AUTODESK UNIVERSITY 2013

Advanced iCopy Part Development in Autodesk Inventor-Complex Adaptive Geometry

Cortney Sieben – Enclos Corp
Lab Assistants- William Graham, Enclos Corp / Michael Schumacher, Enclos Corp / Stan Wile, Imaginit

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Take your design development to the next level by furthering your knowledge of iCopy functionality. By learning how to create an adaptive frame model to develop complex geometric parts there are limitless opportunities for designers to create intricate adaptive components. This class will go one step further, and explain how to create aligned hole/notch pattern features between two separate components with differing profiles and end conditions. Todays advanced geometric designs and tight project schedules require designers to be quick and efficient with their part modeling. If you are looking for an opportunity to learn how to streamline your parts development through the use of advanced iCopy part modeling this class is for you.

Learning Objectives

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

- Determine when icopy Templates are efficiently used
- Create an adaptive frame model to use in an icopy Template
- Create aligned hole/notch pattern features between two separate components
- Streamline parts development to meet tight project schedules (place iCopy Assembly)

About the Speaker

Cortney has worked with Enclos Corporation, who develops customized curtainwall products and facade systems. She is an AutoCad Certified Professional, has over 7 years experience with Autodesk products and specializes in Autodesk Inventor Software. She has worked on projects such as Wynn Encore, Trump Tower in Las Vegas, Hotel and Residence at LALive, Javits Convention Center Remodel, 250 West 55th Street in New York, and the World Trade Center Transportation Hub in New York.

Email: Cortneysieben@gmail.com

Introduction

Curtain Wall, Window, and Building Products fabricators walk the gray area of architecture and manufacturing. Autodesk Inventor is a solid modeling program focused on generating 2d and 3d models for manufacturing. It has automated tools and environments to speed up the process of making models. On any large scale project there will be thousands of parts and assemblies to model and detail. In Inventor this would mean that every part would have to be modeled, constrained in an assembly and detailed in a drawing. This process will take months of years to finish using traditional 3d modeling methods. Even a small increase in productivity can save a large amount of money over the course of a large project.

Inventor has a wide variety of tools to aid in increasing 3D modeling. Not all of them are pertinent for modeling building products. This course focuses on one area of functionality within Inventor that will automate building product design. This tool is

- iCopy

The iCopy tool comes with Inventor and does not need any add-ons or upgraded packages. This course discusses a specific method to use this tool, but each company has their own design challenges. Use this class to learn to tool and then apply it at your own company.

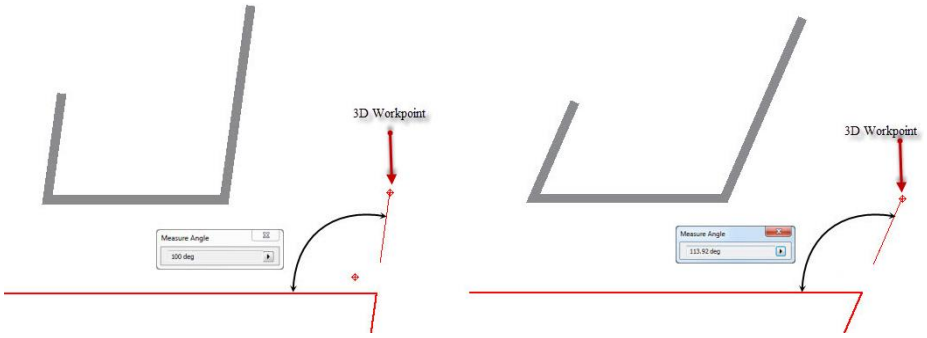
The traditional method for creating extrusions involves drawing the profile, extruding, adding features, and constraining it into an assembly. This process takes time and none of the parts relate to each other. If a design changes, all of the parts affected by the change will have to be updated one by one.

Determine When icopy Templates are Efficiently Used

Many components in curtain wall, window, and building products design are variations of a standard design. A window may get wider. The window may change hardware. A part within that window could change profile slightly. We will focus on the latest of the three examples.

For many types of products it is hard to predict how they need to be modified. The size of a part may be defined but the length and hole pattern will change on each assembly. A project may have hundreds of similar assemblies with various lengths. A huge amount of time could get spent copying assemblies, changing the parameters of the assembly, and constraining them into a general arrangement.

In this example, one of the key parameters that is unknown is the angularity of the profile itself.



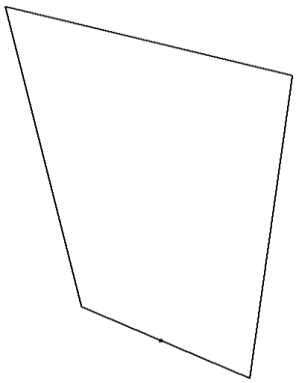
The angularity between the three legs will be based on work planes set within our icopy Template.

Create an Adaptive Frame Model to Use in an iCopy Template

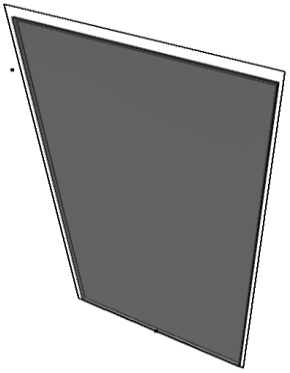
iCopy is a tool that takes a configures assembly and allows it to stretch and change to fit into another assembly. iCopy does two functions, copies an assembly and then changes geometery and parameters to match the design intent.

iCopy needs a few things to work.

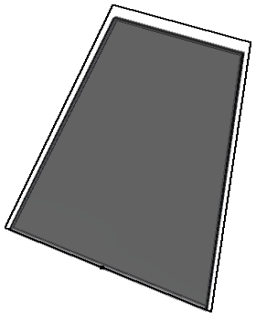
- **A skeletal part.** This part is a referenced part that is stripped down to its bones. It may be a 2D sketch or a set of workpoints. This part needs to be adaptive. Generally the skeletal part should be a 2D sketch that is free to move. If the sketch changes, all of the parts in the iCopy assembly will move with it.
  All of the geometry and parameters that will change in the assembly have to be referenced to this part. This part can get linked to the other components through adaptivity, derived components, of iLogic.



A skeletal model

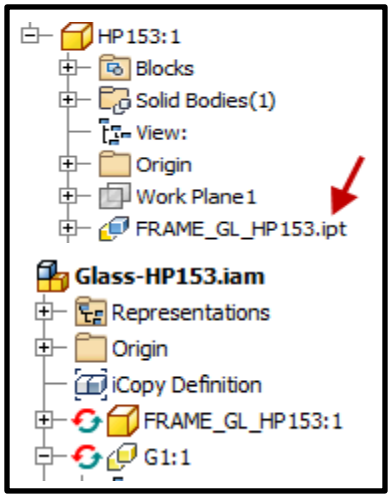


iCopy assembly in its initial state



iCopy assembly after changing the skeletal part.

