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Reliable Modelling Techniques for Complex Assembly Design in Autodesk Inventor

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Autodesk

Learning Objectives

- Learn how to structure assemblies effectively
- Learn how to make use of relationships effectively, and when to use the alternatives.
- Learn how to manage large assemblies.
- Learn how to trigger 'Top Down' parametric changes with iLogic.

Description

How often have you made a simple change to a part file in Autodesk Inventor, only to return to the assembly - which EXPLODES!

I know that this has happened to you - it's happened to me too!

In this class we will learn simple and effective strategies for building parametric, stable assemblies that can easily be updated.

We will discuss best practice for structuring assemblies and how to scale when working with large assemblies. We will learn how to use relationships effectively, and when to use the alternatives.

We will learn how to prepare a [design](#) for 'top down' parametric change, and how we can trigger changes using iLogic.

Finally, we will learn how to document our [design intent](#) to ensure that our colleagues can work with our assemblies as effectively as we can.

Supporting documents

Don't forget to download the dataset and watch the presentation on AU Online.

<https://www.autodesk.com/autodesk-university/au-online?query=paul+munford>

Paul Munford

Paul Munford is a laughter, dreamer, raconteur, CAD geek and Technical Marketing Manager for Autodesk in the UK.

Paul 's background in manufacturing items for the construction industry gives him a foot in digital prototyping and a foot in Building Information Modeling (BIM).

Paul was a speaker at Autodesk University for the first time in 2012, and he says it's the most fun anyone can have with 250 other people in the room.



Advanced Part Modelling

This handout continues the modelling best practice which started with:

'Reliable Modelling Techniques for Complex Part Design with Inventor'

Click on this link to watch the presentation and download the handout and dataset.

<https://www.autodesk.com/autodesk-university/au-online?query=paul+munford>

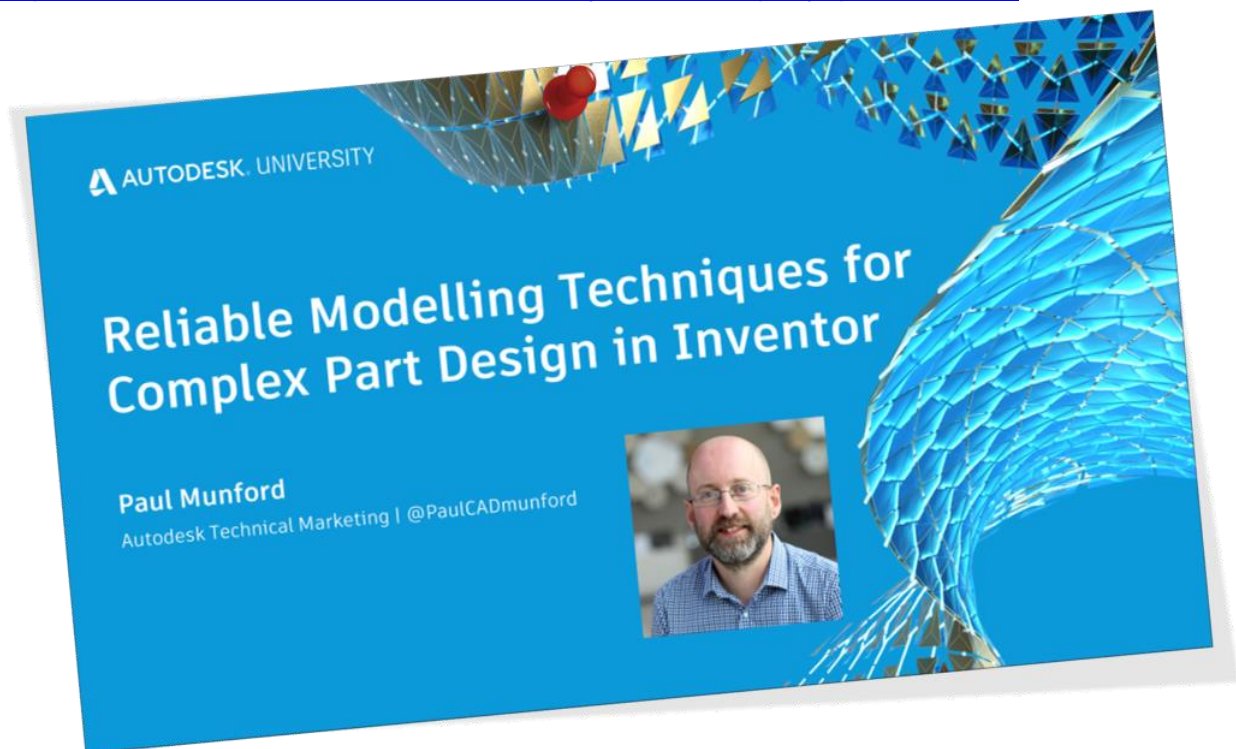


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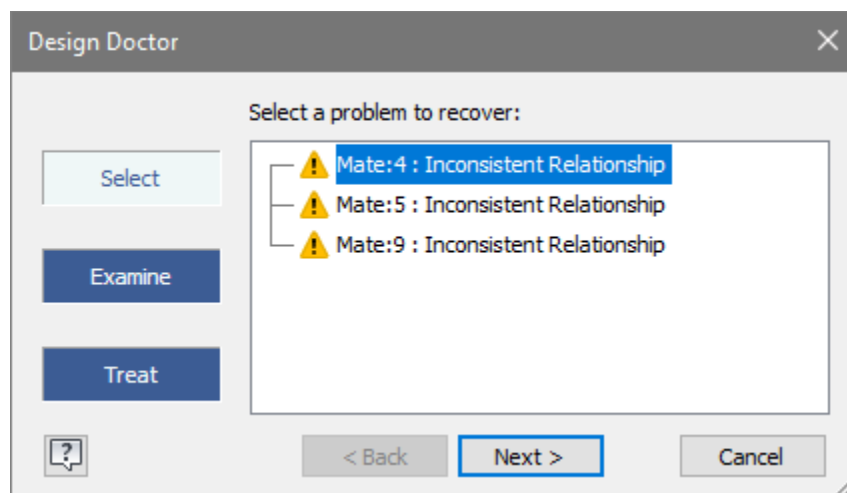
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Advanced Assembly Modelling

Reliable Modelling Techniques for Complex Assembly Design in Autodesk Inventor

Why is parametric modelling so hard? One minute you have a perfectly good model, looking fine, the next minute EXPLOSION!

I know that this has happened to you – it's happened to me too.



How to use Inventor properly?

When I'm teaching Inventor I often get asked:

'How do I use Inventor properly?'

Of course – there is no *'Right'* way to use Autodesk Inventor. It's a tool just like any other. We can use it for lots of different tasks, and in lots of different ways – all of them correct.

So, how can we quantify a 'well modelled' Assembly?

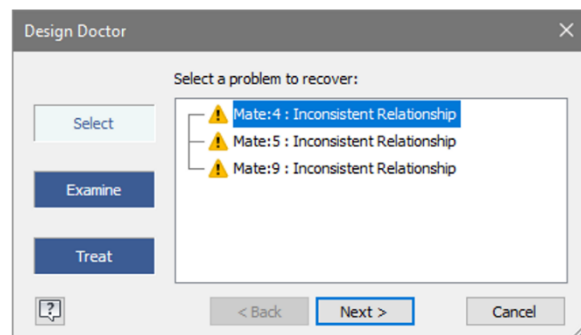
I've given this a lot of thought – and I can only come up with two criteria.

1. The data must be **correct**.
2. The design must be **easy to update** (this is the tricky part).

✓ Correct Data



✓ Easy to update



The problem with parametric modelling

The problem with parametric modelling, is that we must model in *4 dimensions*.

We model in the usual three dimensions, and we must also consider *Time*, or the way our model might change over time.

This change over time is often referred to as '**Design Intent**'. Building a model that can change in a predictable fashion takes a little thought and some planning.

The problem with parametric modelling is...

It's easy to unintentionally create relationships (Booby traps!).

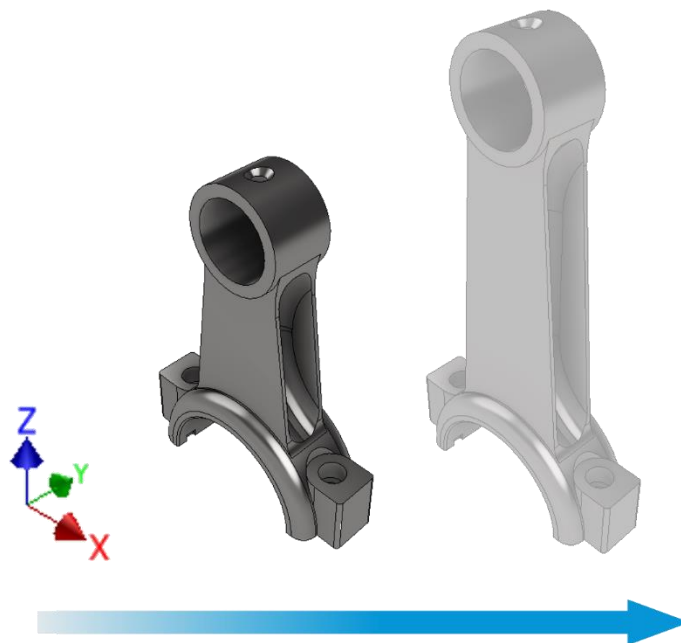
Editing a feature causes all subsequent features to regenerate.

