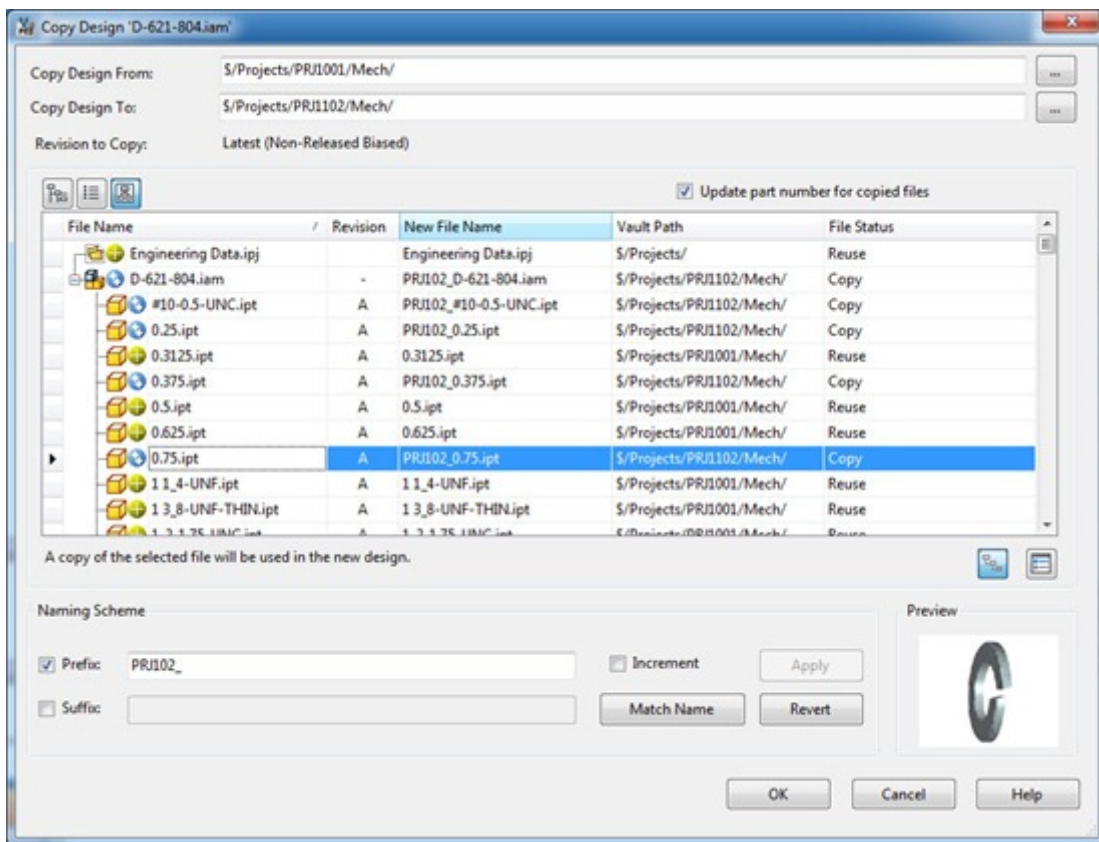


Copying Assemblies with the Vault ‘Copy Design’ Tool

Copying Assembly files with the Vault is very simple, apparently —I don’t know because I don’t have the Vault...

Edit: I do now! We’ve recently upgraded to Vault workgroup 2014, and I can say – it’s awesome!

Vault 2015 has a new and improved Copy Design tool, and you can [read about it here](#):



This is the only downside of this technique. If you are lucky enough to be using the Vault to manage your Data, then I guess you can stop reading now!

Thanks very much to [Steve Bedder](#) of Autodesk for supplying this Vault Screenshot.

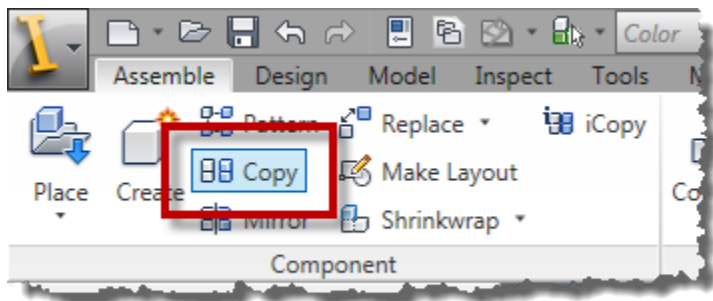
Copying Assemblies with Inventor’s ‘Copy Component’ tool