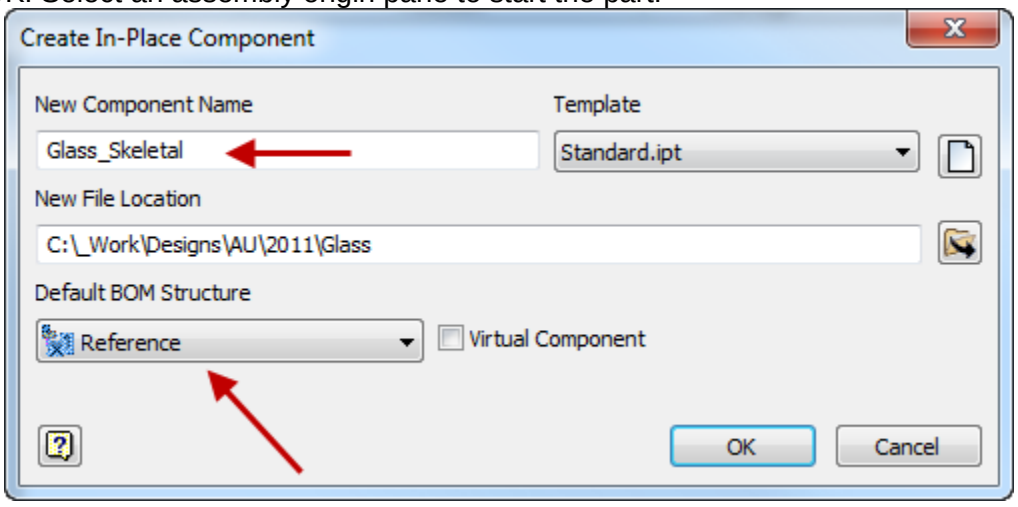
- **Links between files.** For the component to change with the skeletal file, links need to be made between the component and the skeletal file. This links could be created by:
  - **Derived Component –** derive the skeletal part into the other components. The derived geometry and parameters are used to link the skeletal compenents together. From our experience this works. Once in a while the links will break when copying files.
  - Adaptivity – Geometry is projected or linked by Copy Object into the components. Parameters aren't linked, only geometry. Probably the easiest to set up, but the most unstable.
  - iLogic – Link parameters between files using iLogic rules. No geometry is linked. As long as the files keep their labels in the assembly it is stable and flexible. We are not covering iLogic in this course.



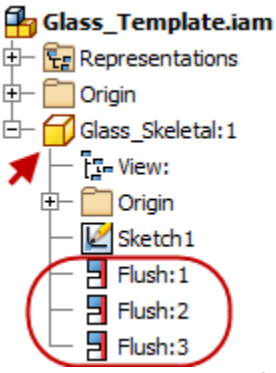
All three methods have their advantages and quirks. It is possible to break the link using any of the methods, so testing of an iCopy model is important before releasing to production. From our experience a mixture of derived components and iLogic works the best.

Steps to create an iCopy assembly

1. Start Inventor
2. Create a new assembly file. Save the file
3. Create a new part file in the assembly. Mark the BOM structure of the part as Reference. Name the file so that it marks the file as a skeletal file. (ex: Glass_Skeletal.ipt). Select OK. Select an assembly origin pane to start the part.

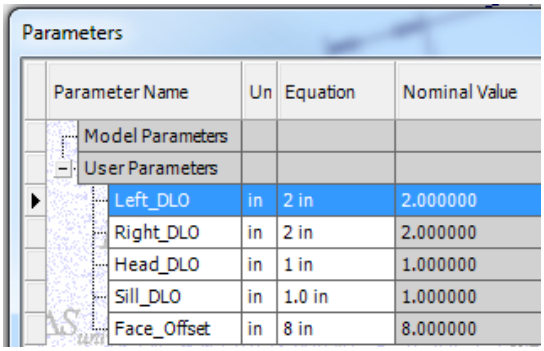


4. Unground the part. Constrain the part to the assembly template. If the part is grounded it will not work.

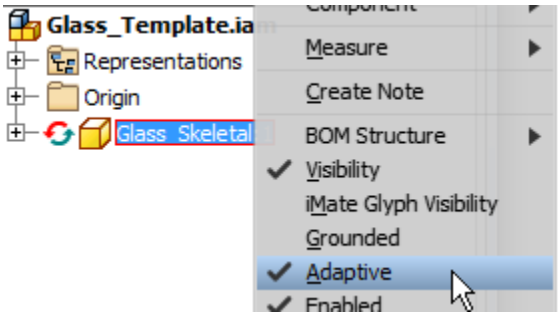


5. Draw geometry that is used to stretch the model in the skeletal part. Keep the input geometry in a single feature and visible. Any additional sketches and geometry should be placed in separate features and the visibility turned off.

