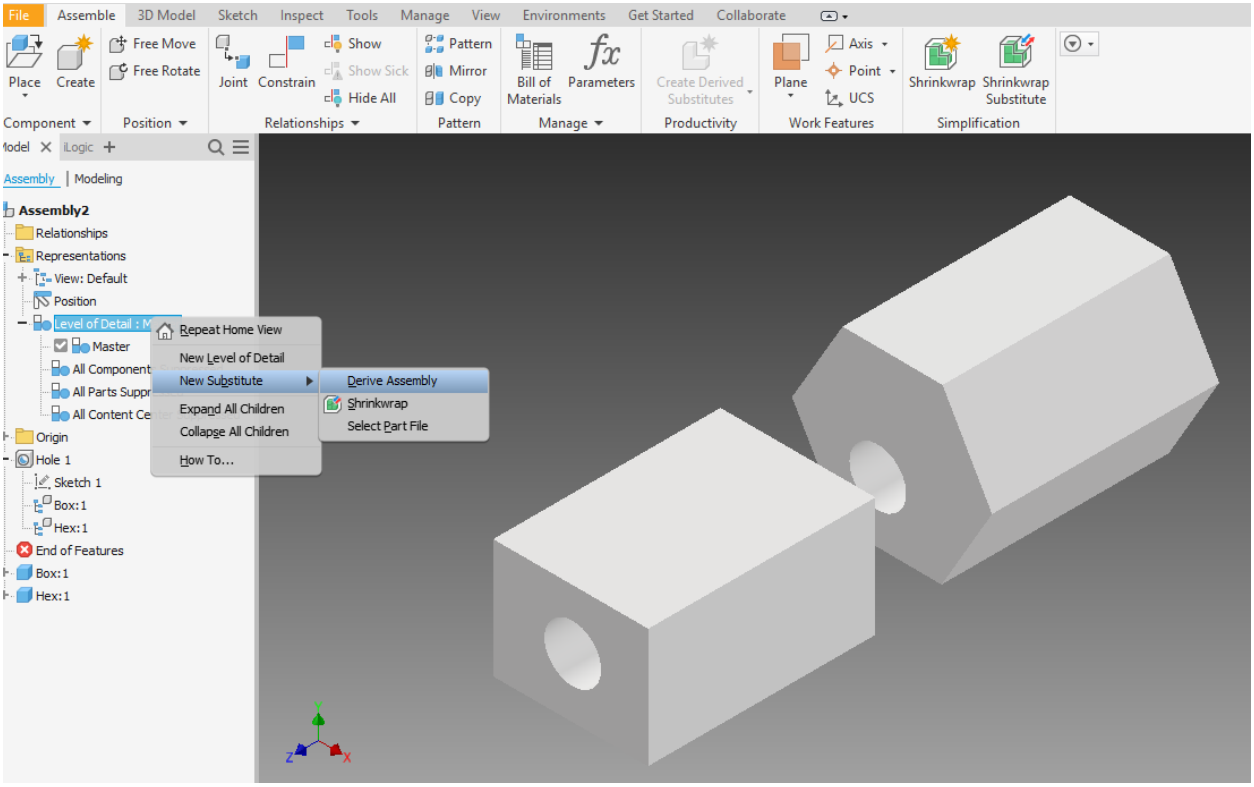
1. Expand View Representations
2. Create New View Representation
3. Create new view representation for each Part
4. E.g. View Rep = Box
5. Turn OFF Hex.ipt
6. View Rep = Hex
7. Turn OFF Box.ipt