To copy an Assembly with the 'Copy Component' tool, you first need to open the Assembly you wish to work with. To copy the entire Assembly navigate to the:

Assemble Tab > Component Panel > Copy

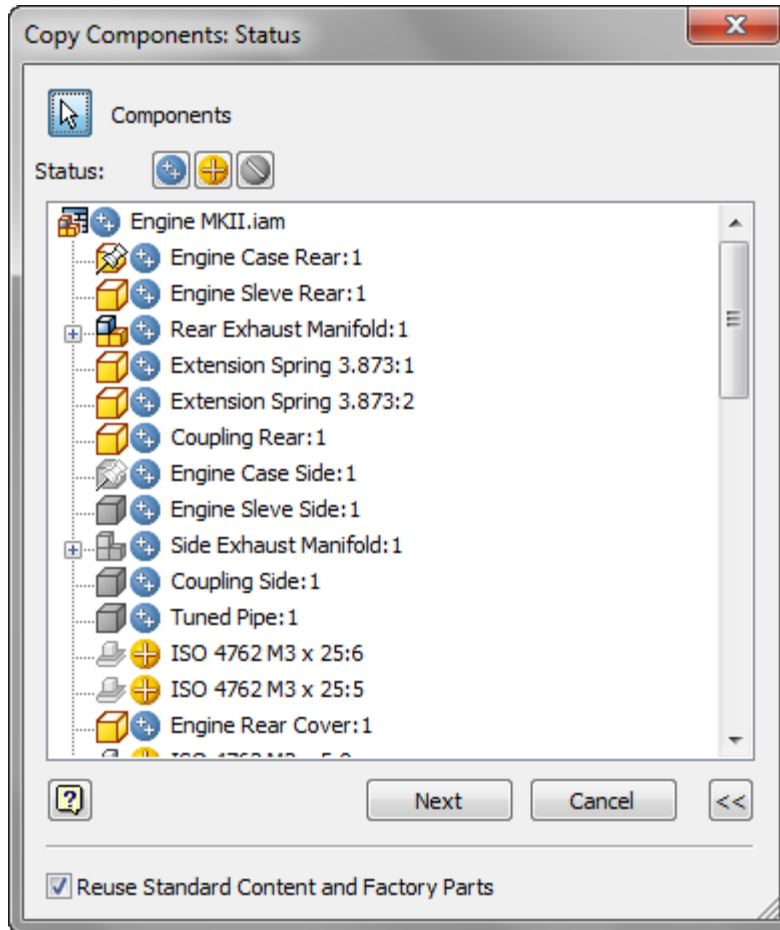
or right click on your selection of Components and choose 'Copy'.

Tip: In Inventor 2015 the Copy tool has been moved to a drop down panel



Copy Components

You will be taken to the 'Copy Components: Status' Dialogue box, which allows you to chose whether to copy or reuse the components that you have selected.



You can select and deselect components from the Assembly browser to add them to the list. Hit 'Next' when you are happy with your selection.