6. Create user parameters to link to other components.

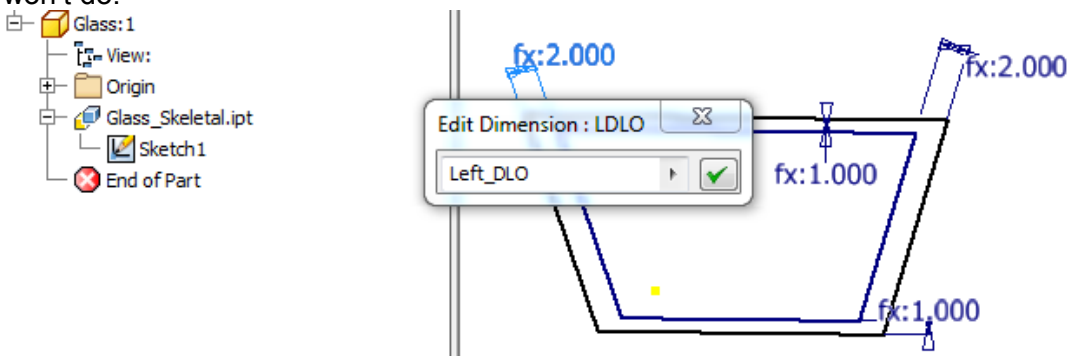


7. Return to top level assembly. Right-click on the skeletal part and select Adaptive. Save the skeletal file.

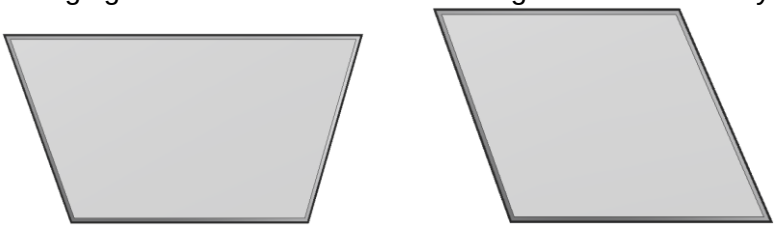


8. Create a part file for a new component.

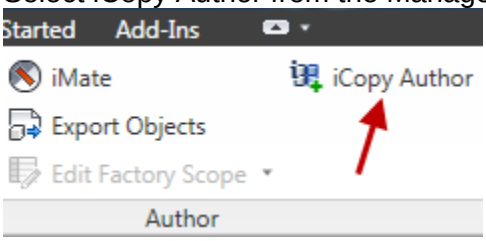
9. Either derive or project the skeletal file into the new component. Use the linked geometry to start modeling the new component. Make links to skeletal parameters when geometry won't do.



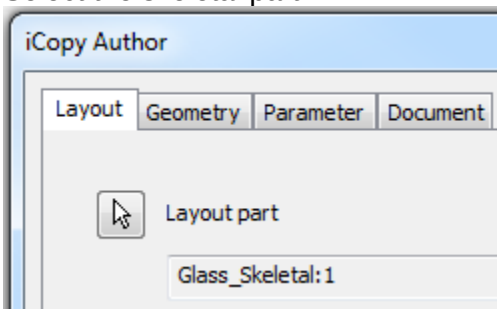
10. Create parts and subassemblies for the rest of the assembly. Link parts and assemblies to the skeletal model as needed. Constrain the parts to the assembly. Test the model by changing the skeletal model and viewing how the assembly changes.



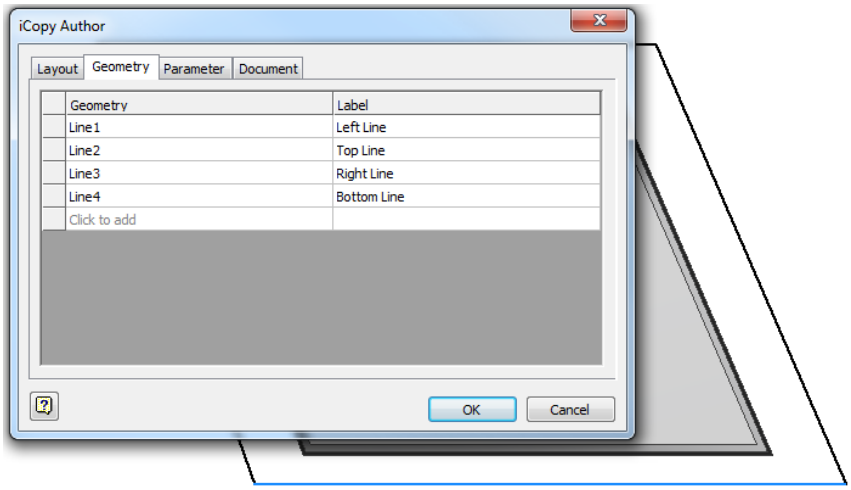
11. Select iCopy Author from the Manage ribbon tab.



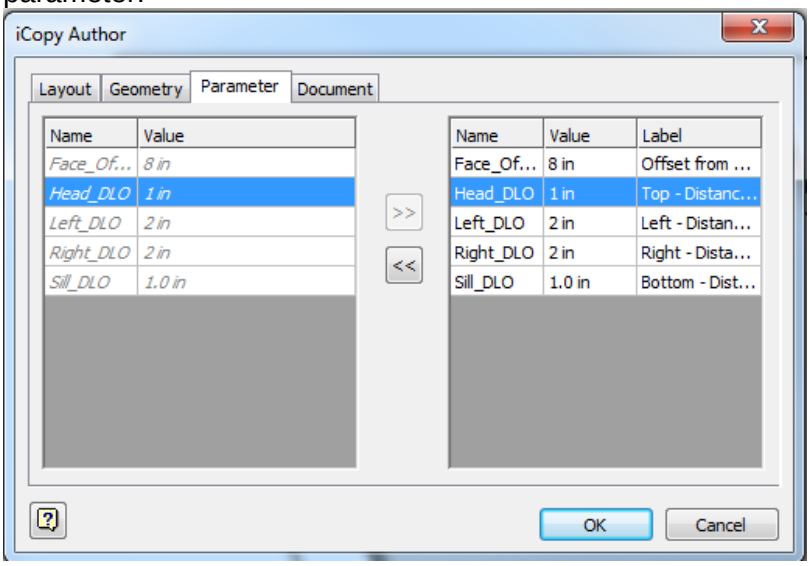
12. Select the skeletal part.



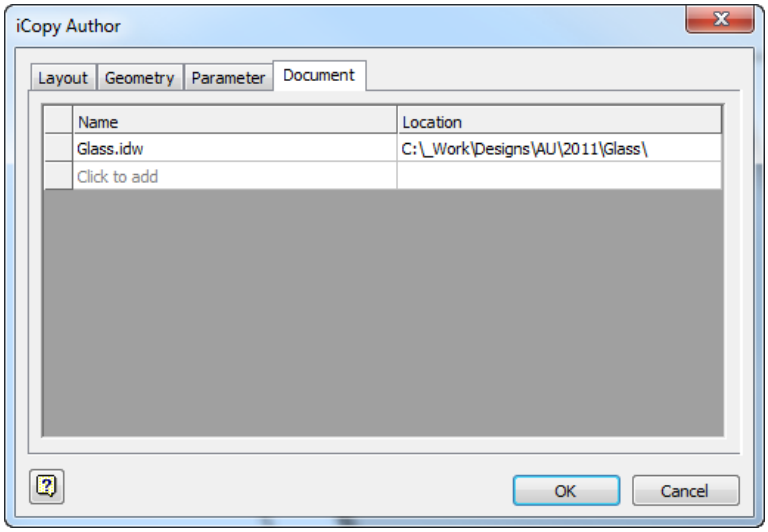
13. On the Geometry tab select geometry from the skeletal part that will stretch the assembly when placed in another assembly. Work geometry and 2D sketch geometry is valid for selection. 3D Sketch geometry isn't. Select the geometry in a logical order for ease of use when placing in assemblies (ex: clockwise selection). Give selection a description. The description will appear for the end user of the iCopy assembly.