Part updates become unpredictable.

Design intent is lost.

Time is lost 'fixing' designs, or re-modelling from scratch.

We would rather model our own designs, than re-use someone else's.



Reliable modelling technique

Reliable modelling technique takes effort up front. Back in the day, working productively in AutoCAD meant mashing the keyboard faster.

Working productively in Inventor is more like playing chess. It pays to sit back and think about what we're going to do before we start.

Reliable modelling requires a plan.

Reliable modelling gives us...

Editable models – Design intent is captured.

Obvious models – Design intent is documented.

Reusable models – We would rather re-use than rebuild.

Component **File type**
 Component **Complexity**
 Component **Quantity**
 Component **Relationships**
 Component **Adaptivity**
 Derived **Components**
 Linked **Parameters**
*Multiple **Updates***
*By Multiple **People***



What makes an Assembly complex?

What makes an Assembly model complex?

If your model only has a few components, you don't have to worry about your colleagues figuring out how to change your design.

If you are creating a model of a [component](#) that isn't subject to change (for example a supplier's component), you don't need to worry about 'Design Intent'.

Building reliable models will take you *longer* in the short term. Building reliable models will save you time, only if you will be changing the design numerous times in its life time, or you can reuse a component in multiple designs.

Reliable modeling is a prerequisite when using a [top down modelling technique](#), or building configurable content such as iParts, iAssemblies or iLogic driven designs.

Relationships

The key to creating complex models in Autodesk Inventor is maintaining control of *relationships*.

If you don't understand the relationships you've built between parameters, sketches, features, bodies, parts constraints and assemblies – your model will *not* update in a predictable fashion.

The bad news is that Inventor won't manage this for you (it can't read your mind!).

The good news is that YOU have full control over this process.

Relationship rules

- No **unintended** Relationships
- Relationships are kept to a **minimum**
- All relationships are **planned** and **purposeful**
- All relationships are **obvious** and **easily understood**

An Assembly file as a Database

The main function of a part file is to have the correct geometry. Without the correct geometry we won't get the part made that we want.

Assembly models have a different purpose. They allow us to conceptualise our whole design in context. They allow us to simulate the movement or performances of components. They help us create a Bill of Materials (BOM), or they allow us to create an animation to show assembly instructions. We can even create drawings to document all our planning.

If we consider an Assembly to be a database, then we can consider all these outputs as different queries of the database.

Of course, a database is only as good as the data that is in it!

Before starting a design, it's a good idea to think about the outputs you require and think about the inputs you'll need to put in as you work.

The Assembly file

Simplistically, an Assembly file contains hyperlinks to your part files, some information about where your parts are in space and some information on how your parts relate to one another.

When you open an assembly file, Inventor will follow the hyperlinks, go out onto disk and look for the components, extract the geometry and load it up into memory.

Inventor will then consider all the relationships (Joints and constraints) that you have applied to see how the part files should be arranged.

For large assemblies (over 1000 unique files) – this can take some time.

If the part files that your assembly needs to load take up more RAM than you have on your PC, Inventor will need to switch some of the part files out into your page file on your hard disk.

When a file that the assembly is referencing is edited and saved. An update is triggered in the assembly. Inventor will check all the relationships in your assembly again to see how the change effects the design.

This is very powerful but can be extremely frustrating if a change to a component 'breaks' (Contradicts) the existing relationships in the assembly.

Express mode

To help, Inventor will automatically load any assembly which references more than 500 components in 'Express Mode'. In Express mode, Inventor saves the part graphics in the assembly file.

This means that Inventor doesn't have to load the file into memory to display it on screen. It also means that your assembly files will be a little larger on disk, because they contain more information.

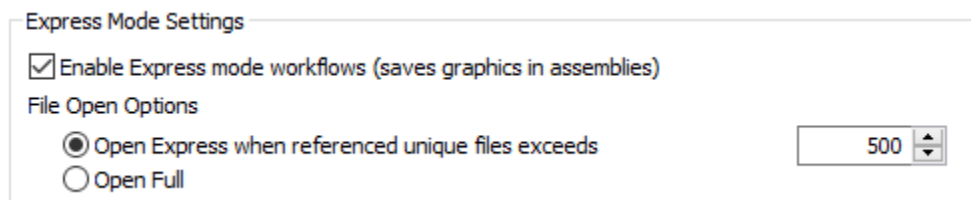
Express mode does limit the operations you can perform. For example, you can't toggle between 'Assembly' and 'Modelling' features in the browser, because the part features aren't loaded into memory.



To load the full assembly, navigate to:
Assembly (Tab) > Express (Panel) > Load Full (Button)

Tip: You can set your preferences for express mode, navigate to:

Tools (Tab) > Options (Panel) > Application Options (Button) > Assembly (Tab) > Express Mode Settings



File naming

Because your assembly file references lots of other files, it's very important that you don't rename files which are referenced by an assembly.

This would be like sending out your mailing address, only to change your house number!

It's also very difficult for Inventor if you have files which have the *same name*.

File resolution problems will slow down the loading of your assemblies. Good housekeeping will help prevent poor performance!

I'll give some tips for [file naming](#) later in the handout...

The Assembly Model Browser

The model browser in an assembly file helps you to navigate the model structure. An assembly file can reference other assembly files, part files and even CAD files from other systems.

Any item in an assembly can be referred to as a 'Component'.

The assembly file which contains your complete design is referred to as the 'Top-Level' or 'Main' Assembly.

References in an assembly file can be 'nested' An assembly file can reference another assembly file, which can reference another assembly file and so on. Part files are considered the 'lowest' level component in an assembly.

These relationships are often described as 'Parent-Child' relationships. The assembly is the 'Parent' of the component.

Parts can update, triggering an update in an assembly. Assemblies can't (usually!) trigger an update in a part. This avoids cyclical relationships.

Unlike the features in a part file, components in an assembly are not ordered. You can drag and drop components up and down the Assembly browser to create any order you like without breaking relationships.

In an Inventor Assembly model, Parameters and Features in one part can drive the dimensions or location of Features in another part.

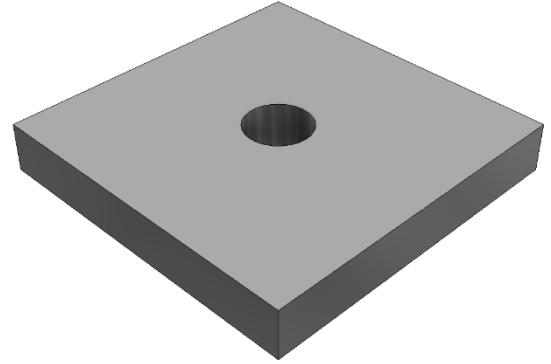
Features can also be created directly in the assembly file. This is limited to features that remove material and is intended to represent secondary machining operations that are applied to components after they are assembled.

Note: *The assembly structure is sometimes described as a tree. The top-level assembly is the root and trunk, the sub-assemblies are the branches and the parts are the leaves.*

Design Intent

If I gave you a model of a 50mm by 50mm by 6mm steel plate, with an M10 clearance hole through the middle of the face, and asked you to change the width of that plate – what should happen to the hole?

- Would we have one hole in the middle?
- One hole at either end?
- One hole in each corner?



Using feature-based parametric part modelling we can decide how our model will change, this is known as '**Design Intent**'.