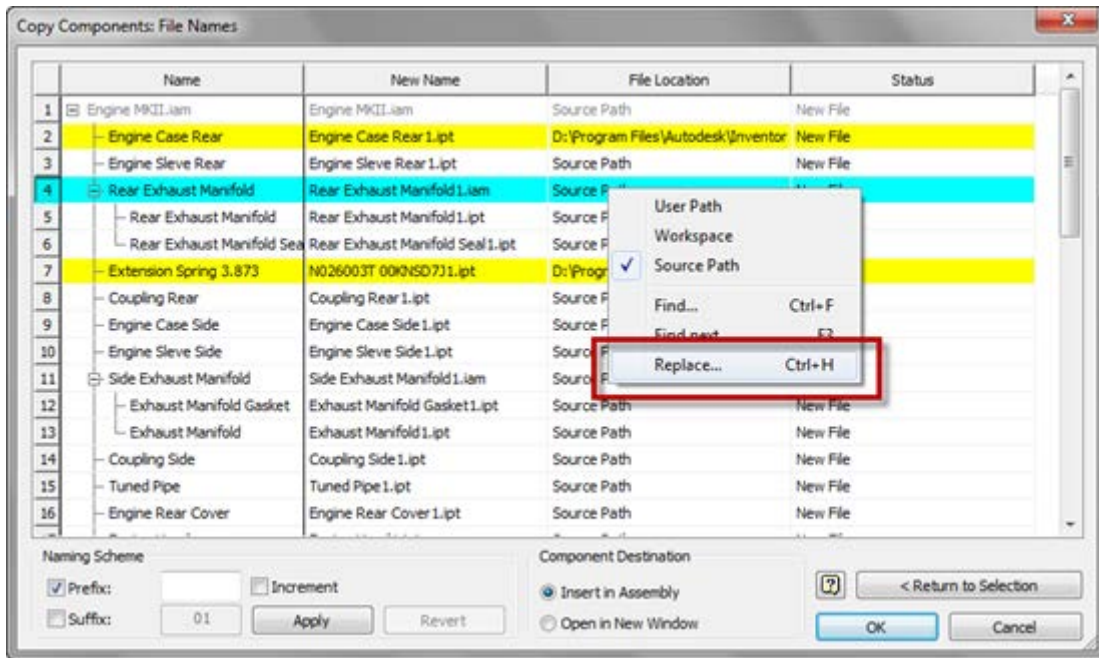
Edit Filenames and paths

The next Dialogue is the 'Copy Components: File Names' Box. This dialogue gives you a lot of options to chose where you'd like to save your copied components and what you'd like them to be called.

You can use the 'Prefix' and 'Suffix' boxes to Automatically revise your file names, or you can edit each one by hand.

Right click over a field in the 'File Location' column to set a new path.

- **'Source Path'** will save the new file in the same location as the old file (probably not a good idea!).
- **'Workspace'** will save the new file in the root folder of your workspace (Again, probably not what you want!).
- **'User Path'** allows you to set your own path. Copy and paste is your friend here.



Tip: Right click over the 'New Name' or 'File Location' fields and choose 'Find and replace' to quickly edit repeated text strings.

Finally, you have the choice whether you want to place your copied Assembly into your current assembly – or create a new file (Probably create a new file).

Hit OK, and Inventor will create a new copy of your Assembly, including copies of all the assembly's components, and handle all the Internal paths. Very neat.

Conclusions

Inventor's Copy Assembly tool is pretty easy to use and gives you a lot of useful options to re-name and path your components. Unfortunately the Copy Component tool's major downfall is that *it doesn't recognise derived references in part files.*

So if you are using [Skeletal Modelling](#) or the [Multi-body modelling technique](#), this technique is out. You may want to consider using the Design Assistant instead.

Copying an Assembly with the Design Assistant.

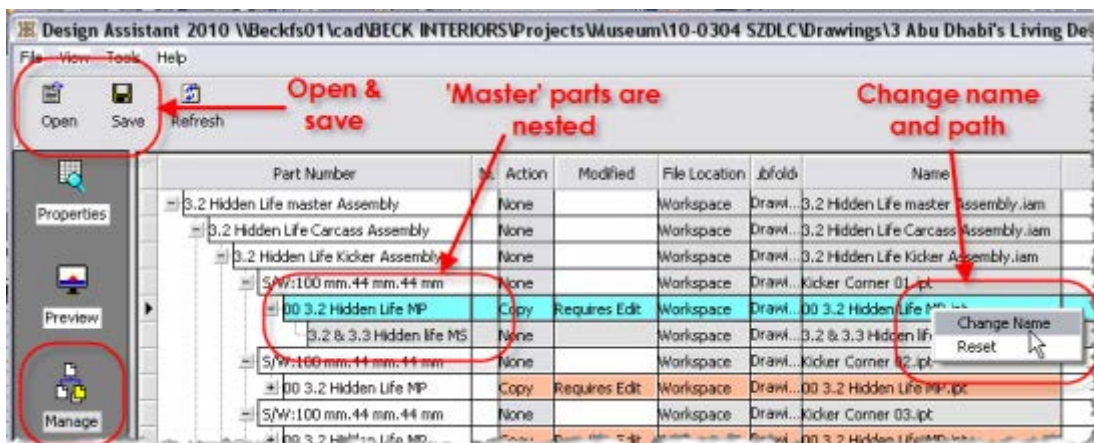
Design Assistant is a standalone program that can be opened from inside Inventor or started up separately. The Design Assistant uses the Inventor Apprentice COM server to edit Inventor files without having to have Inventor running.

While copying Assemblies I recommend that you close Inventor down and open Design Assistant separately.

Look for Design Assistant under:

Windows > Programs > Autodesk > Autodesk Inventor > Design Assistant

Note: When Design Assistant starts up, it will be using the last project that you had open in Inventor. Check that this is correct before you start!



Set your Project file.

Before you start, make sure that you have the correct Project file active. Go to:

File > Projects...