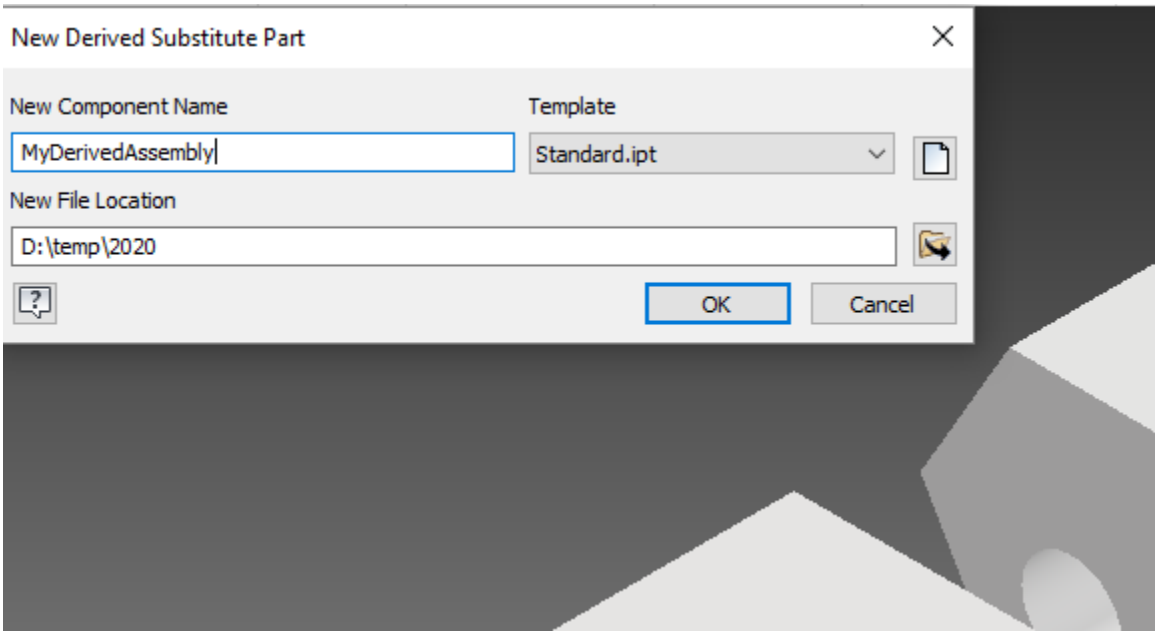


To create a Derived Assembly

8. Expand Level of Detail (LOD)
9. New Substitute
10. Derive Assembly

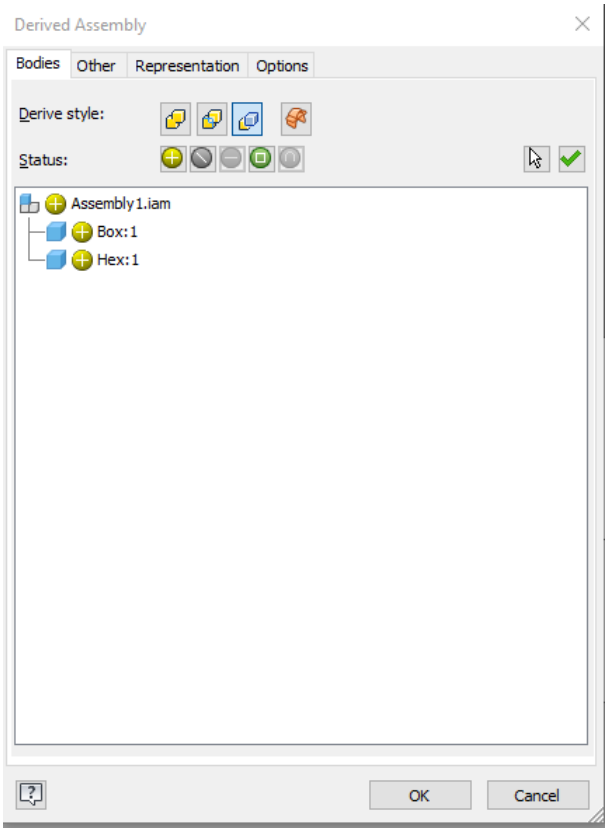


Specify Component Name and File Location and Hit OK

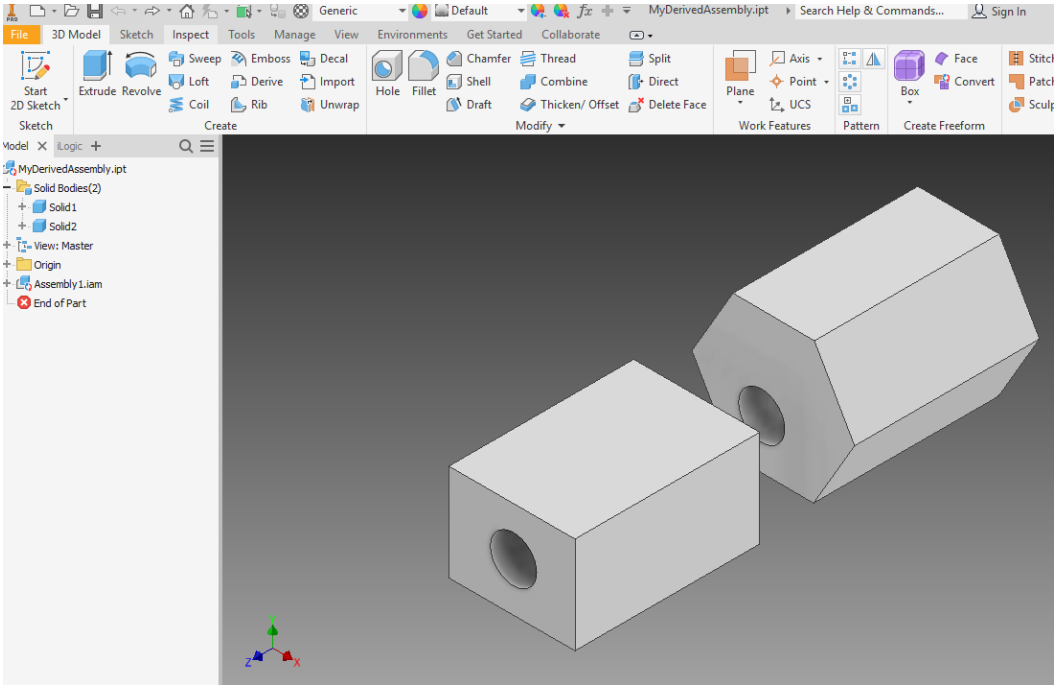


Select Derive Options

Derive Style should be 3rd icon from left – "Maintain each solid as a solid body"