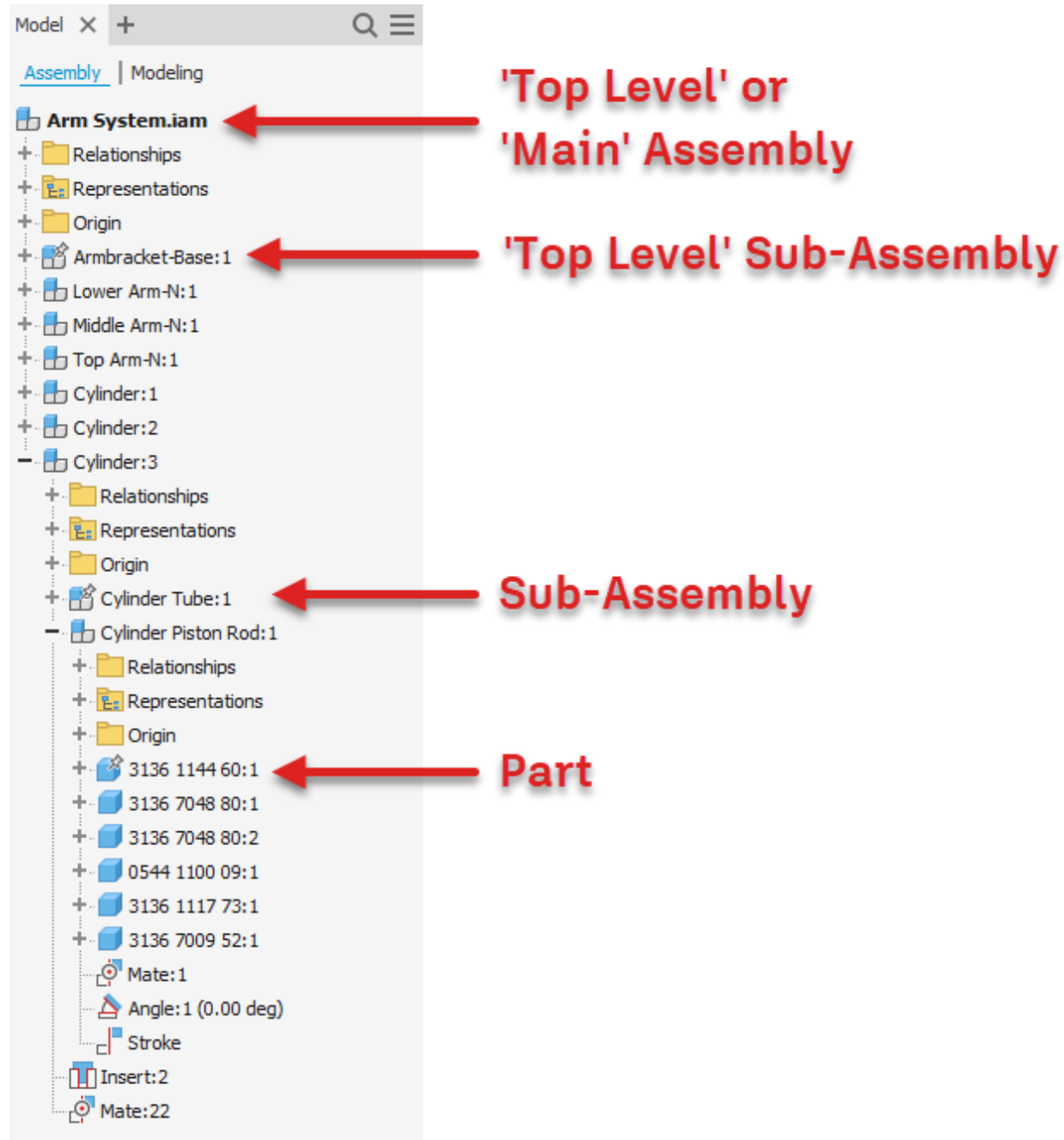
In assemblies we can build relationships between parts using parameters, relationships (Joints and Constraints), Adaptivity and Derive functions. This allows us to describe how the assembly should update when our parts update.

Navigating an Assembly

Use the assembly Browser to explore an Assembly file. Click on the '+' symbol to expand an assembly to see what is contained within it (the '+' is also known as a 'Node').

An Assembly that is contained within another Assembly is known as a 'Sub-assembly'.

Tip: 'Component' is a generic term that refers to both parts and assemblies.

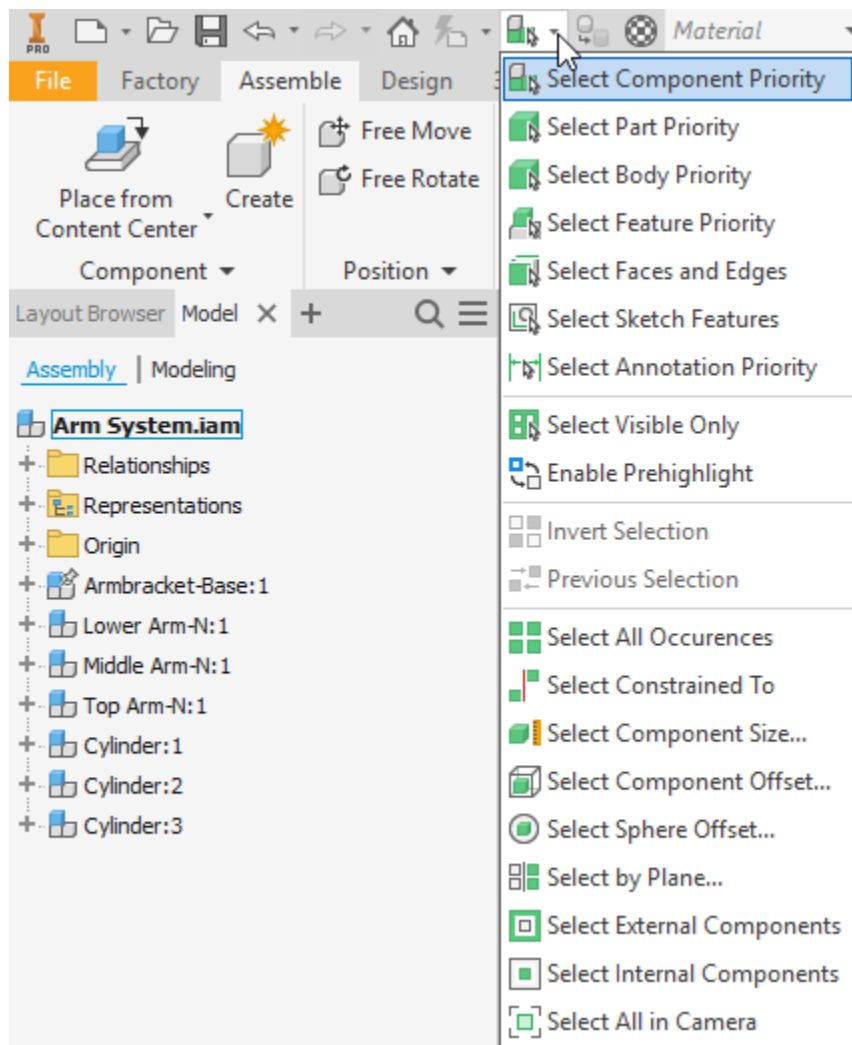


The screenshot shows the Autodesk Assembly Browser interface. The tree structure is as follows:

- Arm System.iam** (Main Assembly)
 - Relationships
 - Representations
 - Origin
 - Armbracket-Base:1 (Top Level Sub-Assembly)
 - Lower Arm-N:1
 - Middle Arm-N:1
 - Top Arm-N:1
 - Cylinder:1
 - Cylinder:2
 - Cylinder:3 (Sub-Assembly)
 - Relationships
 - Representations
 - Origin
 - Cylinder Tube:1 (Sub-Assembly)
 - Cylinder Piston Rod:1 (Sub-Assembly)
 - Relationships
 - Representations
 - Origin
 - 3136 1144 60:1 (Part)
 - 3136 7048 80:1
 - 3136 7048 80:2
 - 0544 1100 09:1
 - 3136 1117 73:1
 - 3136 7009 52:1
 - Mate:1
 - Angle:1 (0.00 deg)
 - Stroke
 - Insert:2
 - Mate:22

Tip: Use the Component Priority Filter tool (In the Quick access tool bar, green symbol) to switch between selecting Components (Top Level Parts or Sub-assemblies) Parts, Bodies and so on.

