Drive Autodesk® Inventor® with the Top Down: Alternative Assembly Modeling Techniques

Paul Munford - Halstock Cabinet makers

MA2604 Top down, bottom up, skeletal modeling, layout models, multi-body master parts: If you are having difficulty turning your part models into assembly models, do not despair! There are many alternative assembly techniques that can help you produce great results.

This class is for drafters who have made their way through a basic Autodesk Inventor 3D CAD software training course, but who are having difficulty using Inventor on their bespoke, one-off projects. This class focuses on "alternative" assembly modeling techniques in Inventor that can help you build unique models quickly and simply, and without the nightmare of assembly constraints or circular adaptive references.

This class will benefit drafters from all engineering and manufacturing industries. Attendees should have working knowledge of Autodesk Inventor. If you produce bespoke, unique, one-off digital prototypes, this class is for you.

Learning Objectives

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

- Identify the differences between the bottom up, top down, and derived geometry Inventor assembly techniques.
- Communicate the benefits of using these assembly techniques over the "standard" constraint-based assembly technique.
- Choose the appropriate technique (or combination of techniques) to use on your project.
- Use these "alternative" assembly techniques to build constraint free assembly models

About the Speaker

Laugher, Dreamer, Bon Viveur - Paul Munford is a CAD geek, and CAD Manager at Halstock cabinet makers in the UK.

Paul is a member of the British standards Institute committee for BS8888 (The British standard for technical drawing) and specializes in using AutoCAD and Autodesk Inventor for creating Cabinetry, millwork & Joinery 'Shop' drawings.

In his spare time Paul nurtures the 'CadSetterOut.com' Blog, and has written for AUGI world magazine, Develop3D and Shaan Hurley's 'Between the lines' Blog.

Paul was a speaker at Autodesk University for the first time in 2012 and says it's the most fun anyone can have with 150 other people in the room.

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Introduction

Top Down, Bottom Up, Skeletal Modelling, Multi-Body Master Part – If you are having difficulty turning your part models into assembly models do not despair! There are many alternative assembly techniques that could help you to produce great results.

This class is for drafters who have made their way through a basic Inventor training course, but who are having difficulty using Inventor to create large assemblies, particularly on their bespoke, one-off projects.

This class focuses on Inventor's 'alternative' assembly modelling techniques that can help you build unique models, quickly and simply without the nightmare of Assembly constraints and circular Adaptive references.

This class is for Drafters from all engineering and manufacturing industries that produce bespoke, unique, one off digital prototypes with Autodesk Inventor.

Why are you looking for an alternative?

If you've been learning Inventor at your local college, Autodesk VAR or in your workplace; I congratulate you. I hope that you find prototyping your designs in Inventor as satisfying and rewarding as I have.

However, all too often I have found people come away from their training with only a basic knowledge of part modeling and assembly building. As soon as you get asked to work on a real project you quickly begin to realize how limited your training has been.

The best class I ever took for Autodesk Inventor was in skeletal modeling, and this class was the turning moment that took my use of Inventor from a frustrating struggle to a design powerhouse.

In fact, my whole use and strategy for modeling in Autodesk Inventor has been built around this technique. In this presentation I will be sharing with you some of my experiences and findings that I hope make your own struggles with Inventor melt away.

Design Intent

If you are reading his then I am assuming that you want the same thing as me – reliable, predictable assemblies that can be changed and updated effortlessly.

Defining how you want your model to change is often termed 'Design intent'. You need to work with Inventor in a structured manner in order to get a predictable result.

You are designing how you are going to build your model, as much as you are designing what you are modeling.

Top down, Bottom up and In-place modeling

I hope that you are all confident in creating parametric parts with Autodesk Inventor. To make sure that we all have the same understanding of assembly building techniques, I'll start by defining the three main methods for building assemblies.

- Bottom Up
- In Place
- Top Down

Bottom up– The traditional method

This is probably the method you learned in your Inventor basics class. Each part is created individually. Each part is added to an assembly file and constrained into position.

Any relationship between parts may be handled outside of Inventor, by a separate ERP system for example. This technique is suited to companies that manufacture multiples of the same parts, which are assembled in different configurations.

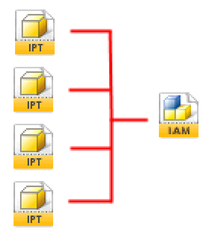


Fig. 1: Bottom up

In place – Adaptive components

In this method, you start with an assembly file and create each part in the context of the assembly file. This technique works particularly well with Inventors Adaptivity tools to build parametric assemblies with relationships between the parts built right in.

The down side is that adaptive parts can be tricky, and you have to be really careful not to build in any circular references. In practice, adaptivity is limited to simple assemblies.

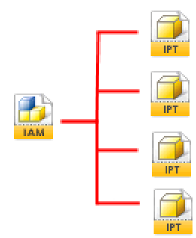


Fig. 2: In Place

Top down – Derived geometry

In this method, you start with a single part file. In this part file you create all the parameters, work features, geometry, features or solids you will need to model your assembly. You then use Inventors 'derive' tools to push or pull the geometry out into individual part files.

When you recombine these parts into an assembly file, you will naturally find that they fit perfectly with no constraints required. The relationships between the parts are geometric. They are handled by the master part file.

Any changes to the master part file are propagated to the individual part files. With no constraints to fail, this can be a powerful technique for modeling large assemblies, and is particularly suited to bespoke work and working in teams.

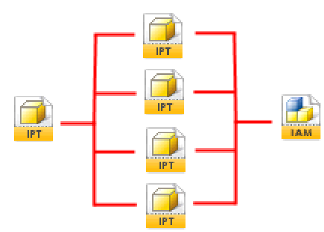


Fig. 3: Top down

Mix and match

In practice, you will use a combination of techniques within the same design. Library parts will be modeled individually and constrained in place, while unique parts will be derived from the master model.

Even adaptivity can be used effectively when goal finding, or working with neutral format models from third parties.

The assembly modeling technique suits the manufacturing process.

Top down: Skeletal modeling

'Skeletal' modeling is the precursor of the more recent 'Multi body Design Part' technique that we will be looking at later in this handout.

Using skeletal modeling, you create all the parameters, work features and geometry you will need in one part file, and then use Inventor's 'Derive' tools to pull this geometry into each part file of your assembly. Finally you add 3D features at the part level.

Since the release of Inventor 2010, this technique has been given a massive boost with the additional functions built around sketch blocks and the 'Make components' tool.

Sketch block layouts

Skeletal modeling is an excellent technique to use for Kinematic (moving) assemblies. 2D Sketch blocks can be assembled with geometric constraints to prove out the function of a design before any 3D modeling happens.

Using the 'Make components' tool, the sketch blocks can be pushed out into part files; Inventor will automatically apply assembly constraints based on the geometric constraints from the skeletal model.

Once you have your design working, you can 'flesh out' your skeletal model, by creating 3D features in the individual part files.

Workflow

- Plan your assembly.
- Add your known parameters.
- Create work planes and other work features as required.
- Create sketch blocks.
- Constrain sketch blocks (within the context of a sketch) and get your design functioning.
- Use the make components tool to derive your sketch blocks into an assembly.
- Add 3D features to the individual files at the part level.

Pros and cons of skeletal modeling

Cons:

- Working with just the geometry on its own is not very intuitive.
- Building the part models from derived geometry can be time consuming.
- Building the part models without context can mean lots of editing in the assembly model later.

Pros:

- The derived parts are very lightweight
- Assemblies can quickly be laid out in 2D to see how they function before time is invested in 3D modeling.

Sketch blocks

To create a sketch block:

- First, create some geometry that you want to use as a sketch block
  (You don't need to add dimension constraints at this stage).
- Next, pick 'Create block' from Sketch tab > Layout panel.
- A Dialog box will open, prompting you to pick your geometry and insert point.
  Check the visibility box to make the insert point visible.
- After picking the geometry you want to block, and the point you want to use to insert your block, give your block a name and a description.
- Hit 'OK', to open the block up in the block edit environment (Note that sketch blocks can't contain projected geometry).
- Finish off adding geometric and dimensional constraints to define the behavior of your blocks. Don't forget to use your named parameter values inside your dimensional constraints to make editing sizes later easy and global.
- Click on the big green tick on the 'exit' panel to close the block edit environment.

