Sub CreateAndConfigureTankBody()

' This code creates a new copy of the tank body assembly template in our new folder structure

' It then updates the tank body geometry based on values we pass to the assembly

Dim strNewTankBodyFileName As String

' This string represents the new name of our unique, copied tank body assembly file

strNewTankBodyFileName = PROJECT\_PATH & PROJECT\_ID & "\Tank Body Assy\Tank Body Assy - " & PROJECT\_ID & ".iam"

' We first check to make sure the tank body assembly file has not been previously created

If System.IO.File.Exists(strNewTankBodyFileName) = False Then

' This here is a sample of how to make a variable that represents an assembly document

' It uses the Inventor API, which you can use freely (for the most part) throughout iLogic rules

Dim subAssy1 As AssemblyDocument

' This statement tells subAssy1 to represent the template file, and opens it up in the Inventor interface

subAssy1 = ThisApplication.Documents.Open(TEMPLATE\_PATH & "Tank Body Assy\Tank Body Assy.iam", True)

' This code is like selecting "File Save As" in the Inventor interface, and we now have our new file saved

subAssy1.SaveAs(strNewTankBodyFileName, False)

' This can be taken from an iLogic snippet, and is used to insert components into assemblies

' This code inserts our newly created tank body assembly into our master tank assembly file

Dim componentA = Components.Add("Tank Body Assy:1", strNewTankBodyFileName, position := Nothing, grounded := True, visible := True, appearance := Nothing)

' This will now close the new tank body assembly file

subAssy1.Close

' This code calculates the horizontal and vertical locations of our gunline assembly (for later use)

GUNLINE\_HOR\_OFF = Round((TANK\_OD / 2) \* .6667)

GUNLINE\_VERT\_OFF = Round((TANK\_OD / 2) \* .25)

' This code represents our typical manway offset, which is 26" above the bottom of the tank

MANWAY\_VERT\_OFF = -TANK\_OD / 2 + 26

' These statements pass parameters from our master assembly file into the tank body assembly file

Parameter("Tank Body Assy:1", "PROJECT\_ID") = PROJECT\_ID

Parameter("Tank Body Assy:1", "PROJECT\_PATH") = PROJECT\_PATH

Parameter("Tank Body Assy:1", "TANK\_OD") = TANK\_OD

Parameter("Tank Body Assy:1", "TANK\_L") = TANK\_L

Parameter("Tank Body Assy:1", "SHELL\_W\_1") = SHELL\_W\_1

Parameter("Tank Body Assy:1", "SHELL\_W\_2") = SHELL\_W\_2

Parameter("Tank Body Assy:1", "SHELL\_Q\_1") = SHELL\_Q\_1

Parameter("Tank Body Assy:1", "SHELL\_Q\_2") = SHELL\_Q\_2

Parameter("Tank Body Assy:1", "GUNLINE\_SIZE") = GUNLINE\_SIZE

Parameter("Tank Body Assy:1", "GUNLINE\_VERT\_OFF") = GUNLINE\_VERT\_OFF

Parameter("Tank Body Assy:1", "GUNLINE\_HOR\_OFF") = GUNLINE\_HOR\_OFF

Parameter("Tank Body Assy:1", "MANWAY\_VERT\_OFF") = MANWAY\_VERT\_OFF

Parameter("Tank Body Assy:1", "GUNLINE") = GUNLINE

Parameter("Tank Body Assy:1", "MANWAY") = MANWAY

Parameter("Tank Body Assy:1", "F\_HATCH") = F\_HATCH

Parameter("Tank Body Assy:1", "F\_HATCH\_OFF") = F\_HATCH\_OFF

Parameter("Tank Body Assy:1", "R\_HATCH") = R\_HATCH

Parameter("Tank Body Assy:1", "R\_HATCH\_OFF") = R\_HATCH\_OFF

Parameter("Tank Body Assy:1", "INLET\_OFF") = INLET\_OFF

Parameter("Tank Body Assy:1", "INLET\_PIPE\_OD") = INLET\_PIPE\_OD

' Once all the parameters are updated in the tank body assembly file, we want to run their rules

' This will allow the tank body assembly to update all its own parts and components itself

iLogicVb.RunRule("Tank Body Assy:1", "Size Dish")

iLogicVb.RunRule("Tank Body Assy:1", "Calculate and Place Shells")

End If

End Sub