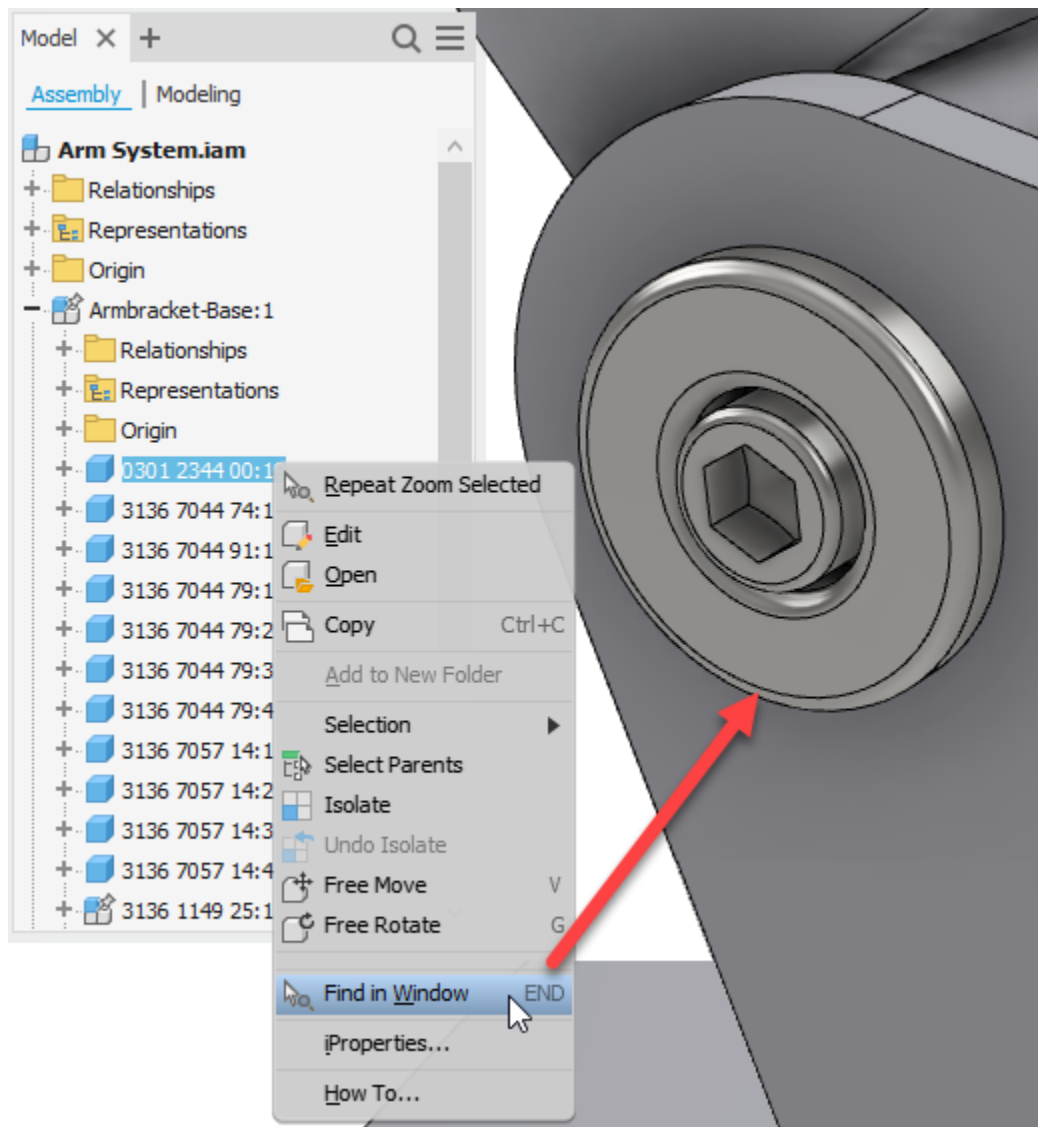
Shortcut **SHIFT+RMB** - Hold down the SHIFT key and right click to bring the filter up at your cursor.



Navigate to a Component in the Graphics Window

When you find a component that looks interesting in the browser, and you'd like to know where it is in the graphic window – Right mouse button (RMB) click on the component. From the right click menu choose 'Find in Window'. Inventor will zoom in on the component.

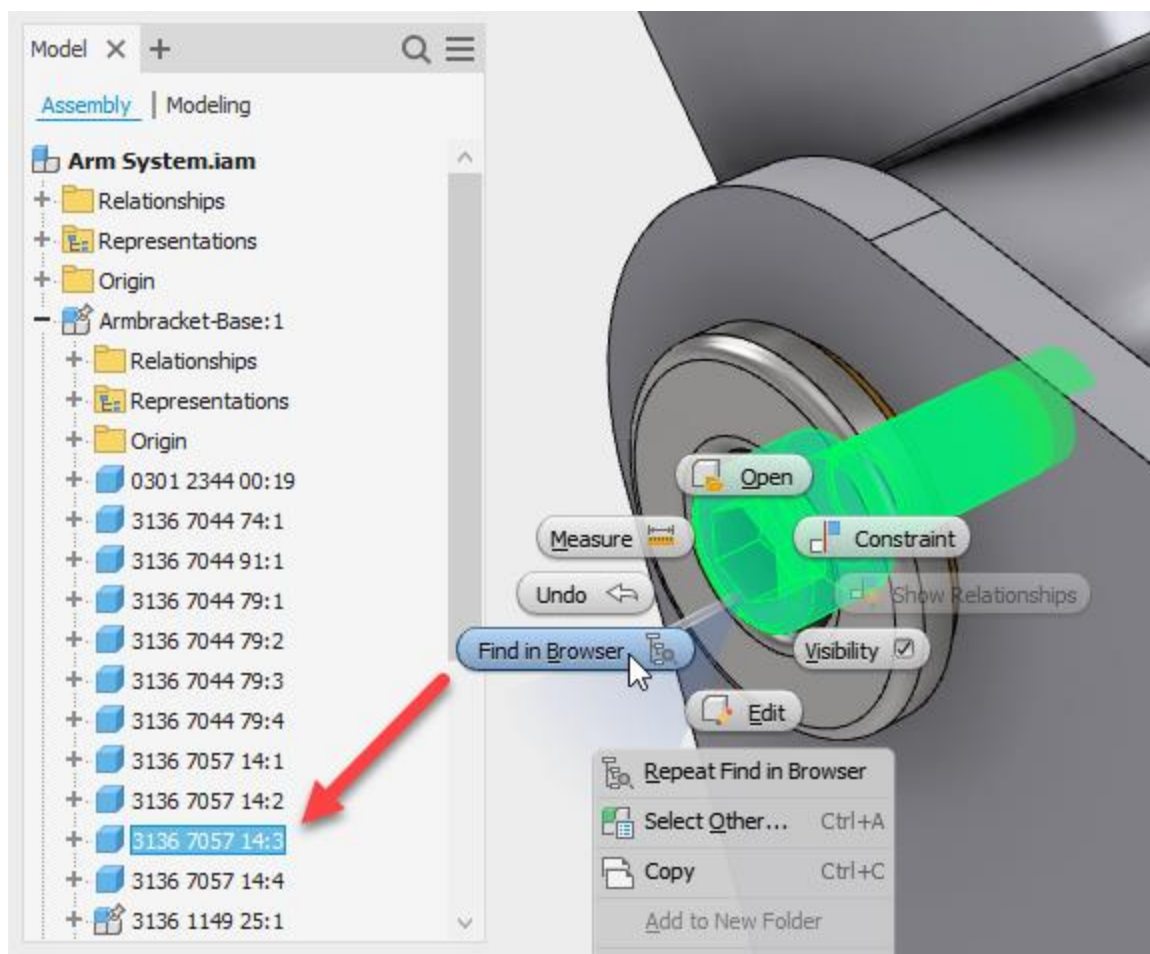
Tip: Alternatively, left mouse button (LMB) click on the component to select it, then press the 'end' key on your keyboard.



Navigate to a component in the browser

When you find a component that you like the look of in the Graphics window, and you'd like to find it in the browser, RMB on the component and choose 'Find in Browser' in the Marking menu.

The browser will expand to the component you are searching for.



Open Vs 'In Place' Edit a component

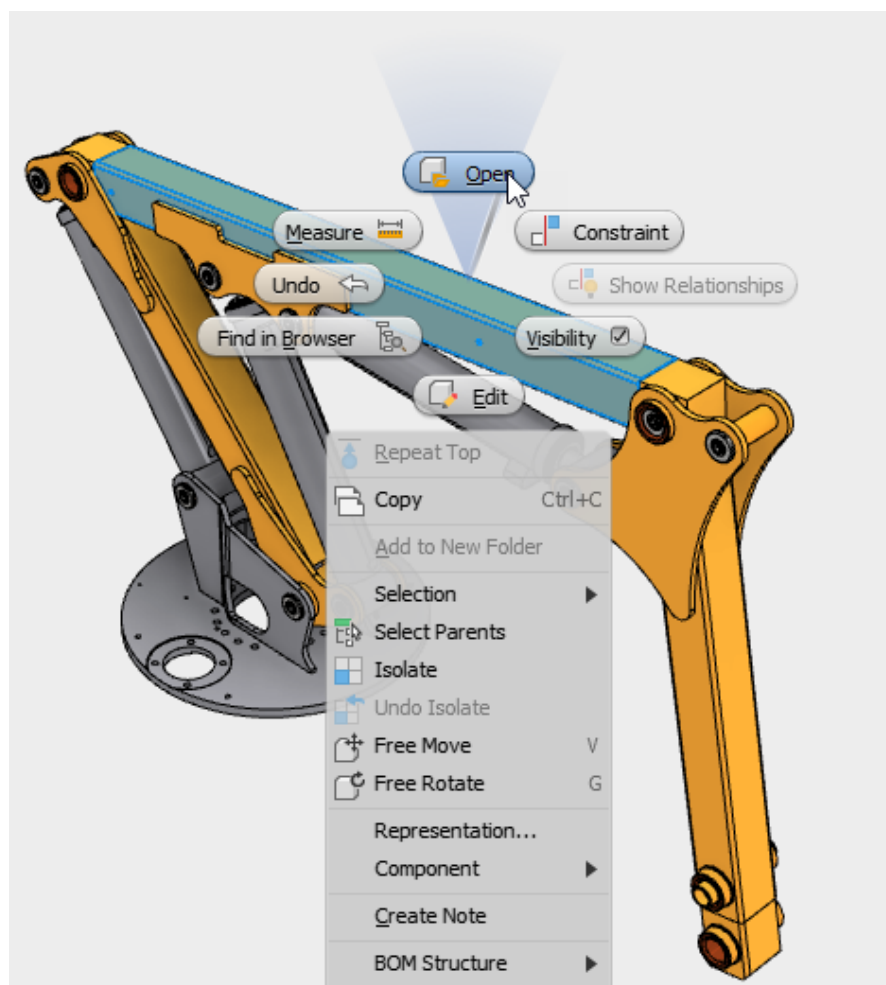
To Open a component for editing in another window. RMB on the component in the browser or the graphics window and choose 'Open'.