Tip*: There are no origin planes inside the block editing environment to constrain your geometry to. Instead, add a fix constraint to fully constrain your sketch block geometry. This will help to keep the results predictable.*

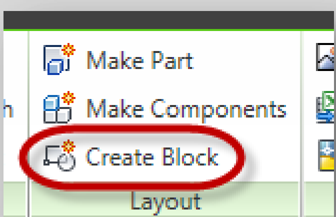


Fig. 4: Create sketch blocks

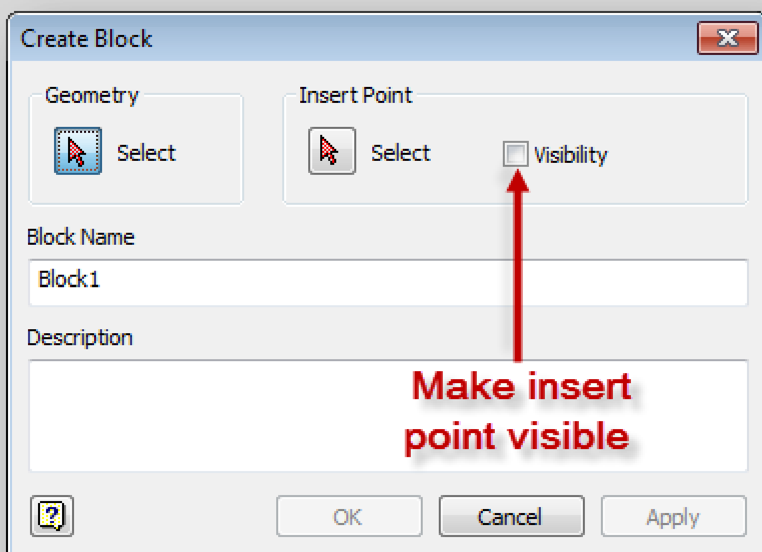


Fig. 5: Create sketch block dialog

The sketch blocks folder
You can see your newly created block definition in the browser tree in the 'Blocks' folder. Each instance of the block can be seen in the sketch where it has been placed.

Placing a sketch block
Place an instance of your block by right clicking on the block definition and choosing 'Place block' (you must be in the context of a sketch to place a block).You can edit the block at any point by right clicking on the definition or the instance and choosing 'edit block'.

Visibility of sketch blocks
Blocks have their own visibility status. Making the sketch invisible will make everything it contains invisible. Within the sketch, individual block visibility can be turned on and off as you need it.

Nesting and flexibility of sketch blocks
Blocks can be nested, and just like assemblies, they can be made 'flexible' to allow you to apply geometric constraints to control their motion.

Exploding sketch blocks
Right click on a block to explode it and return the geometry to your sketch.

Tip: *To change the insert point, visibility of the insert point, block name, or block description, right click within the block edit environment and chose 'Block properties'. Geometry cannot be selected and you must right-click off geometry to see the Block Properties option.*

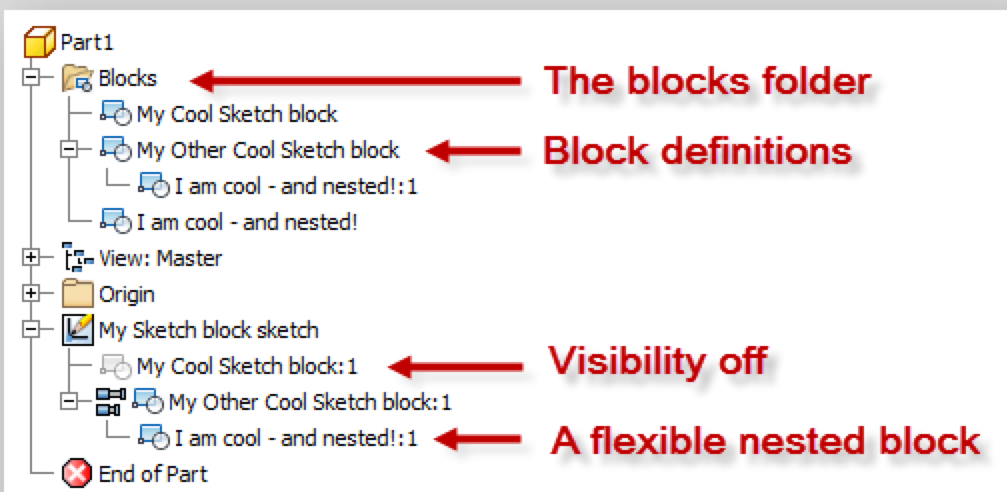


Fig. 6: Sketch blocks in the feature browser

[Check out the Inventor Wiki Help site for more on sketch blocks]

Top Down: Multibody – Design part

In the skeletal modeling technique, we limited ourselves to only defining the *geometry* we needed in our design part; the 3D features are created later.

With the Multibody technique, we can go a step further. With the Multibody design part workflow we define *all* the 3D features we need in the design part. Each part that is required for the assembly is defined as a separate solid. When the design is complete, the separate solids can be derived out using the 'Make components' tool.

The derived parts are grounded in the assembly file. The relationship between the parts is defined geometrically, rather than by using assembly constraints. Any changes to the 'Design' part automatically propagate through to the assembled parts.

Bespoke work

The Multibody workflow is excellent for bespoke work. Because you are designing the whole assembly within one part file, you can build all your 'parts' (solids) in context, making sure that they all fit and work with each other before you create an assembly model.

Because all the parameters, geometry, features and solids are held within one part file, you can build reliable relationships between features without using adaptivity.

Workflow

- Plan your assembly.
- Add your known parameters.
- Create work planes and other work features as required.
- Create sketch blocks.
- Create the geometry you need to define your design.
- Model the 3D features, separating the features for each 'part' into solids.
- Use the make components tool to derive your solids into an assembly.

Pros and cons of derived Multibody modeling

Cons:

- Multibody design parts can quickly become huge, a structured approach is required.
- Parts derived from a Multibody model can be large in file size.
- Assemblies containing lots of derived parts can take a lot of computing power to update.

Pros:

- An intuitive design technique.
- Quick and easy to edit.
- No assembly constraints means no constraints to fail.

Top Down: With iLogic

There is one golden rule that you simply can't avoid using any of the techniques we've discussed so far. You can't link a parameter from an assembly down to a part that is contained in that assembly. We've all tried... it just won't work. Unless... You use iLogic.

iLogic won't allow us to break this rue, but it does allow us to get around it. Using iLogic we can *copy* the value of a Parameter from an assembly down into a part.

The main advantage of this is that the parts don't contain links to each other, so parts can be placed into an assembly, used and then deleted and replaced if the design changes. This allows you to combine the best of library parts with parametric assemblies.

Workflow

- Create an assembly containing named parameters
- Create a part containing the same named parameters
- Place the part into the assembly
- Map the parameter values using iLogic

Pros and cons of derived top down with iLogic

Cons:

- You must design and maintain a library of templates
- Your users must understand which templates to use.
- You must educate your users in iLogic, so they don't get thrown if it doesn't work

Pros:

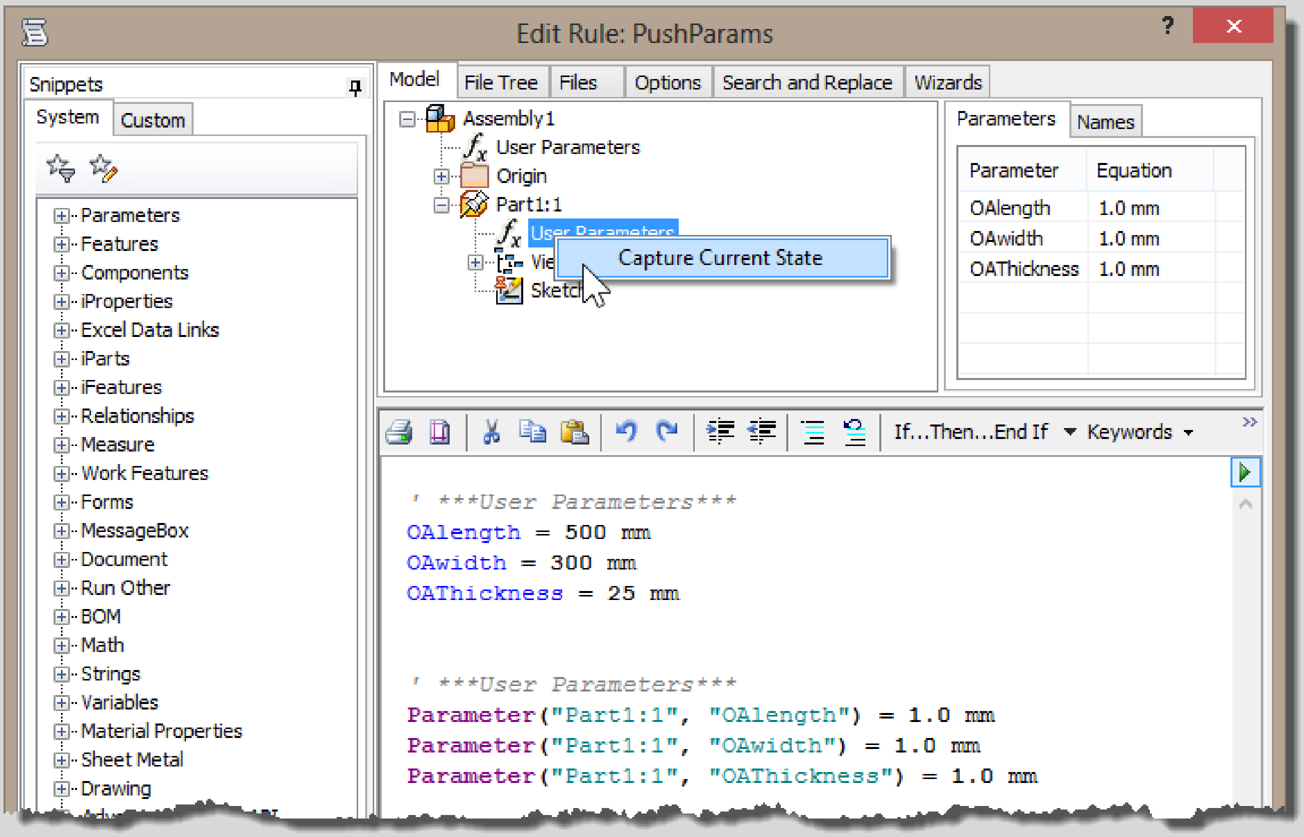
- No linking means fast updates and small files.
- Suits building with library components.
- No need to 'unpick' relationships between parts should a model fail.