And make sure that the project is active that contains the Assembly you wish to copy.

Open an Assembly to Copy

Click on the 'Manage' button on the left, and then click on the 'Open' button at the top to open your assembly.

Note: Notice that the 'Tree' layout shows the relationships between the components – including the references of derived parts.

Set an Action

Right click over the 'Action' column and choose 'Copy' to copy a component. You can pick multiple components from the left hand column before you pick an Action.

The note in the 'Modified' column will change to 'Requires Edit' and the Cells will change to an Orange colour to remind you which ones need editing.

Edit the Path and File name

Double click in the 'Name' cell of the component you wish to copy, browse to your new path and rename the part.

You will need to do this for each and every component that is in your Assembly file (Excluding Library parts).

Notice how references of derived parts and components used in more than one Assembly are changed everywhere they are used, all at once.

Save your changes

When you have edited the values of every part that you want to copy, check it! and then check it again! You really don't want to miss out a component at this stage, it could get really confusing....

When you are happy that you've edited all the components that you need to copy, and that the new names and file paths are correct, hit the 'Save' button at the top.

Note: Until you hit the 'Save' Button, none of the changes will be written out to disk.

Inventor will create an entirely new Assembly, and will create all the new components on disk while creating all the new file paths (hyperlinks) inside the Assembly file.

If you browse to the folder where you are saving the copy of your Assembly, you will see all the little parts being created.

After the new Assembly has been created, Design Assistant will automatically load the new version of the Assembly.

With a really big Assembly this process can take a little while – Plenty of time for a Cuppa.

Conclusion

Creating all those new component names and paths for a large Assembly can be really tedious, however If you need to copy an Assembly that has been created using the [Skeletal Modelling technique](#) or [Master part Modelling technique](#), this might be the only way of copying your Assembly (Not counting the Vault).

Tip: *use 'Pack and Go' to copy the entire Assembly out to a 'Work in progress' project location and then use the Design Assistant to rename, rather than copy your components.*

This saves time and effort, because you don't have to browse to a new file location each time. When you are done, you can copy the entire set of files back into your Project location.

Copying an Assembly with the iLogic Design Copy Tool

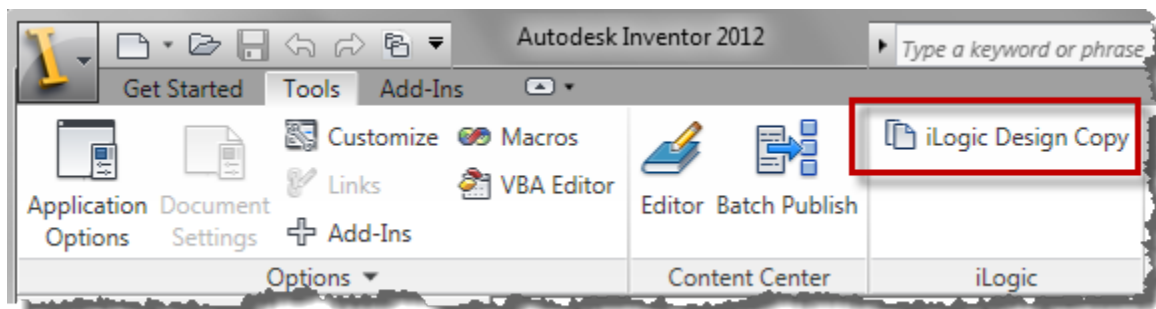
The iLogic Design Copy tool was introduced in the Inventor 2011 Subscription Bonus pack 1, and from Inventor 2012 is in the box.

The iLogic Design copy tool was introduced to help copy Parts and Assemblies that contain – or reference iLogic’s .ILOGICVB ‘Rules’, but you don’t have to be copying an iLogic Part or Assembly to take advantage of the tool.