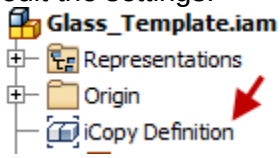
14. On the Parameter tab add of remove parameters. All named parameters from the skeletal part will appear. Only include parameters that you want the end user to change when placing into another assembly. Enter descriptive names in the Label field for each parameter.



15. On the Document tab select drawing of presentation files that will get copied along with the assembly.



16. The iCopy settings are stored in the browser. Right-click on the feature and select Edit to edit the settings.

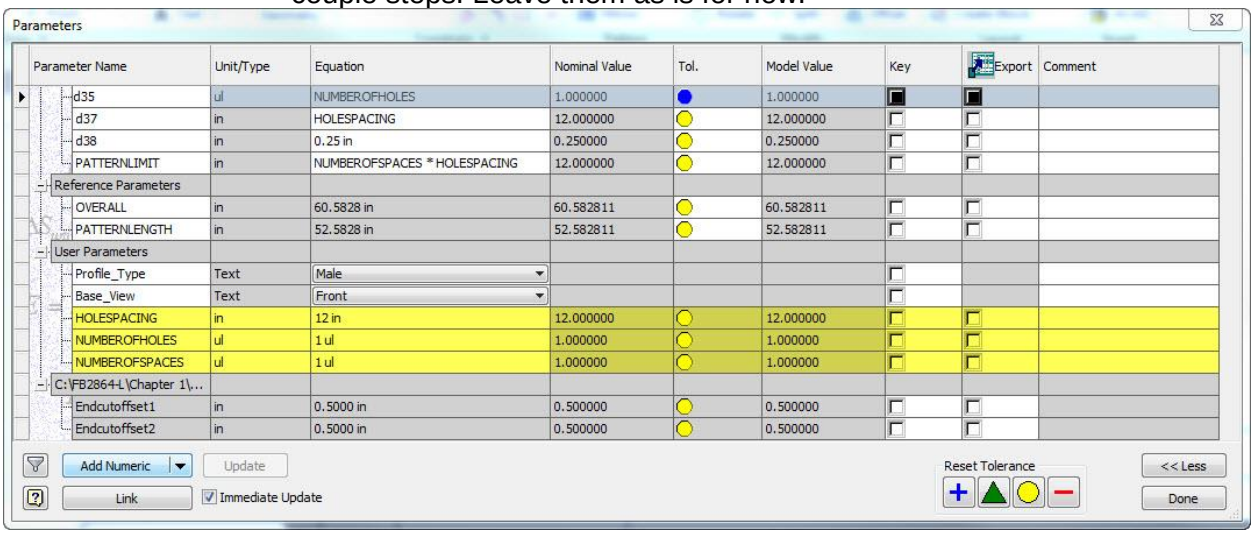


Steps to Create a Patterned Hole Feature

The goal of the following section is to create an adaptive hole pattern feature that automatically updates hole quantity, and # of spaces, each time the icopy frame is stretched/altered. This hole pattern feature will relate to the length of the line of the skeletal frame (refered to as a module), that the part has been extruded along, not the final length of the part. The reason being, we want the hole pattern to align within parts of different lengths that are extruded along the same skeletal line (module) length. By using the length of the module the parts were all extruded along, you can assure that the hole patterns will align between parts even if the part lengths or end cuts of each the parts differ.

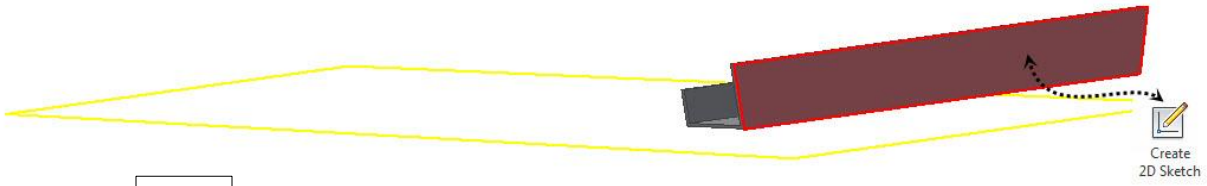
1. Create 3 new Numeric User Parameters as shown below. *Note the unit type*. To change unit type click in the Unit/Type column and select the units you desire.

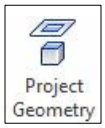
    - For this step “HOLESPACING” will have an *Inches* type designation
    - The “NUMBEROFHOLES” / ”NUMBEROFSPACES” Parameters will have a *Unitless* type designation.
    - In the equation column, set “12 in” as the value for the “*HOLESPACING”* Parameter.
    - We will update the values for the “*NUMBEROFHOLES”* and “*NUMBEROFSPACES”* Parameters as we continue on with the next couple steps. Leave them as is for now.



2. Create a 2d sketch 

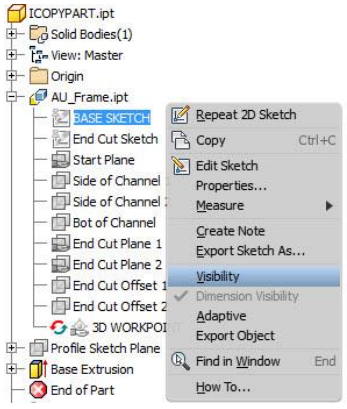
on the face of the extrusion you would like to place the hole pattern along.



3. Project 

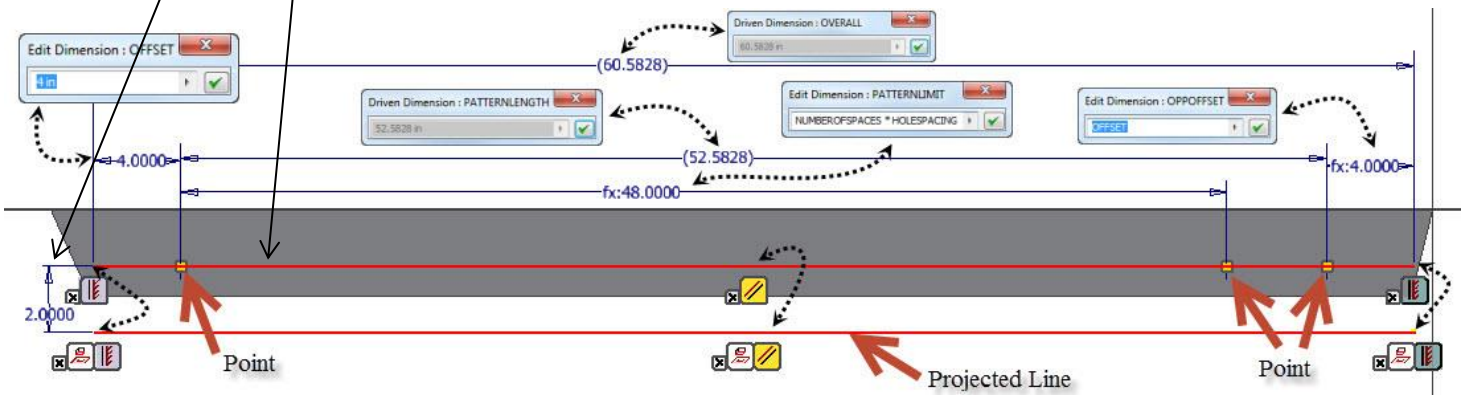
the corresponding skeletal line (The line that you ran the length of your part along) into your sketch. (You will be using this line as the reference length of your hole pattern.