Hooking up an iLogic assembly

First create your Assembly file and your Part file with matching Named parameters.

In the manage tab go to 'iLogic' panel and click 'add rule', name the rule 'PushParams'.

The iLogic window will open. Don't be intimidated, you don't need to know anything about programming to use iLogic!



Select 'User Parameters' in the browser to see the values in the box to the right. Right click over the 'Use Parameters' node, and chose 'Capture current state' to place the values into the code area. Do the same for the part parameters you wish to map.

Next, we're simply going to cut an past the values until we get the result we want. In this case, we want the value of the parameter in the part file to equal the value of the parameter in the Assembly file.

```
Parameter("Part1:1", "OAlength") = OAlength
Parameter("Part1:1", "OAwidth") = OAwidth
Parameter("Part1:1", "OAThickness") = OAThickness
```

To run the code, close the ilogic window. The code is now embedded in the assembly and will run every time you change an Assembly parameter.

The Hybrid approach

Of course, you don't have to pick one technique or another. Skeletal modeling and Multibody design parts can be combined with regular Inventor parts, iParts or even non parametric parts to complete your assembly.

The iCopy tools use derived parts and adaptivity together for a powerful method of creating many variations on a similar design.

Although an assembly model created by deriving parts in from a Multibody design part doesn't need any constraints, there's no reason why you can un-ground parts and add constraints to show motion or multiple positions.

Dividing a large or complex model up into separate Multibody parts, which are coordinated by a skeletal model, can keep the complexity of each Multibody part model down.

Mixing up skeletal modeling and Multibody parts to collaborate with your chums

One of the great benefits of working with Autodesk Inventor (Against 2D CAD) is the ability to collaborate on the design and modeling process. You can have multiple people working on the same model in real time.

Scary thought? It's not impossible – *and it can completely revolutionize how your drawing office works.*

Workflow

- Plan your assembly.
- Add your known parameters.
- Create work planes and other work features as required.
- Create sketch blocks.
- Create the geometry you need to define your design.
- Derive your 'Master' skeletal model into multiple parts files.
- Each user models the 3D features for one area separating each 'part' out as a solid.
- Bring together the 'Design' models into a preview assembly for cross checking and clash detection.
- Each user uses the make components tool to derive the solids into sub-assemblies.
- Build the final assembly from the sub-assemblies.

Pros and cons of derived Hybrid modeling

Cons:

- Your whole team needs to work in a structured and methodical manner

Pros:

- The workload can be shared across the whole department

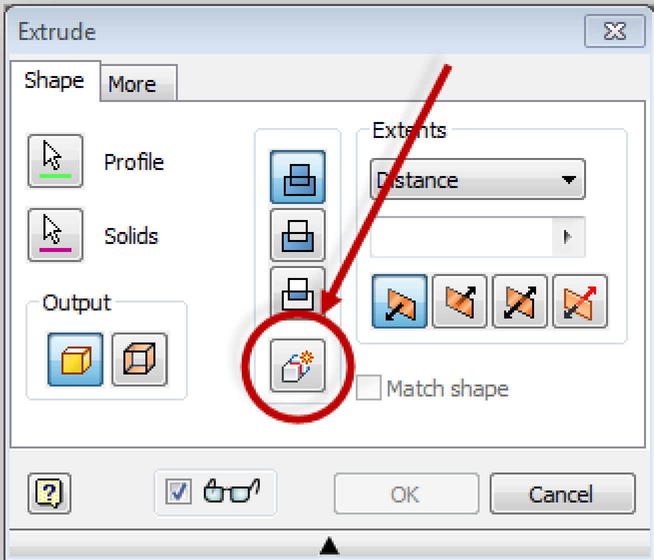


Fig. 7: Make a new solid

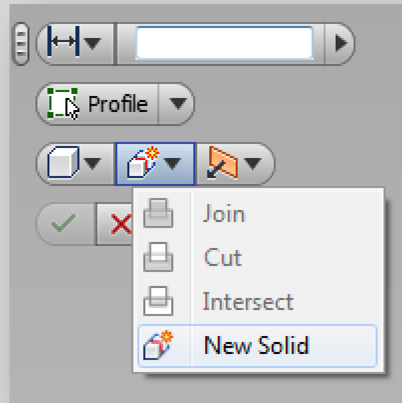


Fig. 8: Make a new solid - in canvas

Multi body parts

Multi body (or Multi solid) parts are a new feature that was introduced with Inventor 2010. Creating multiple bodies within a single part file is easy – but the controls are subtle, and easy to miss.

Creating a base solid

The first feature that you create in your part file will automatically create a new body. You can see this in the feature tree under the 'Solid bodies' folder.

Any new feature that you add will automatically be added to this body– unless you ask Inventor to create a new body!

Creating a new body

The option to create new solids can be found in the sketch based feature commands. Create a new body in the Extrude, Revolve, Loft, Sweep, and Coil commands by selecting the 'New Solid' option in the dialog box. You can also use the spit command to divide a single body in two.

Note: *If you forget to hit 'new solid', don't worry. You can go back and edit your feature at any time to turn it into a solid. However, this doesn't work the other way. You can't turn a solid back into a feature – so watch out!*

Check out the Inventor Wiki help site for more on Multi body part modeling

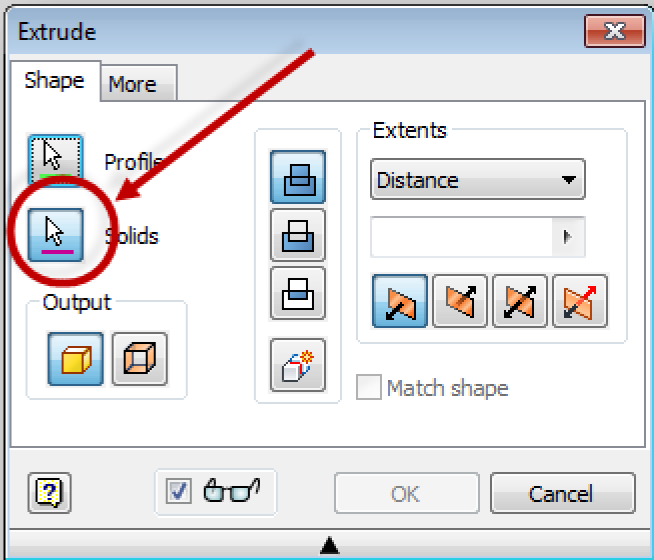


Fig. 9: Pick a solid

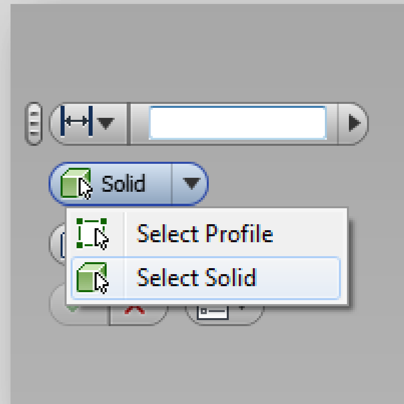


Fig. 10: Pick a solid - in canvas

Picking between bodies

As soon as you have multiple bodies, you will need to choose which body to add your new features to. By default, Inventor will add your new feature to the last solid you created. Use the 'Solids' button to add your feature to a different solid.

Sorting the feature tree

You will notice that your features are nicely organized under their bodies in the solid bodies folder, as well as being shown in their usual place in the feature tree.

However, you might find that you can't re-order your solid bodies. *The order of the solid bodies in the solid bodies' folder tree directly relates to the order of the features in the feature tree*.

To sort your bodies folder, you need to sort your feature tree. Drag the primary feature that the solid is based on up and down the tree. The solids in the solid bodies folder will update automagically.

Working with bodies

Cut features can run across multiple solids, making it easy to coordinate cut extrudes and holes between solids/derived parts.

Solid bodies can be moved and rotated independently of each other. Bodies can also be used for Boolean operations such as join, intersect or cut.

Bodies can be made invisible, independently of the other bodies in the part file.

Multibody workflow tips

Multibody parts are by their nature feature rich. They can quickly run in to tens or even hundreds of solids, and it is imperative that you work in a structured manner for predictable results.