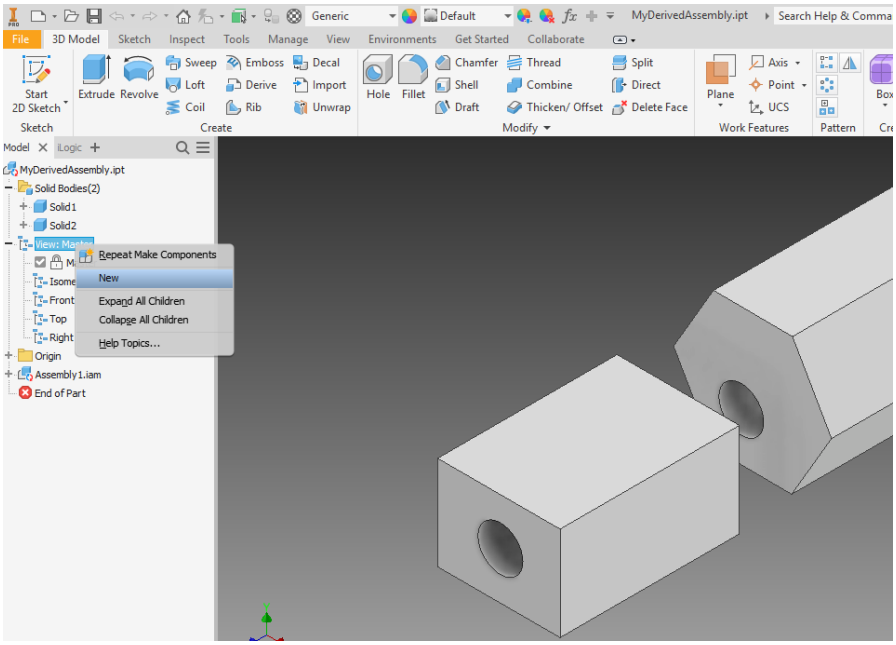
Open the newly created Derived Part



Now you can manipulate each Solid Body Separately

Turn OFF Individual Solids (or surfaces)

Right Click on Solid in the browser and turn off visibility. You can also create new view representations to control visibility states



To Extract Each Solid out to a separate Part File