Starting up iLogic Design Copy

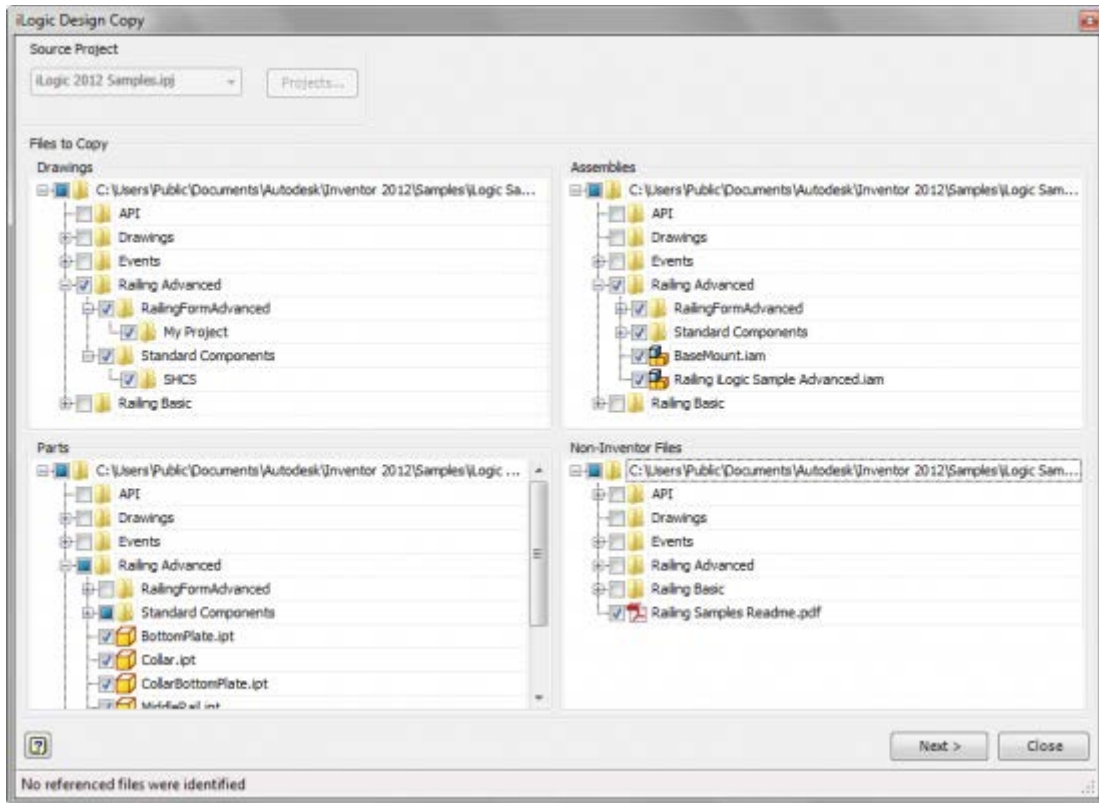
The iLogic Design Copy tool is found under the ‘Tools’ tab > ‘iLogic’ panel

Note: The iLogic Design Copy tool is only visible when no other documents are open!



Selecting Files to copy

The iLogic Copy Design tool’s dialogue takes a very different approach to the Copy Assembly tool or the Design Assistant.



A series of ‘Tree’ browsers allow to you to pick in ever finer detail which files you wish to copy.

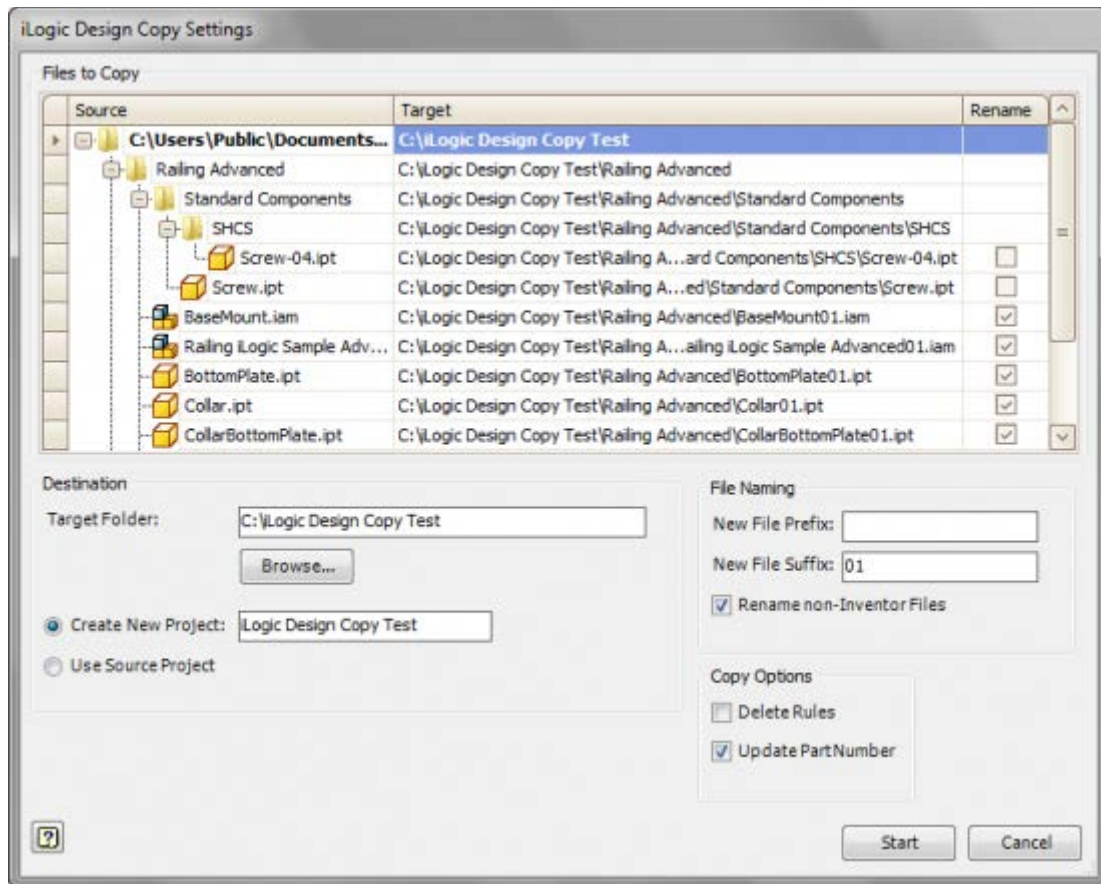
- Use the **Files to copy** tree to pick the folders where the parts are stored (Remember to include Library folders in the search).
- Use the **Assemblies** Tree to pick the Assembly you wish to copy
- The **Parts** Tree will be automatically filled out, but you can add or remove individual files at your whim.
- Use the **Non-Inventor** files tree to add any other relevant files to the selection set.