The tips outlined below will help you to minimize the number of cross feature relationships that you build, and maximize the flexibility of your Multibody models.

The aim is to be able to drag the collection of features that represent a part up or down the feature tree without mangling your model.

You don't have to follow these tips in the order below. For example, you can go back and add parameters and work features later in the process. However, following this structured approach will help you to work in a productive manner.

Named parameters

Before you even start modeling, plan your process. Analyze the design concept that you are modeling and look for dimensions that define the design. Add these dimensions as named parameters, and refer back to your named parameters wherever possible.

Suggestions include the overall width, depth and height of the design. Distances to datums and grids. Material thicknesses, draft angles, tolerances and limits. Check the 'export' box in the parameters manager to make sure that any coordinating parameters are available to be derived out to the part files.

***Tip**: Use the named parameter description field to remind yourself and your colleagues of what each parameter is doing.*

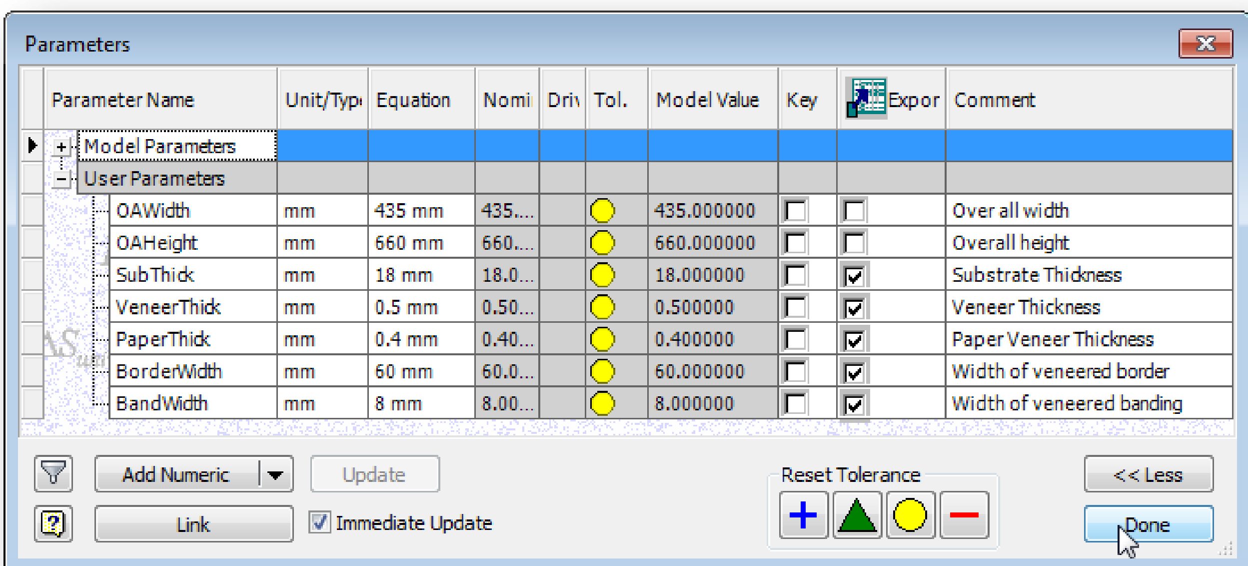


Fig. 11: The parameters manager

Work features
Use your named parameter values to create Work Planes to represent your Grid and Datum lines, along with any other work features you will need to coordinate your model.

***Tip**: When building Multibody parts it can reduce flexibility to use the origin planes. Instead 'Overlay' the origin planes with your own work planes set to an offset of 0. This will allow you to make adjustments later.*

Defining geometry
Create some sketches on your work planes, referring to your named parameters, to define the overall width, depth and height of your design. Use construction geometry and constrain all of these sketches to the origin (or your own datum point).

Use these sketches as the structure to 'hang' the rest of your sketches upon.

Sketches
Keep it simple; keep it simple – KEEP IT SIMPLE.

When part modeling, creating only one closed profile loop in each sketch is recommended. By its nature you will want to use sketches with multiple closed loops to coordinate features in your Multibody design parts. *This can be the source of most of your problems when editing Multibody parts.*

- Don't create sketches on features. Use your work features instead.
- Look for opportunities to split your profiles over multiple sketches.
- Keep projected geometry to a minimum – use your named parameters and construction geometry to coordinate sketches.
- Use sketch blocks wherever possible.
- Don't let Inventor 'over constrain' your sketches.

***Tip**: Make use of 'Constraint persistence' to turn off automatic constraints when you need to be certain of what constraints are being added. You can quickly toggle constraint persistence off by holding down the CTRL key while sketching.*

Sketch Blocks
If you need to use the same profile in multiple areas, create this profile as a sketch block. Don't forget to link your sketch blocks to your named parameters for easy adjustment later.

Feature modeling
Don't put *every* feature into your design model. Look for opportunities to put 'secondary' features that don't need to be coordinated in your design model (such as fillets and chamfers) into the individual part files. You can coordinate the dimensions of these features by deriving your named parameters into the part.

Library features
Multibody parts can be 'assembled' using iFeatures. If you are regularly adding the same features to your Multibody design parts, consider saving them out as iFeatures and adding them to your iFeature gallery.

'Flexing' the design part

If you are building your design around parameters that you are confident will change (After all, this is why you are using Inventor – right?) 'Flex' your design part regularly to check for any cross feature relationships that might fail.

Fix failures *now* before you build another 101 solids that rely on your base geometry!

Breaking it down

Look for opportunities to break your design model down into smaller chunks by creating derived parts which contain the coordinating geometry only. Model up each multi body sub-part in context, and then bring all the design parts back into an assembly for checking before you derive out your parts into assemblies.

Sorting the feature tree

Keeping your main coordinating sketches bunched together at the top of the tree can make editing the design part later much more intuitive. Place any secondary sketches immediately before the features that they are controlling.

Naming features

Name all your sketches, primary features and solids as you go. Don't forget that it might be YOU who has to come back and edit this model in 6 months' time. Make it easy to see which work features and sketches are controlling which features.

Tip*: Feature names must be unique. I suggest that you suffix the feature name with the feature type to help you see which features are related.*

Automating the prefix and suffix

The '[Make components'] tool will automatically pick up the names of your solids to use as part names. You can change this in the make components' Dialog box if you want.

Tip: *Check out this special secret setting which you can use to ask Inventor to automatically add a prefix or suffix to your solid name when creating parts.*

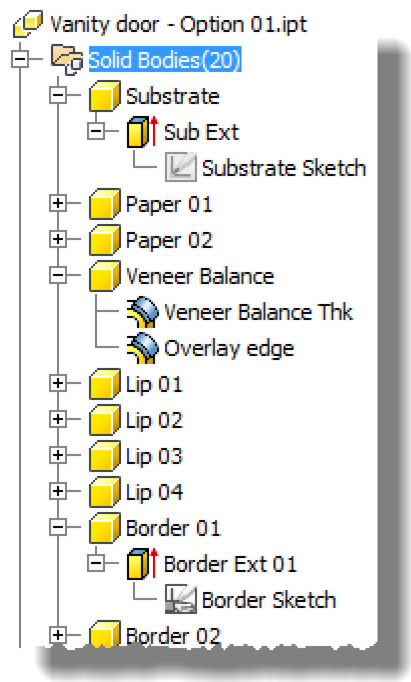


Fig. 12: Example feature naming

Get the code: *You can also use this iLogic code to rename your solids before deriving them into parts.*

Materials and templates

Each part file can only contain one material. You could change the *appearance* of each individual solid in the design part, but I don't recommend it. Instead, allocate materials and appearances to the individual derived parts.

Allocating appearances in derived parts will override the appearance setting in the part file. Check out this link for advice in dealing with this issue. Default setting for the option "Use color override from source component" in the Derive Part dialog.

The 'Make components' tool allows you to pick individual template files for each solid that you are deriving. Creating and using template files which represent your commonly used materials will save you hours.

Deriving the whole enchilada
The Make components tool can be used to derive sketches, sketch blocks, work features solids and even parameters into your part files. *Don't forget to click the '[Include parameters](#)'* button*!*

Editing Multibody models
Even after you have taken all of these precautions, editing a complex, feature rich Multibody design part can be a headache.

If you need to edit work features, sketches or features near the top of the feature tree – roll the end of part marker (EOP) up underneath the feature or sketch before you edit it. After you have made the change, roll the EOP down again a few features at a time and fix any problems as you go.

Tip*: DON'T flip back to the assembly with the EOP rolled up. The solids in your design part will be suppressed, and all the derived parts in the assembly will fail. This is an annoyance but getting them back is easy. Go back to the design part and roll the EOP back down, then flip back to your assembly and update.*

'Pulling' geometry with the Derive tool

The 'traditional' method of deriving geometry into part files is to 'pull' the geometry through using Inventor's 'Derive' tool.

This has largely been superseded by the 'Make part' and 'Make component' tools, but it's still a technique worth knowing about.