This will open the selected component in a sperate tab (window). You can make your edits and save. You can use the tabs at the bottom of the browser window to switch between the open assembly and component files.

Editing a component file in this way will cause the component to update inside the assembly immediately.



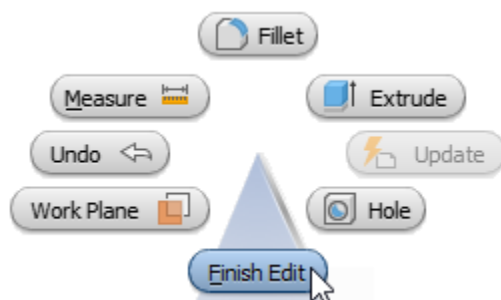
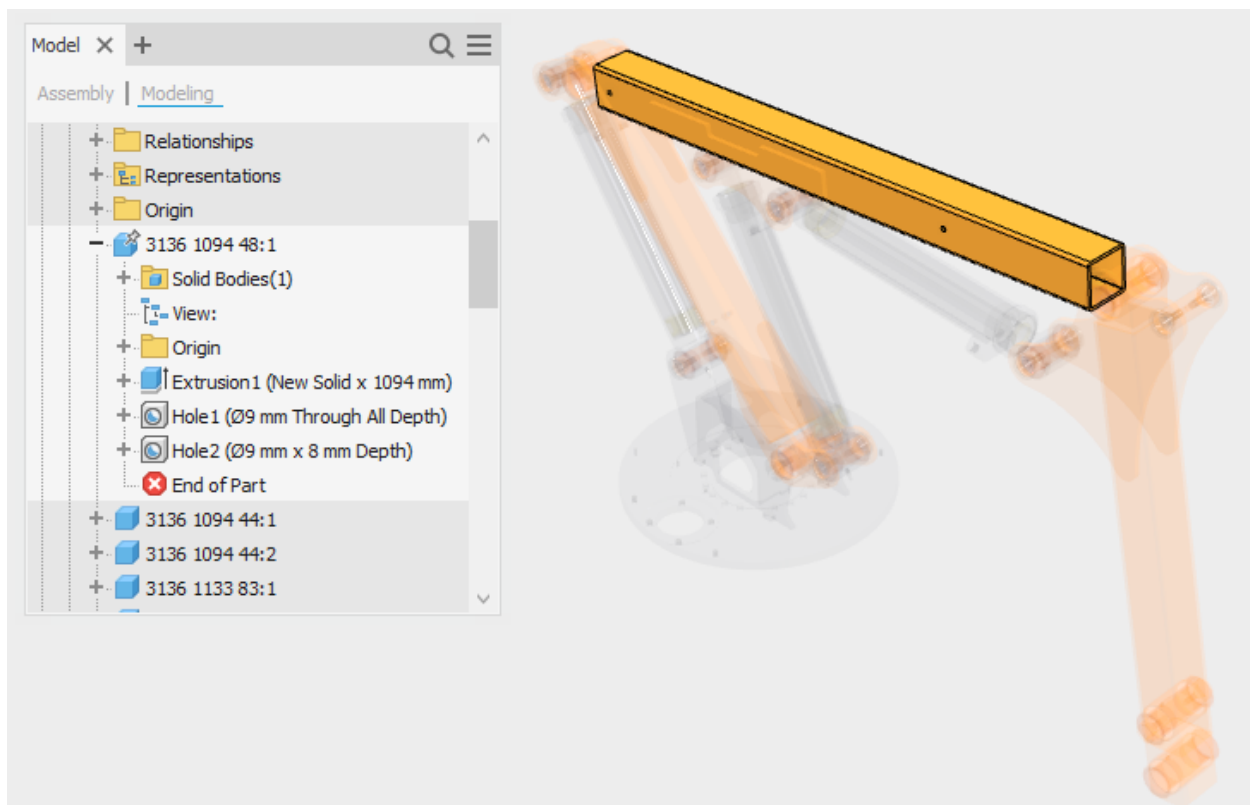
Tip: You may need to click the 'Global Update' button in the Quick Access Toolbar (QAT) to see the update (The Global update button looks like paper being hit by lightning).



To edit the component 'in place' (within the context of the Assembly) double left click on the component in the browser or the graphics window.

Note: During an in-place edit, the rest of the components will become translucent in the graphics window and will have a dark grey background in the browser. You will see the part modelling tools in the Ribbon.

In-Place editing is useful when you want to use '**Adaptivity**' to create relationships between features in different parts.



To finish an in-place edit, right click in some free space and choose 'Finish Edit' from the Marking menu.

Tip: When you need to perform multiple edits on components, you don't need to 'Finish Edit' and return to the top-level assembly each time.

Double left clicking on any component in the browser will finish editing the current component that you are in and open the next immediately.

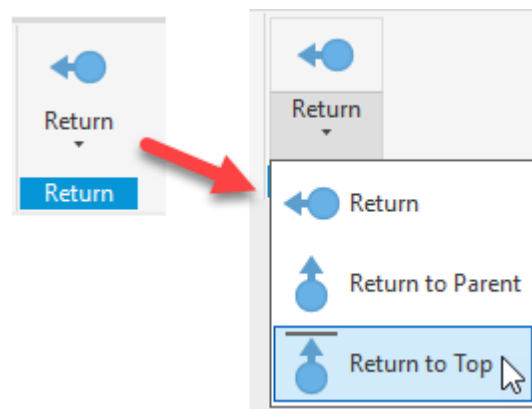
You might notice that the Return panel has a blue highlight? This indicates that you might want to use this tool next.

Navigate to:

3D Model (Tab) > Return (Panel) > Return (Button)

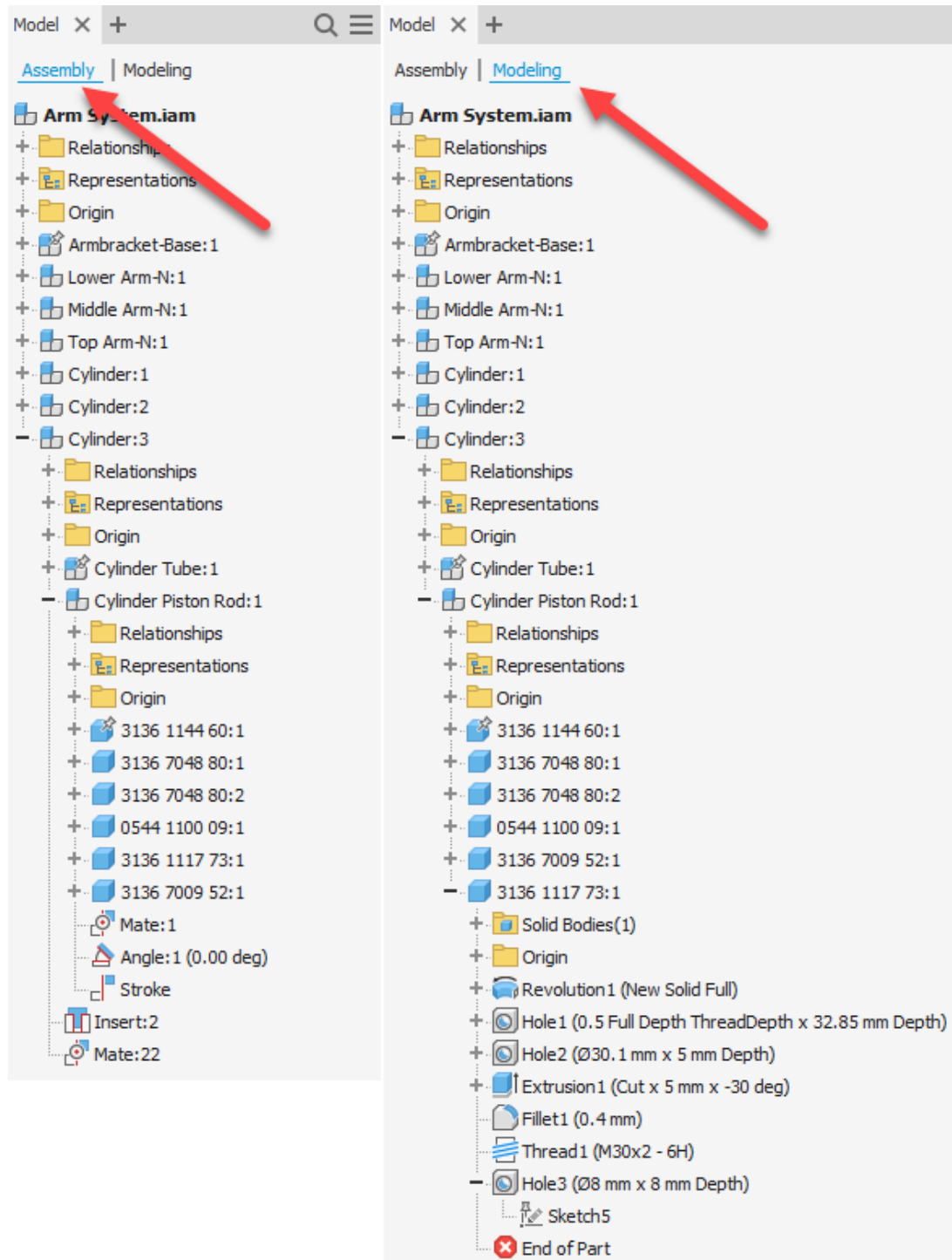
Clicking on the drop-down arrow reveals three options:

1. 'Return' – returns you to the last component that you were editing – retrace your steps.
2. 'Return to Parent' – will navigate you one step up the browser to the parent assembly which contains the file that you were editing.
3. 'Return to Top' – returns you to the top-level assembly. This is the same as 'Finish Edit'.



Assembly Vs Modelling Browser

The assembly browser shows only components and their relationships. You might think that you need to edit the part to see its features. In fact – you just need to switch to the 'Modelling' view.



Assembly relationships

Degrees of freedom

When we insert a component into an assembly. It is free to '*float*' around wherever it likes.

We consider this component to have Six Degrees of Freedom (DOF).

1. *Translate along X*
2. *Translate along Y*
3. *Translate along Z*
4. *Rotate around X Axis*
5. *Rotate around Y Axis*
6. *Rotate around Z axis*

We can fix a components position (remove all degrees of freedom) by 'Grounding' it.

To Ground a component, right click on it and chose 'Grounded'.

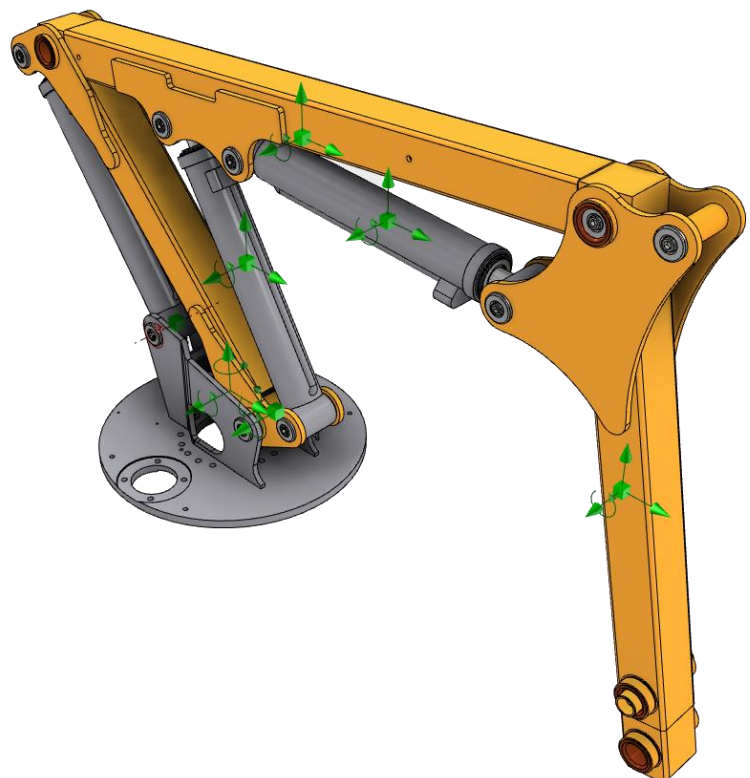
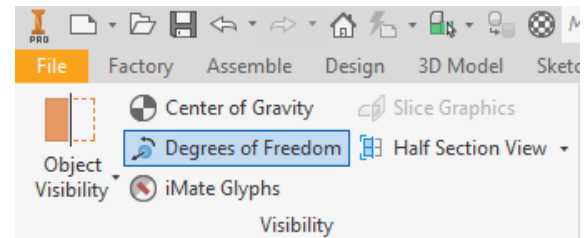
Tip: To show the remaining degrees of freedom of your components navigate to:

View (Tab) > Visibility (Panel) > Degrees of Freedom (Button)

Glyphs will show over each component showing their degrees of freedom.

As you place more constraints, degrees of freedom will be removed.

Click the 'Degrees of Freedom' button again to turn the D.O.F glyphs off.

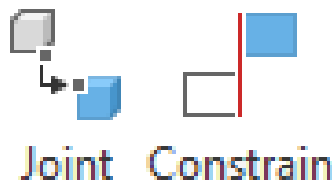


The Base component

Usually we ground the first component inserted into an assembly. This component is known as the 'Base' component. It is usually grounded so that its 0,0,0 or 'Origin' point matches with the Assemblies 0,0,0 point.

All other components are placed in relation to the base component, using assembly relationships.

The relationships used are either '**Joints**' or '**Constraints**'.



Assembly Joints

Joints create a relationship between two components that remove all degrees of freedom. A Joint can be edited to allow degrees of freedom.

Tip: Joints are great for quickly 'snapping' components together.

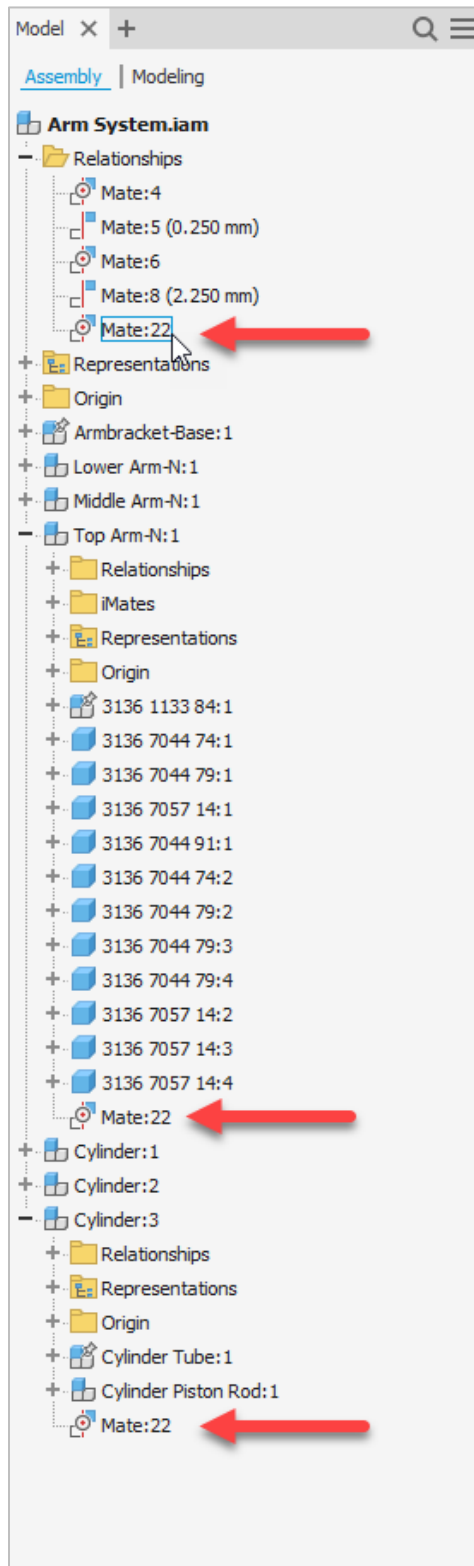
Assembly constraints

Constraints create a relationship between two components that typically remove one degree of freedom at a time. It typically takes three constraints to remove all six degrees of freedom.

Constraints include 'Assembly' constraints that are used for putting components together, and advanced constraints, such as 'Motion' or 'Transitional' constraints that coordinate movement between components.

Tip: Constraints are great when you need to create complex motion between multiple components.

Assembly relationships in the browser.

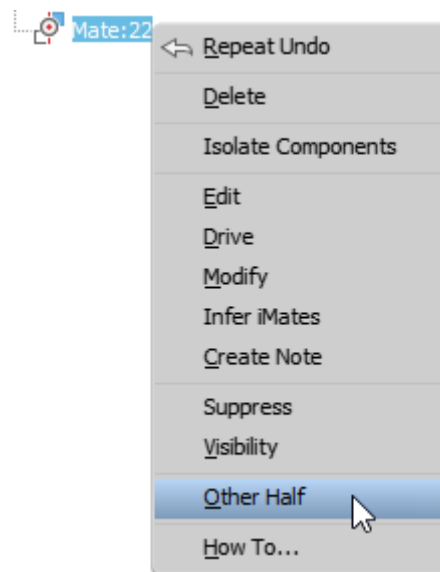


Assembly relationships are always shared between two components.