Hit Next when you’re happy with what you’ve selected.

Renaming the Files

The iLogic Design Copy tool allows you to add your own suffix or prefix onto the component names.

You can’t change the file names completely, but you can choose not to change a file name by unchecking the box in the ‘Rename’ column.



Tip: You can see an excellent Video of [the iLogic Design Copy Tool in Action](#) on the Inventor Wiki Site.

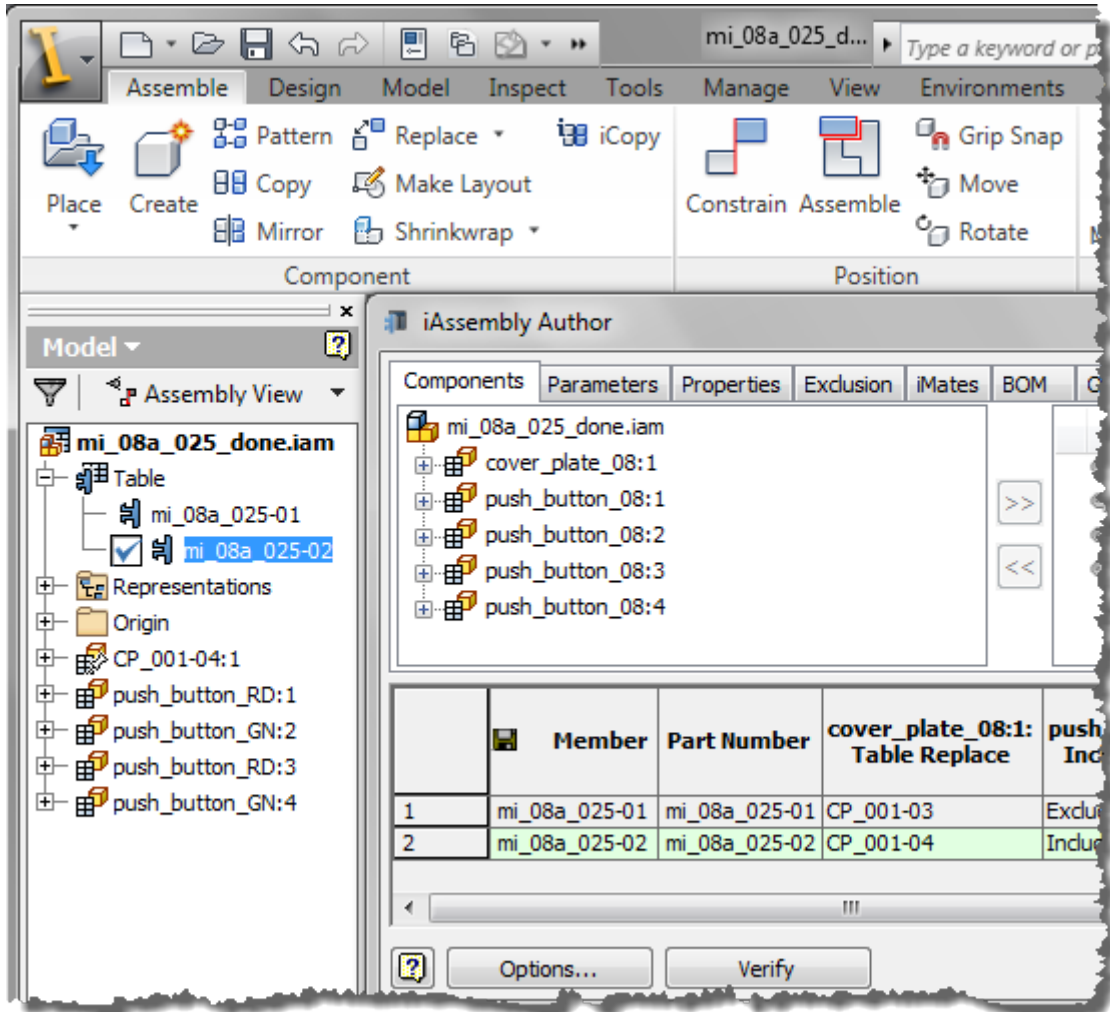
Conclusion

The iLogic Design Copy Tool is a well thought out departure from the previous examples. The fact that the component names can't be changed in their entirety feels limiting, but I guess that if you know about this and plan for it in advance it shouldn't be a problem.

Thanks very much to [Mark Flayler of IMAGINiT Manufacturing Solutions](#) for bringing this new tool to my attention.

Using an Inventor iAssembly to create multiple designs

Much like iParts, iAssemblies are table driven configurable Assemblies that are great for when you have a limited number of variations to a design – and you know what configurations you are going to need in advance.