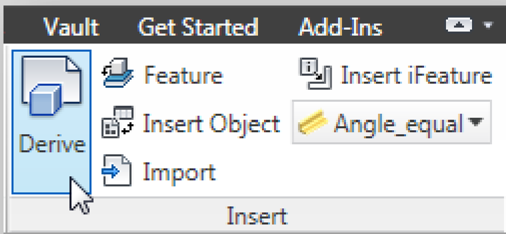


Fig. 13: The Derive tool

You will find the derive tool on the Manage tab > Insert panel.

Workflow

- Create a new part
- Close and delete the default base sketch
- Click on the 'Derive' tool
- From the file open Dialog, pick a part or assembly that you wish to derive from
- From the Derived part Dialog pick the solids, surfaces, 2D & 3D sketches, sketch blocks, work features or parameters that you want to derive into your new part
- *Note that you can mirror the geometry as it comes in*
- Hit OK to complete the process

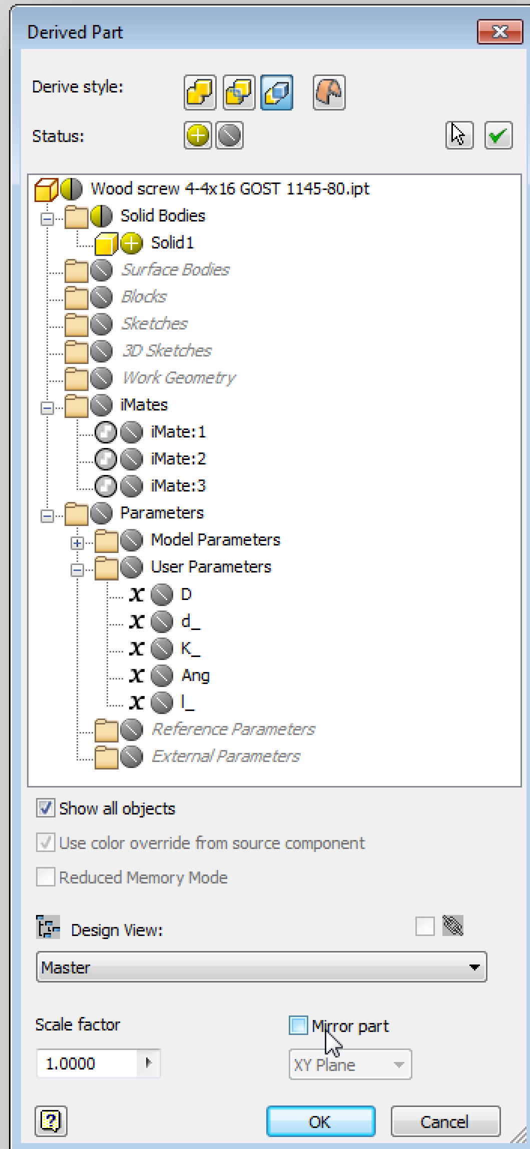


Fig. 14: Creating or editing a Derived part

Tip: *You can speed up the building of multiple part files from skeletal geometry by temporarily saving this IPT as a template file.*

'Pushing' geometry with the Make part tool

As well as 'pulling' geometry into our parts, we can also 'push' geometry from our multibody master part into our part files.

You will find the Make part tool on the Manage tab > Layout panel

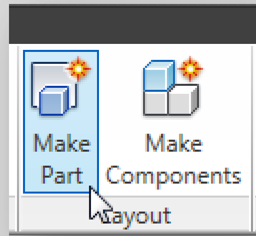


Fig. 15: The make part tool

Workflow

- Open your skeletal or multibody 'Master' part.
- Click on the 'Make' part tool
- From the Make part Dialog pick the solids, surfaces, 2D & 3D sketches, sketch blocks, work features or parameters that you want to derive into your new part
- Chose a part name, template, BOM structure and file location for your new part.
- *Note that you can derive geometry into an existing part.*
- To optionally push your part out into an assembly, check the 'place in target assembly box'.
- *Note that you can create a new assembly, or pick an existing assembly.*
- Click on 'OK' to complete the process.

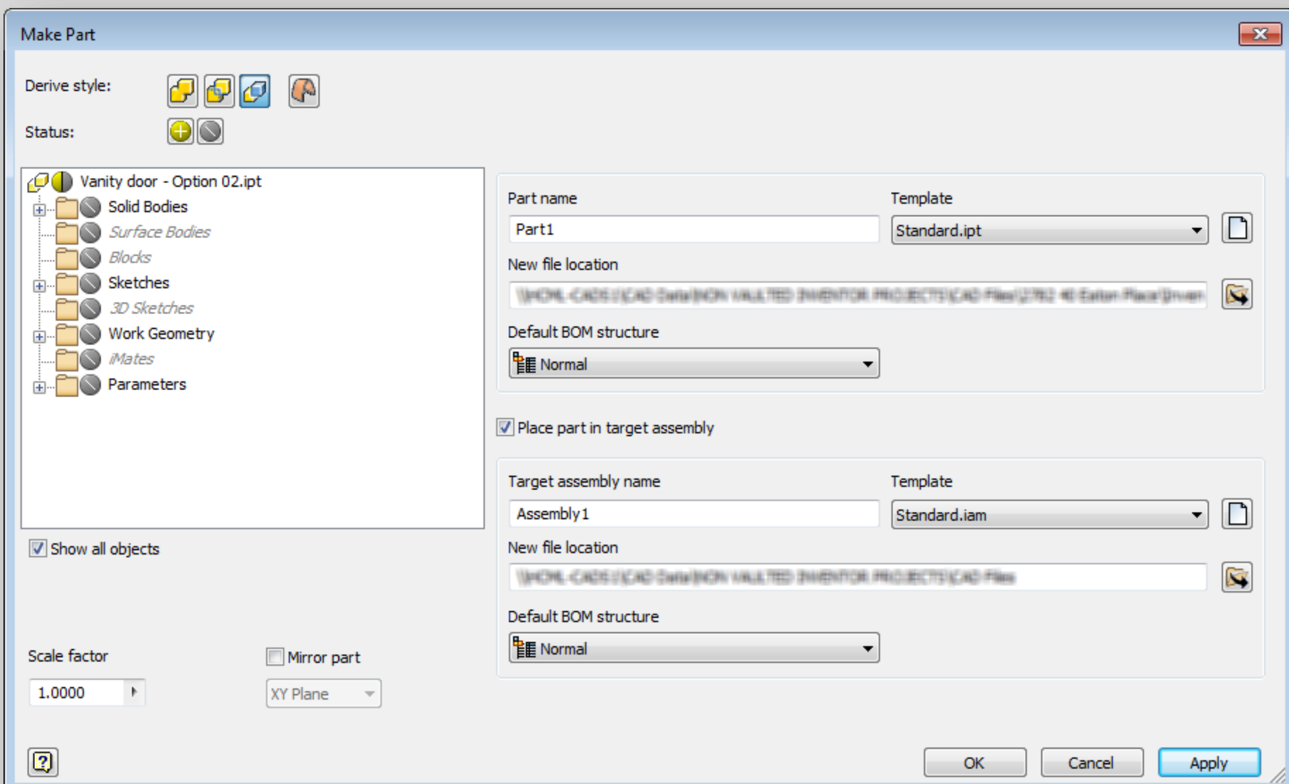


Fig. 16: The make part dialog

'Pushing' out multiple derived parts with the 'Make components' tool.

The Make components tool automates the process of deriving out multiple bodies into multiple parts, subassemblies and assemblies. It is a powerful tool and a real time saver.

However, like any process you will want to make sure that you get the first one right before you go into mass production. I recommend that you experiment with a small section of your assembly and get it right before you go spitting out hundreds of parts.

You will find the Make part tool on the Manage tab > Layout panel

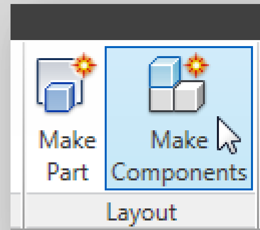


Fig. 17: The make components tool

Workflow

- Open your multibody master part
- Check that your named parameters are available for export, save if necessary
- Click on the 'Make component' tool
- From the Make components: selection Dialog, pick the solids that you want to derive
- Optionally choose to derive your parts into an assembly
- *Note that you can derive your parts into an existing assembly*
- Click the 'next' button to move onto the Make components: Bodies Dialog
- *Note that you can go back to the Selection Dialog at any time with the '<<Return to selection' button (Cont' on next page)*