In the Assembly browser, you will find the relationships listed in three places.

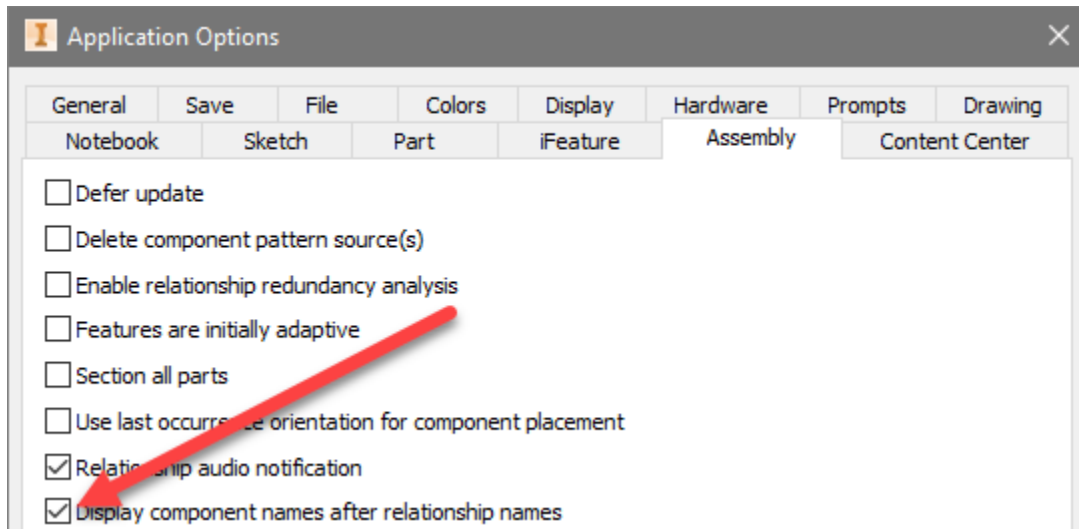
1. Under the first component referenced.
2. Under the second component referenced.
3. All relationships are listed in the **'Relationships'** folder.

Tip: Right click on a Joint or Constraint and choose **'Other half'** to find the other half of the relationship.



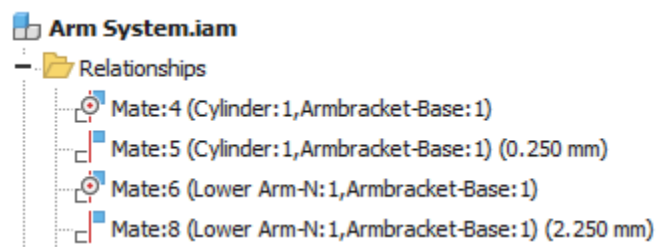
Tip: Display component names after relationship names, Navigate to:

Tools (Tab) > Options (panel) > Application Options (Button) > Assembly (Tab)



Tick the 'Display Component names after relationship names' check box.

Apply and close.



Visualizing Relationships

It can be difficult to visualize how the relationships are related to the components.

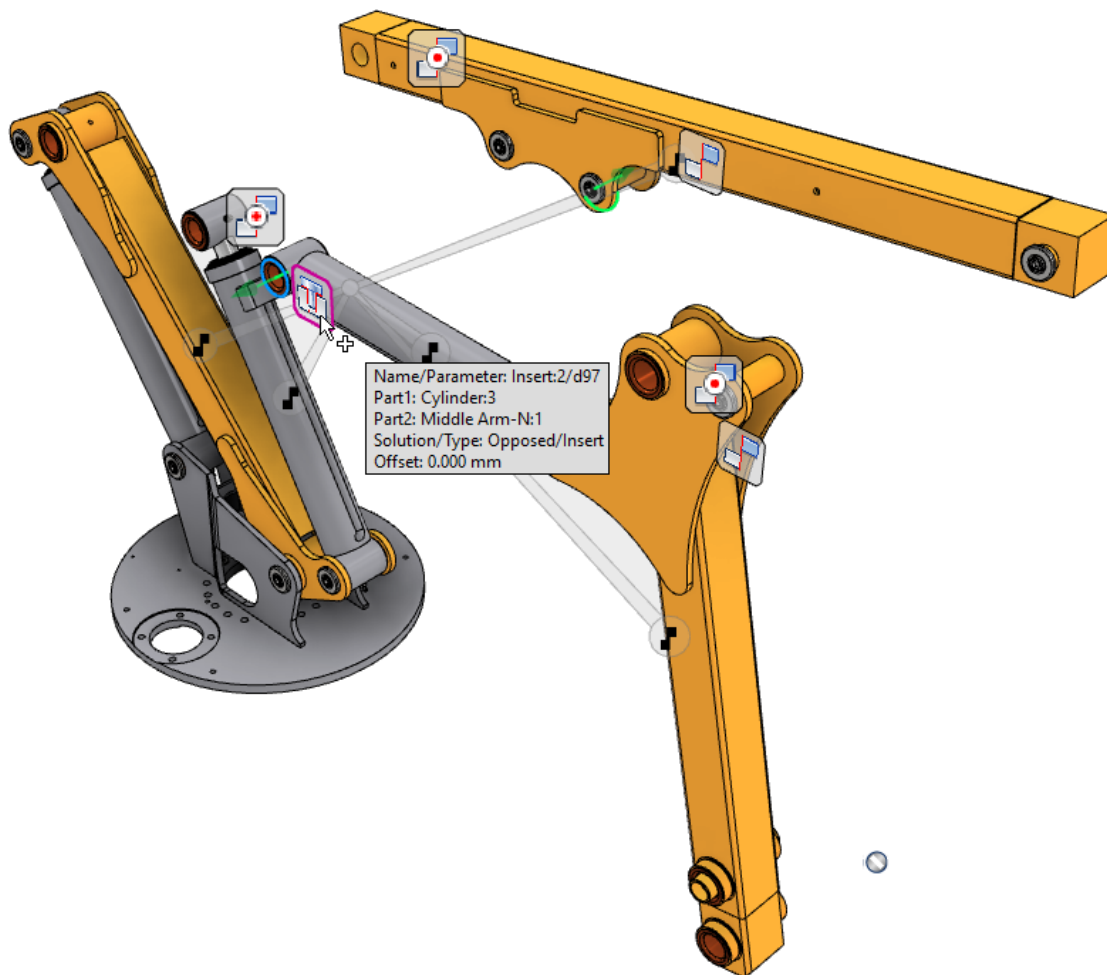
A simple tool to help visualize this is 'Free Move'. Navigate to:

Assemble (Panel) > Position (Tab) > Free Move (Button)

With the Free Move command active, left click and drag on a component. 'Rubber Band' lines will stretch between the components, indicating the components that are related.

Glyphs will pop up to indicate the relationships. Hover over the glyphs to highlight the features that the relationship is acting on and to get more information about the Joint or Constraint.

Filter your selection between components and parts to continue 'un-packing' the relationships.



Notice in the Quick Access Toolbar (QAT) that the 'Global update' button (It looks like paper being hit by lightning) is highlighted?



Click the button to reapply all the relationships and move the components back to their original positions.

Diagnosing an under constrained assembly

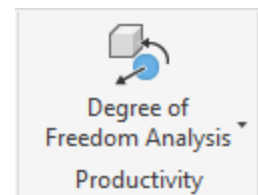
The simplest way to test for an under constrained condition is to left click on a component and drag it! If the component shouldn't move, and it does, you need more constraints!

(CTRL+Z to undo your movement).