4. Turn off the visibility of the frame sketch in your browser. Do this so you do not accidently reference the wrong geometry when you create your hole pattern sketch in the next step.



This hole pattern sketch is based on a few key factors. Typically in the industry of mechanical engineering the structural guidelines indicate that a part requires a hole pattern the length of the part, and also requires a hole a set dimension from each end of the part. In the example I am showing below, the part requires a hole pattern of 12” on center with a hole 4” in from each end of the part.

5. Add dimensions and constraints to your sketch as shown below.
   a. Start by drawing a line parallel to the Projected Line from step 3.
   b. Add a vertical dimension between the sketch line and projected line to set the location of the holes vertically on your part. *(Tip: By dimensioning from the projected line, you always assure that the line will be there and the sketch will not break. If you dimension to model geometry there is a greater likelihood of the sketch breaking because the model geometry has changed.)*
   c. Add vertical constraints between the end points of the projected line and the sketch line. This is to assure that the sketch line will always stay the same length as the projected skeletal line.
   d. Add 3 points on the sketch line and dimension as shown below. These represent the first hole in the pattern, the last hole in the pattern and the extra hole that will hole the offset at the far end of your part. The last hole. We will pattern between the first and second hole.
   e. Continue to name the Parameters as shown below. *The *Overall* and *PATTERNLENGTH* parameters will be Driven Dimensions.* See below for a description of the purpose of each parameter.



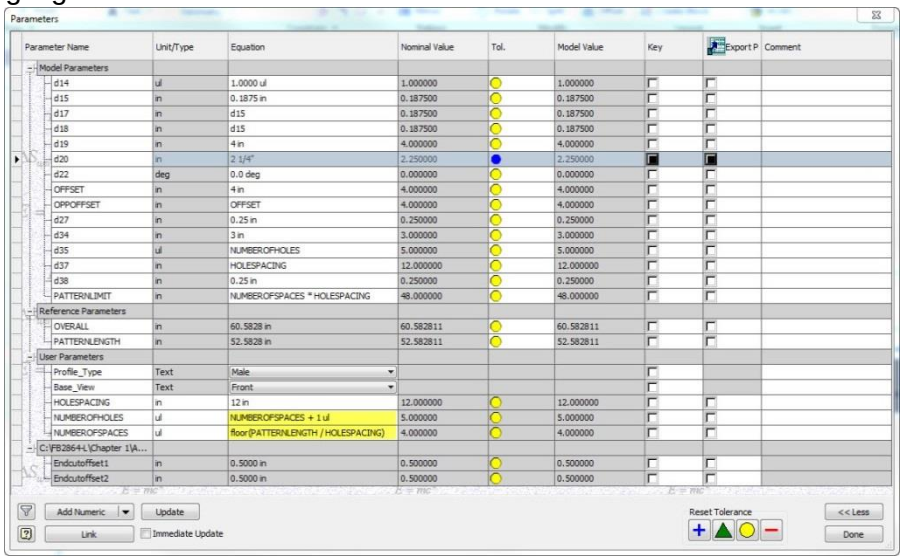
OVERALL= This dimension will be a driven dimension that is representing the length of the skeletal module line. Since we want the length of the module dimension to stretch when the length of the extrusion does, this dimension needs to be driven.

OFFSET = This is the dimension to the first and last point on our sketch line. This will hole the required 4” dimension to the holes at each end of the part.

PATTERNLENGTH = This is the dimension between the first and last hole of the part. The pattern limit is the allowed space that the *12” o.c.* hole pattern can occupy. This is a driven dimension.

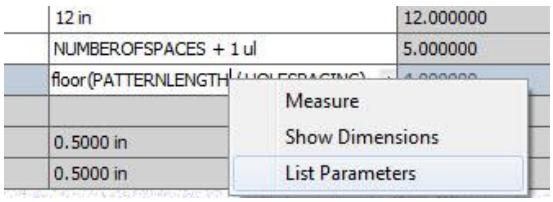
PATTERNLIMIT= This dimension is placed between the first and second points of the part. It is a function parameter will set the actual number of holes in that part that can be placed 12" o.c. Type in NUMBEROFSPACES * HOLESPACING to set this dimension. *(We created these user parameters in step1)* When we update our parameters in the next step the *patternlimit* dimension will adjust to fill the allowed space of the *patternlength*.

6. You will notice that your sketch may not look quite like the one shown above. The reason for that is we have a couple equations to type into the Parameters dialogue. Open the Parameters dialogue box and type into the *Equation* column, the equations as shown highlighted below.



(When you want to add an existing parameter to an equation, right click and select *list parameters* and select the parameter you wish to use from the list. It is important to name the parameters you wish to use in an equation because the list will only show named parameters.)

**tip: An allowed space of 74.625" divided by the hole spacing (12") will result in a number of spaces of 6.22. Since you cannot have fraction of a space, that number needs to be rounded down to the nearest whole number. By typing "floor" in front of an equation, it will round the resultant number down to the closest whole value. (Ex. 6.22 will be rounded to 6) The opposite is true as well. By typing "ceil" in front of the equation the number will round up. (Ex. 6.22 will round up to 7)*