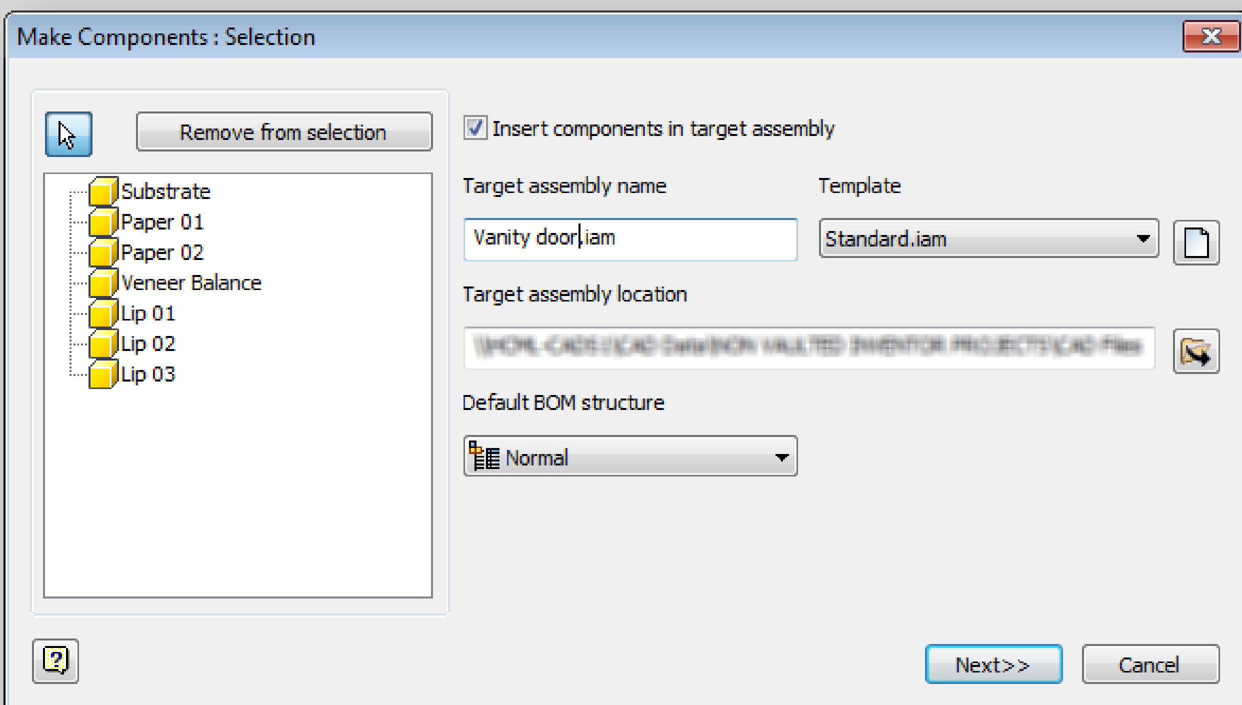


Fig. 18: The make components - Selection dialog

Workflow (Con't)

- Optionally change the component name, template, BOM structure and file location for the individual parts
- *Note that you can change the template or file location for multiple parts at once, by highlighting them and picking the buttons at the top of the Dialog*
- Don't forget to pick the 'Include parameters' button to derive your named parameters into your part files.
- *Note that you can derive the solids out as surfaces only, and create mirrored derived parts from this Dialog*
- Click on OK to complete the process

Templates

Creating templates for commonly used materials can save you a lot of time. You can allocate which template to use for which part in the Make components: Bodies Dialog.

Include parameters

Don't forget to click the 'include parameters' button, to derive through your named parameters into your part files.

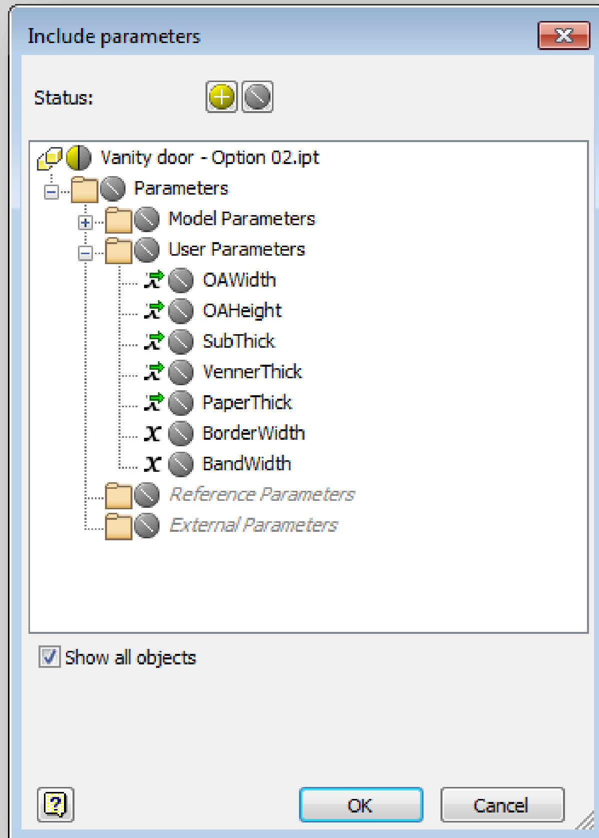


Fig. 20: The include parameters dialog

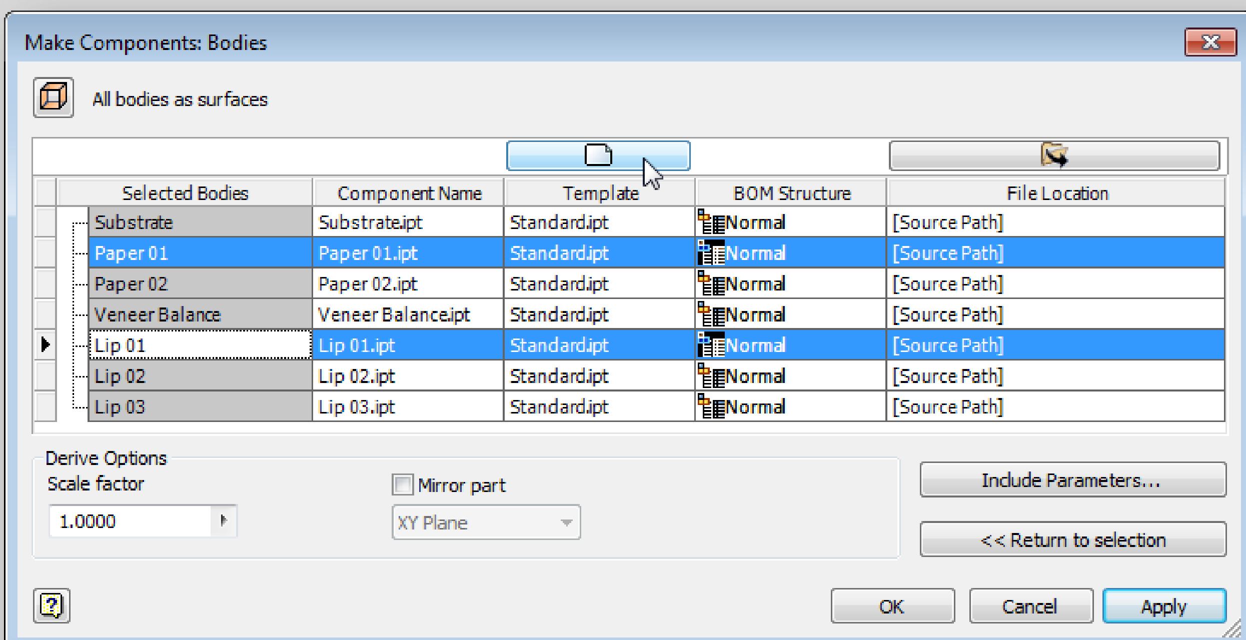


Fig. 19: The make components - Bodies dialog

Auto naming of parts

By default, Inventor uses the name of the solid as the name of the part file in the make components Dialog. You can change the name of the part file in the make components Dialog simply by typing a new name in.

To automatically add a prefix of suffix to the part names, go to:

Tools tab > options panel > Document settings

Click on the 'Modeling tab', then the 'Make components Dialog' > 'Options…' button.

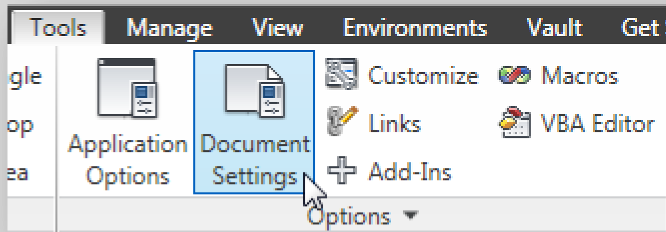


Fig. 21: Tools > Document settings

You will find options to set the prefix and suffix of part and assembly names, as well as setting default file locations and templates.