You can use an iAssembly in one of two three ways;

You could use the iAssembly on its own, as a 'Drawing Factory'. When you create a drawing from an iAssembly you can chose which of the iAssembly configurations you want to document in the drawing. You don't need to create a new copy of the iAssambly (and it's components) each time to document the different versions of the design.

You can also use an iAssembly as a sub assembly within a Master Assembly. The iAssembly will reference the correct configuration of the parts it contains, so you don't need to create new copies of all it's components each time.

Tip: iAssemblies that contain iParts which have Custom columns can have new members added to the iAssembly on the fly as the iAssembly is inserted into a Master assembly. You could use this technique to generate entire assemblies!

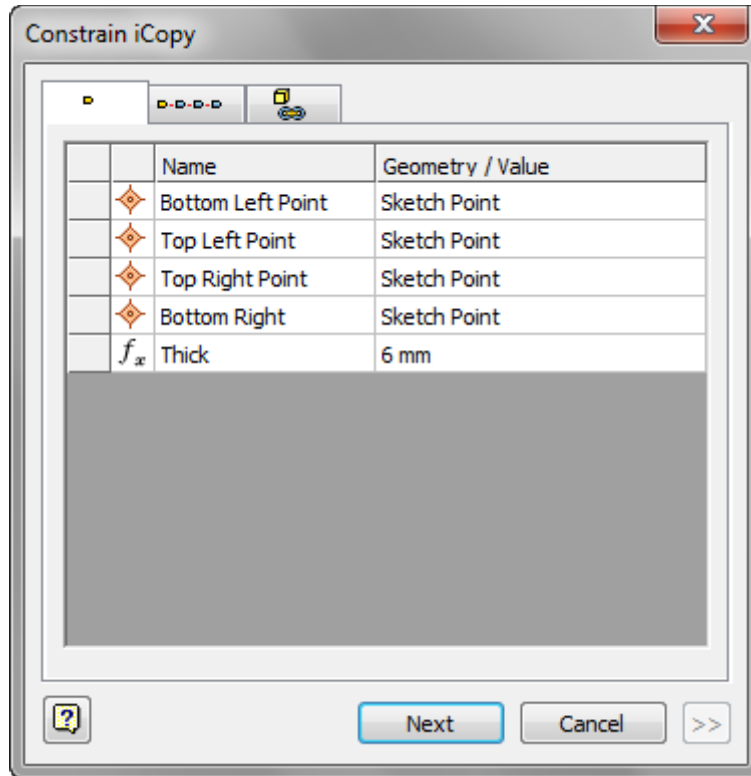
If you do need to copy an iAssembly (Perhaps to create an archive version before you make changes to the iAssembly table, or the components that belong to the iAssembly) you could use Pack and Go. Other wise you will need to use 'Copy Assembly' or the Design Assistant, as described above.