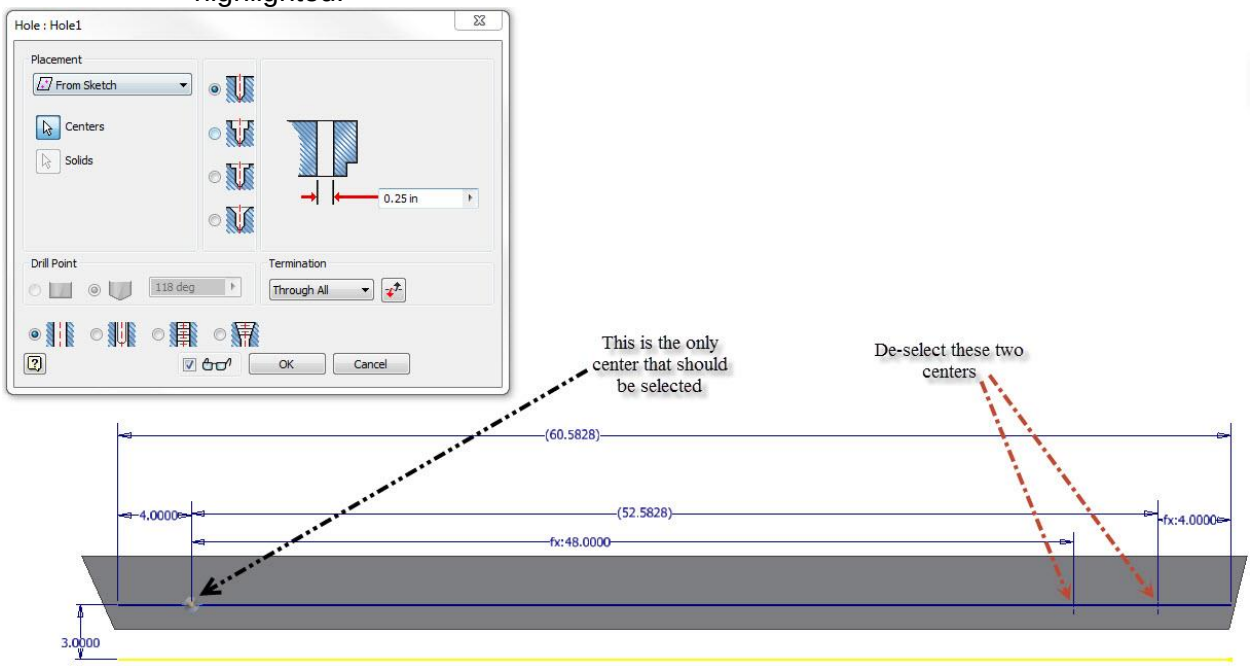


Select done. Finish sketch. When you select the update button you should see your hole pattern sketch stretch to fill the allowed space. The left over dimension between the last hole in the pattern and the last hole in the part should be less than the set hole spacing of 12".

7. Create hole and Pattern features.

   a. Select the *Hole* command. 

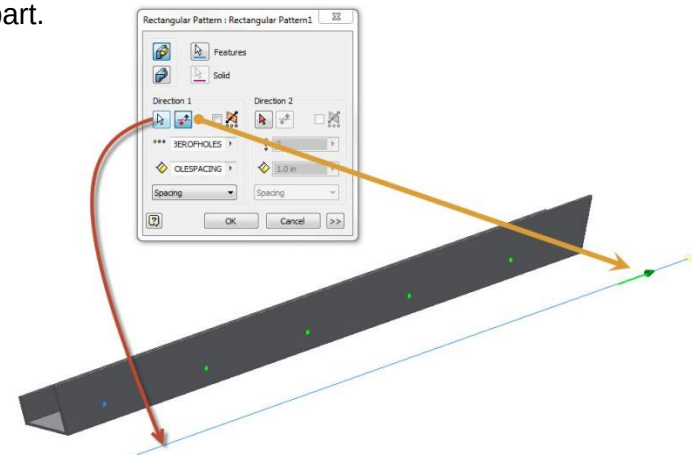
   b. Make sure the only center point selected is the point furthest to the left as shown below, by holding down CTRL and picking the other two points until they are not highlighted.



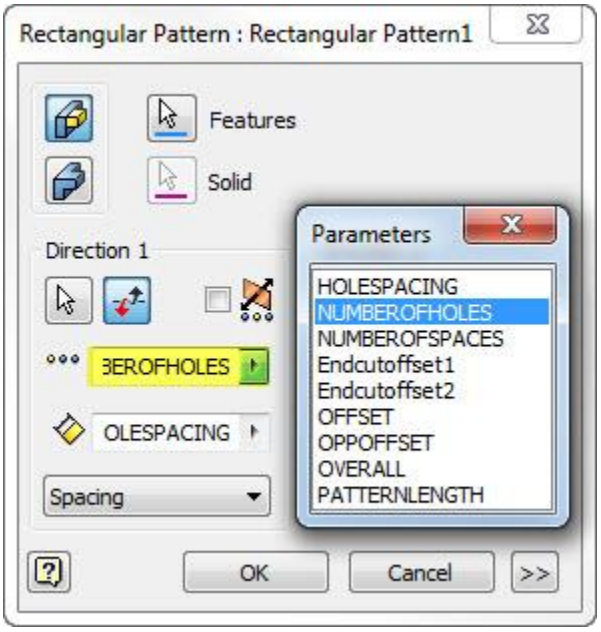
   c. Select **OK**. Your part will have one through hole.

   d. Create a *rectangular pattern*. 

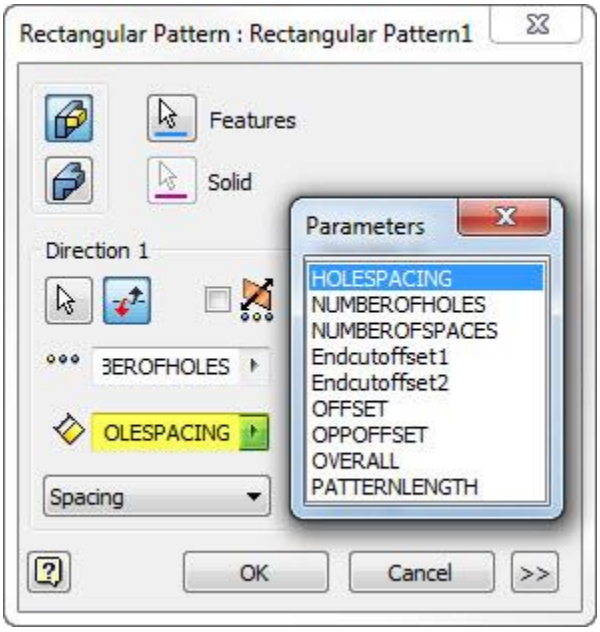
   e. Select the hole feature we created in step 7b. Set the direction by selecting the line from the skeletal frame as shown below so the holes run the length of the part.



f. We will now use a couple of the parameters we previously created in step 7 to set the *Hole Quantity* (fig. F_1)and *Spacing* (fig. F_2) for the pattern. Select ok.