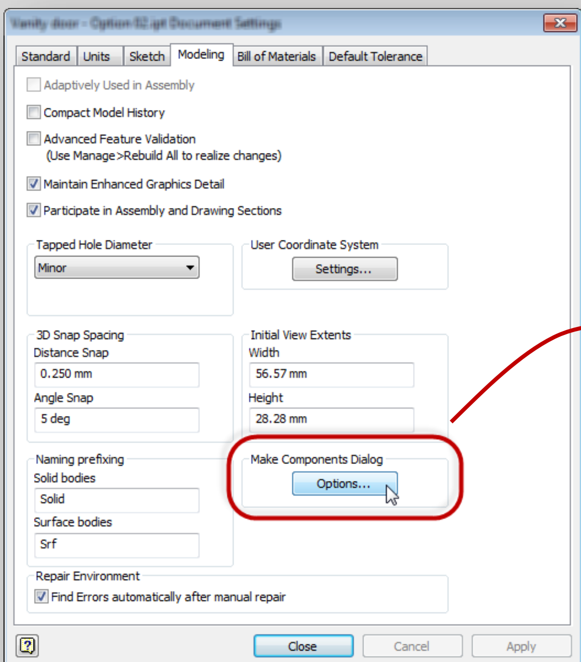


Fig. 23: Make components dialog > Options

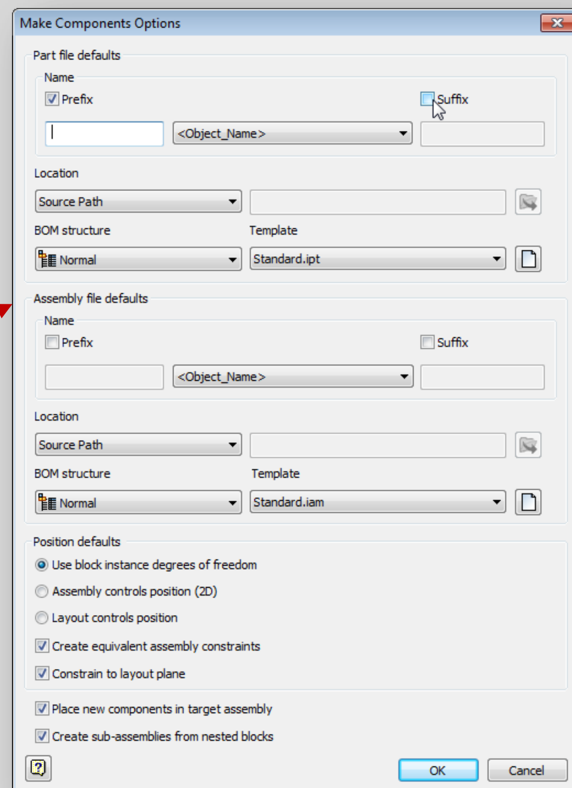


Fig. 22: Make components Options

Editing parts

If you want to change or add to the geometry that you derived out into your part files, open the part file expand the feature tree to show the derived body, right click on the derived body and chose edit.

This will open the edit derived part dialog.

For more on Derived part modeling check out the Autodesk wiki

Tips on large assembly models with master parts

Large and complex assemblies have their own considerations. Any advice you ever read on managing large assemblies applies to assemblies derived from design parts too.

Here are a few tips that specifically apply to the derived part workflow.

Don't go Multibody mad

Don't put *every* part that you need into your design model. Look out for opportunities to build Library parts such as hardware and fixings in the traditional manner. Using library parts and iParts will speed up your design process.

Use feature based parametric design sparingly

This one is counterintuitive. We are using Inventor because we want a parametric design tool – right? But think about it; do you have library parts that represent items you buy in from outside the company? Do *they* really need to be parametric?

Even if you design these parts in Inventor, saving out and using a copy in a neutral file format can really bring the complexity of your assembly models down.

Keep it simple

Keep each part as simple as possible, particularly for parts that you won't be manufacturing in house. That 0.1 fillet on all the edges might be technically accurate, but if you are not manufacturing the part, you don't need that information.

Not only will simplifying your library parts speed up your assembly model, but it will also speed up your drawings, and prevent your drawings being cluttered up with feature lines that aren't required.

Keep it simple (Again)

Don't create features or solids in your multibody design parts that represent the same part in different locations. Instead use the same part in your assembly multiple times and constrain it into position. If you can - patterning parts in assemblies rather than copying them is even less taxing on Inventor.

Tip*: The make components tool has an option to Mirror solids as you derive them into parts.*

Use Inventor for what it's good at

Don't create holes in the design part for features such as bolts. Instead, create sketch geometry in your master part to define the hole center, derive the hole center through to the part, and use the bolted connection generator to add the hole and bolt in one swift move.

You can also mix and match design parts with the frame generator in a similar fashion.

Faster updates

The advantage of creating an assembly model based on the same design part is that changes to the design part will automatically propagate to all the parts derived from it.

The disadvantage is that Inventor will check every single derived part in your assembly for changes whenever you save your design part – *even if you didn't make any edits to the design part*. Depending on the size and complexity of your assembly and the power of your computer, this could add a long time to your design process.

One way to speed up the process is to suppress the link between the derived parts and the design part. Don't forget to un-suppress the links when you actually *want* changes to your design part to propagate through!

Check out this iLogic code to suppress all the links at once.

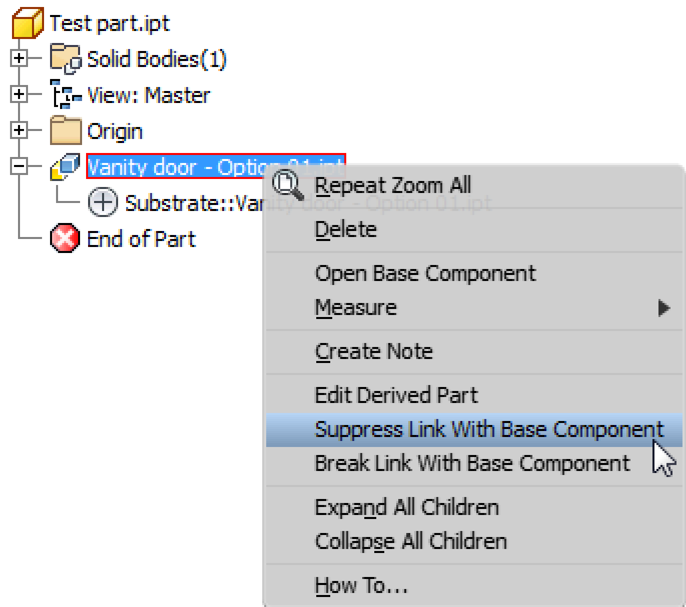


Fig. 24: Suppressing links in derived parts

In conclusion

I hope that you come away from this presentation with a renewed enthusiasm for building your parametric design models with Autodesk Inventor. It is said that you don't know what you don't know. But now you know. So go away and build!

Credits

Thanks very much to Scott Moyse and John Evans of DesignandMotion.net, Jon Landeros of Inventor tales and Curtis Waguespack of Inventor from The trenches for their contributions (Any mistakes are my own!).

Finally thanks very much to my bosses at Halstock Cabinet makers, Richard, Roger and Jasper, for their support, and thanks to the rest of the CAD team at Halstock for being my guinea pigs.

References

For more on Autodesk Inventor and derived modeling techniques:

Cadsetterout.com - Assembly techniques for woodworkers
Designandmotion.net - Top down design with Inventor
Inventor-tales.blogspot.co.uk - Tips for multibody part modeling
Opendesignproject.org - A very detailed tutorial on derived part modeling
Blogs.rand.com - A great starter tutorial on multibody part modeling
Wikihelp.autodesk.com - The Inventor Wiki page for top down design
Inventortrenches.blogspot.co.uk - Tips on iLogic and multibody parts
beinginventive - Turning off color overrides on derived components

Appendix A – Get the code!

Thanks to Curtis Waguespack author of 'Mastering Autodesk Inventor' and writer of [Inventor from The trenches](http://Inventortrenches.blogspot.co.uk/) for sharing this iLogic code.

Suppress derived part links

One disadvantage of the derived part technique is that Inventor will check the master part for updates from every part in the assembly that contains a link.

This code temporarily suppresses the link to speed up your assembly updates. Don't forget to un-suppress the link when you want the whole assembly to update!

```
'--Suppress the link to a master part---
'-------Code by Curtis Waguespack-------
'http://Inventortrenches.blogspot.co.uk/

oSuppress_YN = InputRadioBox("Select an option", "Suppress Derived Links",
"Un-Suppress Derived Links", True, "iLogic")

' Get the top level assembly document
Dim oDoc As AssemblyDocument
oDoc = ThisApplication.ActiveDocument

' Set a reference to the assembly component definition
Dim oComp As AssemblyComponentDefinition
oComp = oDoc.ComponentDefinition

' Loop each component occurrence to get its document
Dim oOcc As ComponentOccurrence
For Each oOcc In oComp.Occurrences
Dim oPartDoc As Document
oPartDoc = oOcc.Definition.Document

If TypeOf oPartDoc Is PartDocument Then
' Set a reference to the part component definition
Dim oPartComp As PartComponentDefinition
oPartComp = oPartDoc.ComponentDefinition

' Loop each derived part component in this part document
Dim oDerPartComp As DerivedPartComponent
For Each oDerPartComp In oPartComp.ReferenceComponents.DerivedPartComponents
oDerPartComp.SuppressLinkToFile = oSuppress_YN
Next
End If
Next
```

------- end of ilogic ------

Ground components

If you are using the Make components tool, Inventor will automatically ground your components about the origin for you. Alternatively you could use the Assembly bonus tool 'Ground and root component'.