Conclusion

If you know in advance that you will need to create a limited number of variations of a design (Think Kitchen units), it is worth investing the time in creating iParts and iAssemblies. This will save you time and effort down the line.

Using iCopy to create copies of Assemblies

[iCopy](#) is a new tool that was added to Inventor as an [Autodesk Labs](#) plugin for Inventor 2010. From Inventor 2011, iCopy is included in Inventor.



iCopy is slightly different from an iAssembly, in that the iCopy factory assembly uses the [skeletal modelling](#) technique to create infinite sizes of a design.

iCopy assemblies are used in conjunction with a 'Skelton' part (much like the Frame Generator). iCopy handles the creation and naming of the new assembly file and all its component parts.

iCopy can also be used to automate adding patterns of Assemblies, like rungs on a ladder or curtain walling panels.

Conclusion

Like iAssemblies, iCopy is great if you know in advance that you are going to need a number of variations of a design. iCopy is much more flexible than iAssemblies and can be used to quickly create infinite sizes of a design (Think Doors and Windows).

Note: There is no reason why iAssemblies and iCopy couldn't be used in conjunction with each other (although the thought is pretty mind boggling!).

Tip: Check out this [iCopy Tutorial](#) for more details on how to use this tool.

Copying Assemblies with the ‘Copy Design’ tool from the SDK

The ‘Copy Design’ tool is part of the Autodesk Inventor Software Development Kit (SDK). This tool has actually been developed as an example for people wanting to learn how to use the Inventor Application Programming Interface (API).

Strictly speaking, Copy Design is not intended for the general user, and you will have to load it yourself.

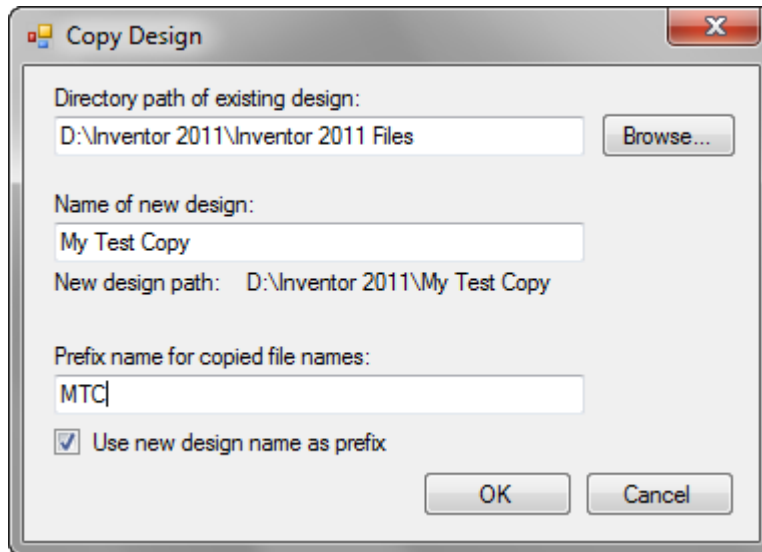
The Copy Design tool uses the Inventor Apprentice COM server (Just like the Design Assistant), so you don’t have to have Inventor open to run it (in fact it might be safer if you don’t have Inventor running at all).

To load the SDK, close down all your programs and navigate to:

C:\Users\Public\Documents\Autodesk\Inventor 2012\SDK

Note: This is the location for Autodesk 2012 running under Windows 7. The SDK will always be found in the installation location for the current platform and version of Inventor.

Double click on the User.MSI file to run the installer. The CopyDesign.EXE will be extracted. Double click on the EXE file to run.



The 'Copy Design' tool could better be described as the 'Copy drawing' tool in that you need to pick a directory file that has a drawing file in it for the Copy Design tool to work.

Note: The original drawing file must be an IDW. DWG's are not recognized by the Copy Design tool.

The Copy Design tool will copy any IDW's it finds in the 'existing design' folder into a new folder of your choice, and it will prefix the components names and re-map the hyperlinks if you ask it nicely.

Conclusion

Because the Copy Design tool runs outside of the Inventor process it works extremely quickly, however the component renaming abilities are extremely limited.

Having to insert an assembly into an IDW file for the Copy Design tool to recognize is no great hardship, but if you use DWG files this tool is no help at all.