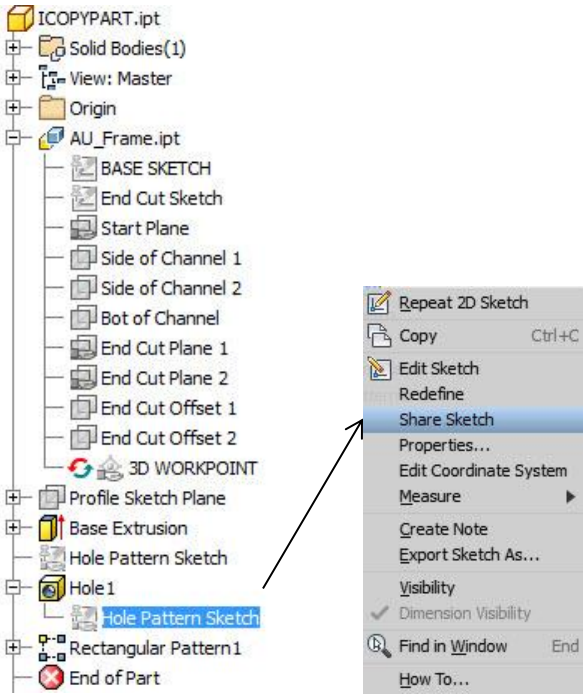
(Fig: F_1)



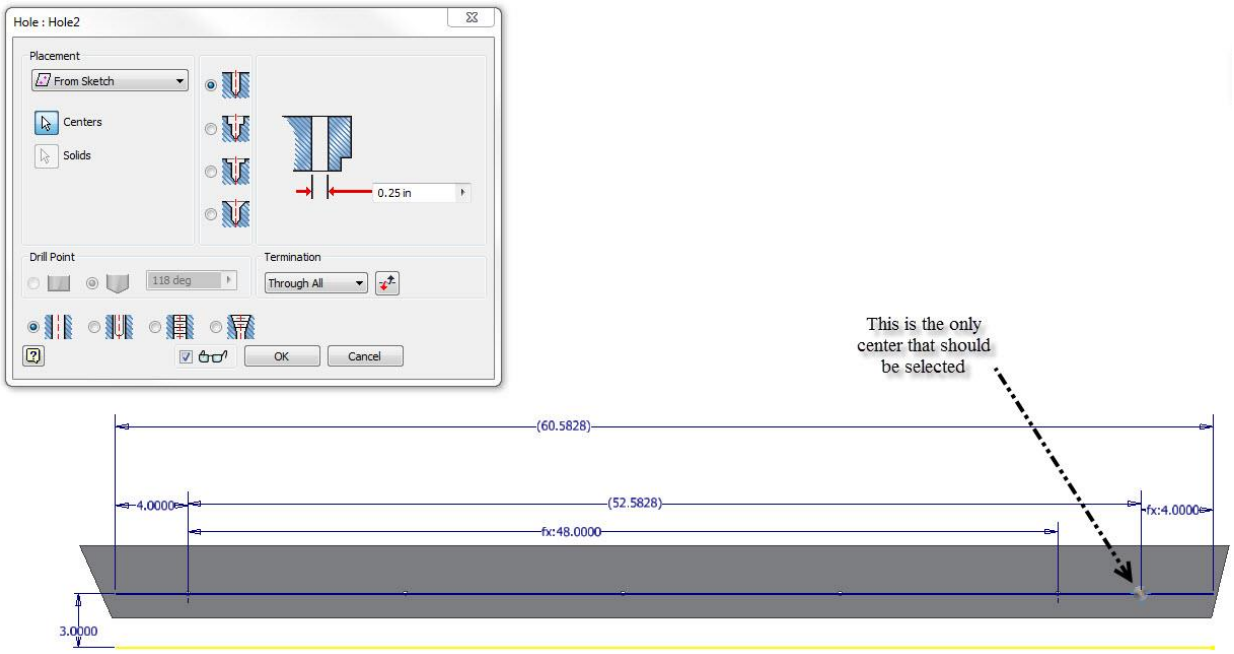
(Fig: F_2)

Now we need to extrude the last hole through the part, since it will be a separate feature.

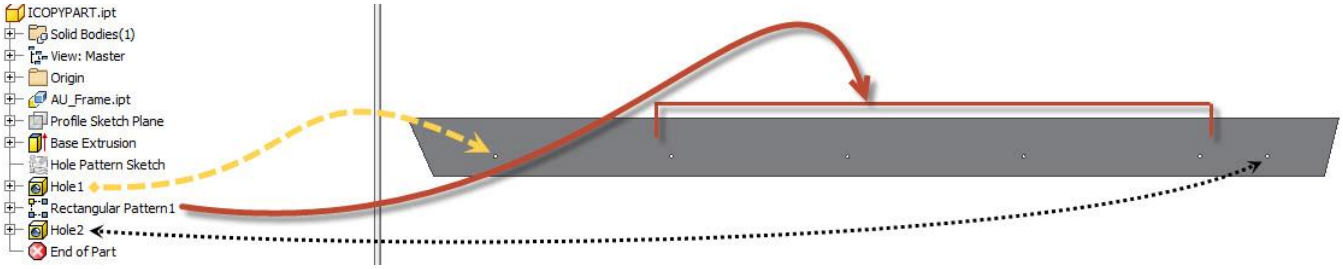
8. Right click on the *"Hole Pattern Sketch" you created* and select ***Share Sketch***.



a. Select the *Hole* command. Make sure the only center point selected is the point furthest to the right as shown below. If other holes are selected, deselect them by holding down CTRL and picking the other two points until they are not highlighted. Select **OK**.

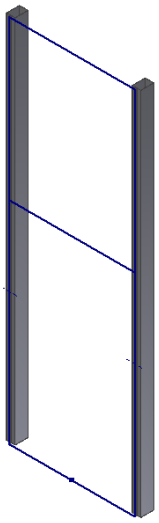


Your part will now have a *Hole* feature, followed by a *Pattern* feature of that hole, and one last H*ole* feature as shown below.