This code will ground your components around the origin, without adding additional assembly constraints.

Tip: *To add large numbers of parts to an assembly without having to pick an insert point, drag and drop the parts into Inventor from windows explorer.*

```
'---------Ground all components---------
'-------Code by Curtis Waguespack-------
'http://Inventortrenches.blogspot.co.uk/

' set a reference to the assembly component definintion.
' This assumes an assembly document is open.
Dim oAsmCompDef As AssemblyComponentDefinition
oAsmCompDef = ThisApplication.ActiveDocument.ComponentDefinition

'Iterate through all of the occurrences
Dim oOccurrence As ComponentOccurrence
For Each oOccurrence In oAsmCompDef.Occurrences

'check for and skip virtual components
'(in case a virtual component trips things up)
If Not TypeOf oOccurrence.Definition Is VirtualComponentDefinition Then

'set a reference to the transient geometry object
oTG = ThisApplication.TransientGeometry

'set the occurence rotation and translation to 0,0,0
oOccurrence.Transformation = oTG.CreateMatrix

'ground each occurence
oOccurrence.Grounded = True

Else
End If
Next

iLogicVb.UpdateWhenDone = True
```

------- end of ilogic ------

Rename all solid bodies

This code allows you to rename all the solids in your multibody part using a prefix. When you press the OK button the rule renames each solid body using the prefix and an increment number, adding a leading zero for each number less than 10.

```
'---------Rename all solid bodies-------
'-------Code by Curtis Waguespack-------
'http://Inventortrenches.blogspot.co.uk/

'check for custom iProperty and add it if not found
Dim prefix As String = "Prefix"
customPropertySet = ThisDoc.Document.PropertySets.Item _
("Inventor User Defined Properties")

Try
      prop= customPropertySet.Item(prefix)
Catch
     'Assume error means not found
         customPropertySet.Add("", prefix)
End Try

'write the part number to the Prefix iProperty if it is empty
if iProperties.Value("Custom", "Prefix") = "" Then
iProperties.Value("Custom", "Prefix") = iProperties.Value("Project", "Part Number") & "_"
else
end if

'check that this active document is a part file
Dim partDoc As PartDocument
If ThisApplication.ActiveDocument.DocumentType <> kPartDocumentObject Then
MessageBox.Show ("Please open a part document", "iLogic")
End If

'define the active document
partDoc = ThisApplication.ActiveDocument
Dim solid As SurfaceBody
Dim i As Integer

'get input from user
prefix = InputBox("Enter a prefix for the solid body names", "iLogic", iProperties.Value("Custom", "Prefix"))

'write input back to custom iProperty
iProperties.Value("Custom", "Prefix") = prefix
i = 1
'rename all solid bodies incrementing suffix
For Each solid In partDoc.ComponentDefinition.SurfaceBodies
solid.Name = prefix + IIf(i < 10, "0" + CStr(i), CStr(i))
i = i + 1
Next
------- end of ilogic ------
```

Purge derived part parameters

It's tempting to derive all your parameters through into your part files, however this can slow down updates on your assembly.

This code from Luke Davenport 'purges' all derived parameters that aren't being referenced. You can find the original post here:

http://www.cadlinecommunity.co.uk/Blogs/Blog.aspx?ScoId=352228f8-6b7a-447f-a6bd-5b94590d3175

This iLogic code Is from Luke Davenport's Cadline Blog – it looks For any unused derived parameters in part files And 'un-derives' them.

```
' Start of iLogic code - © Copyright Luke Davenport @ Cadline 2013
Imports System
Imports System.IO
Sub Main
' Check whether open document is an assembly and exit rule if not
oDoc = ThisDoc.ModelDocument
If oDoc.DocumentType = kPartDocumentObject Then
MessageBox.Show("This rule can only be run in an assembly file - exiting
rule", "iLogic")
Return
End If
Dim i As Integer = 0
Dim doc As Document = ThisDoc.Document
Dim docname As String
Dim doclastname As String
Dim oParam As Parameter
Dim ParamName As String
Dim ParamInfo As String
Dim Exists As String = "" ' Used for saving out txt file
Dim CurrentState As String
Dim NewState As String
Dim BrokenLinkCount As Integer = 0
Dim BrokenLinkCountText As String = ""
Dim OutputText As String = ""

' Avoid overwriting an existing text file if it already exists
Dim txtfile As String = ThisDoc.FileName(False) & ".txt"
If System.IO.File.Exists(txtfile) Then 'txt file already exists
Overwrite = MessageBox.Show("A text file called " & ThisDoc.FileName(False) &
".txt already exists, overwrite it?", "Cadline
iLogic",MessageBoxButtons.YesNo, _
MessageBoxIcon.Question,MessageBoxDefaultButton.Button1)
If Overwrite = vbNo Then
MessageBox.Show("Exported text file will be named: " &
ThisDoc.FileName(False) & "_Cadline.txt", "Cadline
iLogic",MessageBoxButtons.Ok, _
MessageBoxIcon.Information,MessageBoxDefaultButton.Button1)
Exists = "_Cadline" ' To avoid overwriting an existing file
Else
IO.File.Delete(txtfile)
End If
```

```
End If
For Each doc In oDoc.AllReferencedDocuments
If doc.DocumentType = 12290 Then 'part file
'Find if there are any derived parts in the document
If doc.IsModifiable = False OrElse doc.ComponentDefinition.IsiPartMember Then
' Ignore it
Else ' Document is writable - go ahead
Dim oRefComp As DerivedPartComponent
Dim oRefComps As DerivedPartComponents
oRefComps = doc.ComponentDefinition.ReferenceComponents.DerivedPartComponents
If oRefComps.Count > 0 Then
' Find if there are any derived assemblies in the document
Dim oRefAsms As ReferenceComponents
oDerAsms =
doc.ComponentDefinition.ReferenceComponents.DerivedAssemblyComponents
If oDerAsms.Count > 0 Then
MessageBox.Show("There are derived assemblies present in this assembly!" &
vbLf & vbLf & _
"Unfortunately this iLogic rule only works with derived parts - exiting now",
"Cadline iLogic",MessageBoxButtons.Ok, _
MessageBoxIcon.Information,MessageBoxDefaultButton.Button1)
Return
End If
docname = doc.DisplayName
For aa = 1 To oRefComps.Count
oRefComp = oRefComps.Item(aa)
Try
OutputText = "The following parameters have been deleted (un-derived) in file
"
Dim oDPDef As DerivedPartDefinition = oRefComp.Definition
Dim oDParams As DerivedPartEntities = oDPDef.Parameters
Dim oDParam As DerivedPartEntity
For Each oDParam In oDParams
CurrentState = oDParam.IncludeEntity
If CurrentState = "True" Then
oDParam.IncludeEntity = False
NewState = oDParam.IncludeEntity
If NewState = "False" Then
i = i + 1
ParamName = oDParam.ReferencedEntity.Name Call
WriteParamInfo(ParamName,docname,doclastname,Exists,OutputText)
'Recognise when part changes (line space needs adding to .txt file)
doclastname = docname
End If
End If
Next
oRefComp.Definition = oDPDef
Catch
BrokenLinkCount = BrokenLinkCount + 1
OutputText = "A broken (or suppressed) link to the derived part was found in
file "
Call WriteParamInfo(ParamName,docname,doclastname,Exists,OutputText)
End Try
Next
```

```
End If
End If
End If

Next
InventorVb.DocumentUpdate()

If BrokenLinkCount > 0 Then
BrokenLinkCountText = "(Note that " & BrokenLinkCount & " parts were found
with 'broken' links to the derived " _
& vbLf & "part. These have not been updated and are listed in the output
file)"
End If

'Inform user how many parameters have been deleted
If i = 0 Then ' no unused parameters found
MessageBox.Show("No unused derived parameters found!" & vbLf & vbLf & _
"No changes made to current assembly", "Cadline iLogic", _
MessageBoxButtons.Ok,MessageBoxIcon.Information)
Else ' unused parameters found
USure = MessageBox.Show(i & " parameters have been deleted" & _
" - do you want to see a list of these?" & vbLf & vbLf & _
BrokenLinkCountText, "Cadline iLogic", _
MessageBoxButtons.YesNo,MessageBoxIcon.Information)
If USure = 7 Then
Return
Else
'Open generated text file
System.Diagnostics.Process.Start(ThisDoc.FileName(False) & Exists & ".txt")
End If
End If
End Sub

Sub WriteParamInfo(ByVal ParamName As String, _
ByVal docname As String, _
ByVal doclastname As String, _
ByVal Exists As String, _
ByVal OutputText As String)
Using writer As StreamWriter = _
New StreamWriter(ThisDoc.FileName(False) & Exists & ".txt", True)
If Not String.Compare(doclastname,docname,True) = 0 Then
writer.WriteLine("")
writer.WriteLine(OutputText & docname)
writer.WriteLine("")
End If

If OutputText.Contains("The ") Then
writer.WriteLine(" " & ParamName)
End If

End Using
End Sub
```