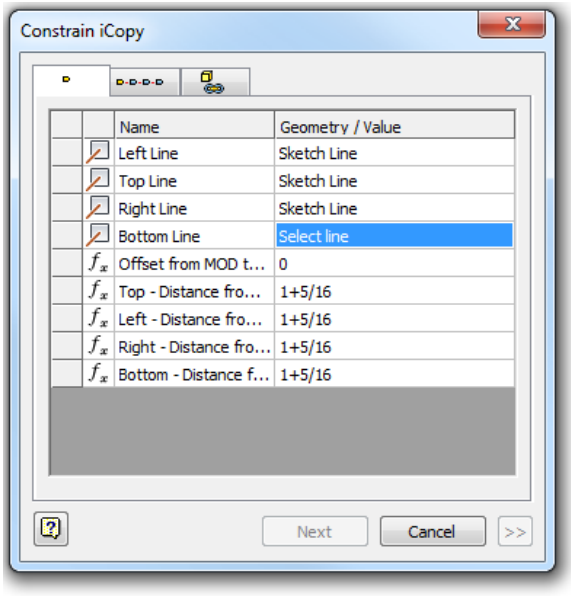
Place an iCopy assembly

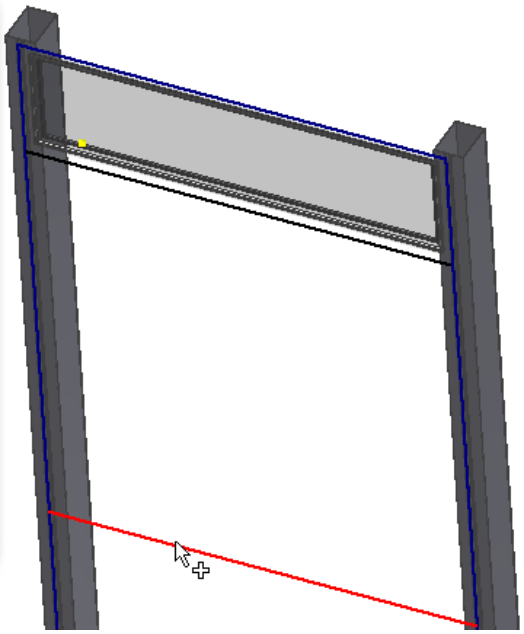
1. Open an assembly that has geometry set up to place the iCopy assembly. The geometry may be a skeletal layout file, or work geometry.



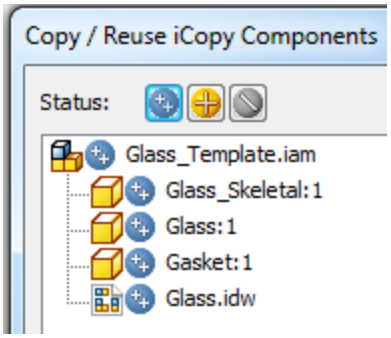
tip: Multiple iCopy templates can be placed within one project file as long as the file names of each placed assembly and its components contain different file names.

2. Select iCopy from the Assemble ribbon tab. Select an assembly that is configured for iCopy. Select Open.
3. Select geometry and enter parameters that are set up in the iCopy assembly. Select Next. If you get an error, cancel out of the dialog and look at the iCopy assembly for problems.

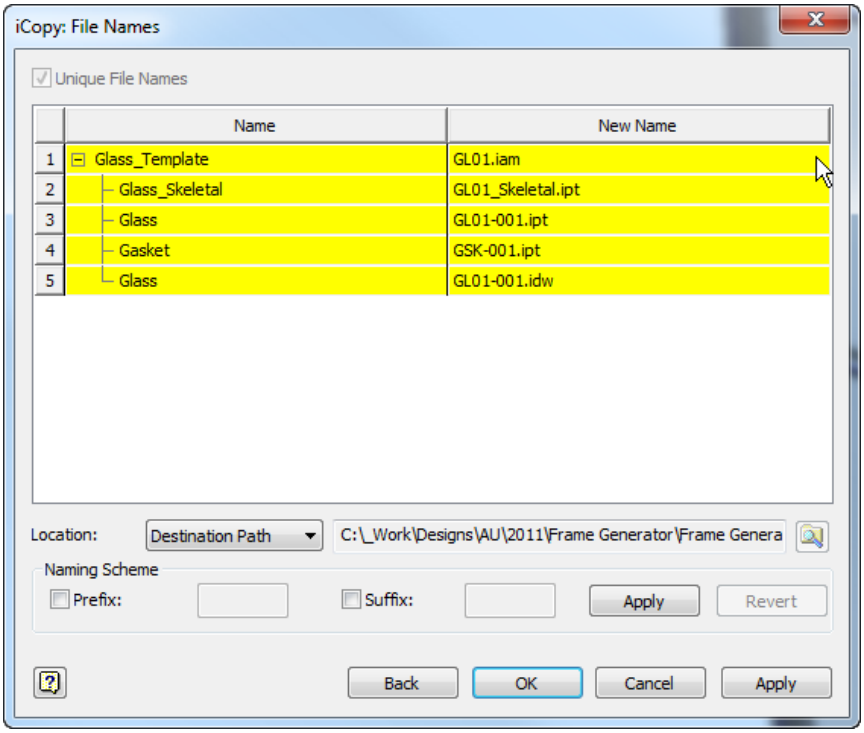




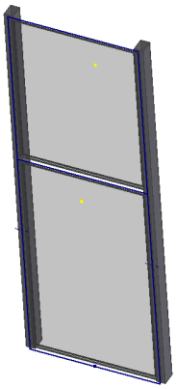
4. Select files to copy, reuse, or exclude. Notice that linked drawings and presentation files will also copy. Select Next.



5. Enter new file names and locations for components that are copied. Select OK.



6. The assembly will appear. The iCopy is now nested in its own group in the browser.



CONGRATULATIONS, YOU ARE FINISHED!!!

Tricks and Tips for using iCopy:

- iCopy templates can have many parts and/or assemblies nested within. The more parts within an icopy template the longer it will take to place because inventor regenerates the icopy model after every pick. Extremely large models place slow. Small models place quickly. Find a happy medium when developing a template.

- The first icopy template you make will take a long time. What took me 75 minutes to demonstrate in this class took a week to develop during the first run through.

- You can utilize ilogic and frame generator in icopy templates to create smarter faster models.

- When constraining in icopy models use the best practice: Constrain to Origin axis/planes. Surface geometry is unstable, if a surface changes that a constraint is set to, the constraint will break. If instead, the constraint was placed to an origin plane, then the constraint is constant because an origin plane will never change or disappear

- Once you create one template, it can become a template for a new unit. If you have one extrusion that will change in the new template you only need to make that one change instead of building a new icopy template from scratch.

- In turn, make sure your template model is perfect. If there is one mistake on a single icopy template, the issue can compound hundreds of times and you will have to fix it in every instance of that template.

- Test your frame model before adding parts to it… If your frame model does not constrain into a general assembly on its own, it certainly will not work when parts are added to it.

- When developing a frame sketch use as few constraints as possible. An unintentional perpendicular constraint can be disastrous when trying to constrain a polygonal shape into a general assembly. To avoid placing unwanted constraints while sketching, hold down CTRL while picking sketch points.

- You can use an excel spreadsheet to fill out parameters and link values from revit models, but that's for another class…

…. Perhaps next year