Degree of freedom Analysis

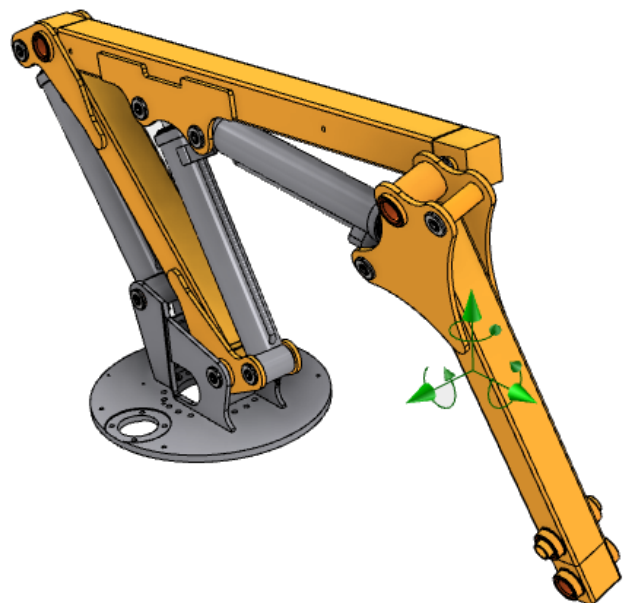
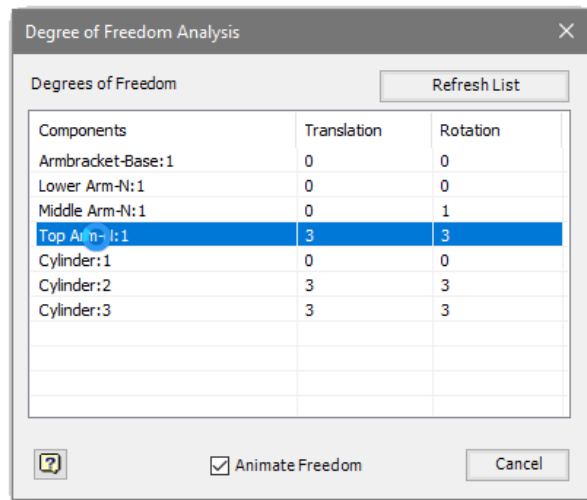
Another method is to use the Degree of Freedom Analysis tool. Navigate to:

Assemble (Tab) > Productivity (Panel) > Click on the drop down > Degree of Freedom Analysis (Button)



The degree of Freedom Analysis dialog will display the remaining DOF for each component.

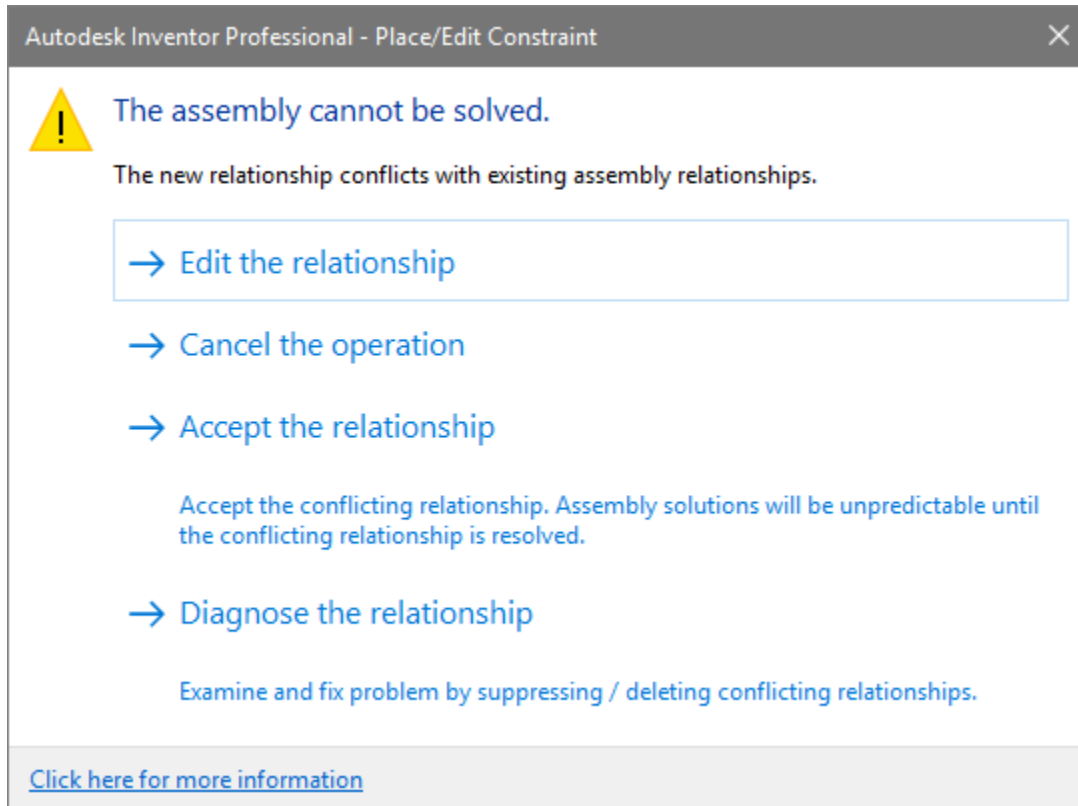
Ticking 'Animate Freedom' and selecting a row in the dialog will show the DOF Glyph and cause the component to 'Boogie' around its remaining degrees of freedom.



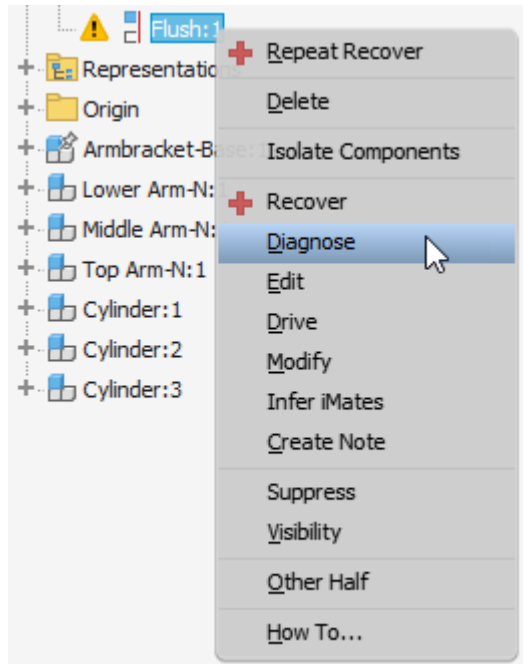
Diagnosing an over constrained assembly

Creating a Joint or Constraint that conflicts with your existing relationships will trigger an error.

Follow the options listed to fix the error.



Fixing conflicting constraints



Sometimes, editing a part file will cause a relationship to fail.

You will notice a 'Hazard' symbol next to the 'sick' relationship (It looks like a yellow triangle with a black exclamation mark). You may notice a red cross in the Quick Access Toolbar (QAT).

Right click on the component for the 'Diagnose' tool.

The 'Relationship conflict Analysis' dialog will open.

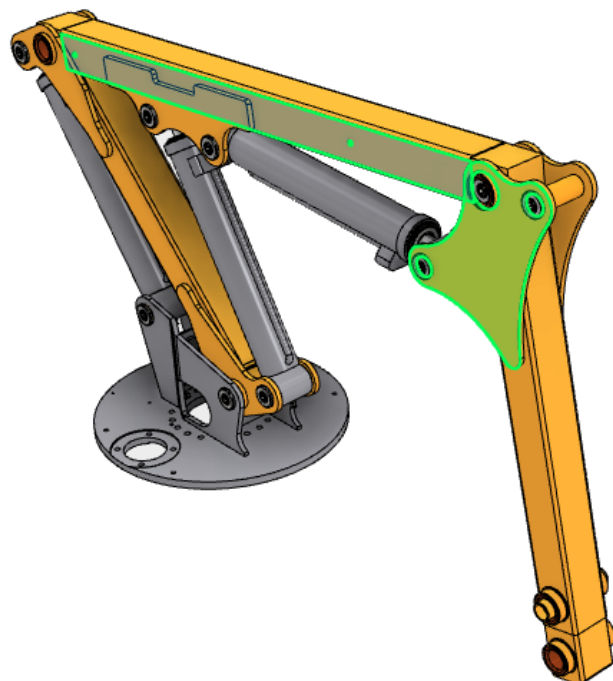
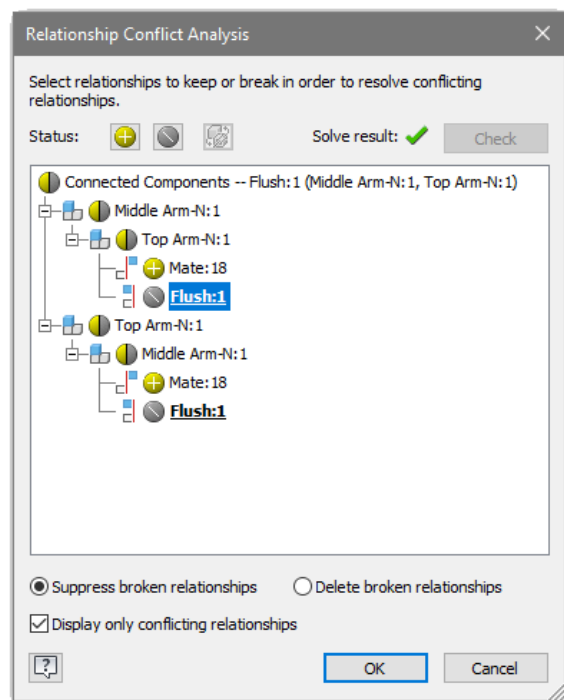
Tick the 'Display only Conflicting relationships' box to filter down to problem constraints.

Click on the yellow circle containing a plus symbol next to each relationship to suppress or un-suppress them.

Click 'check' to see if the problem is resolved.

Choose 'Suppress broken relationships' or 'Delete broken relationships'.

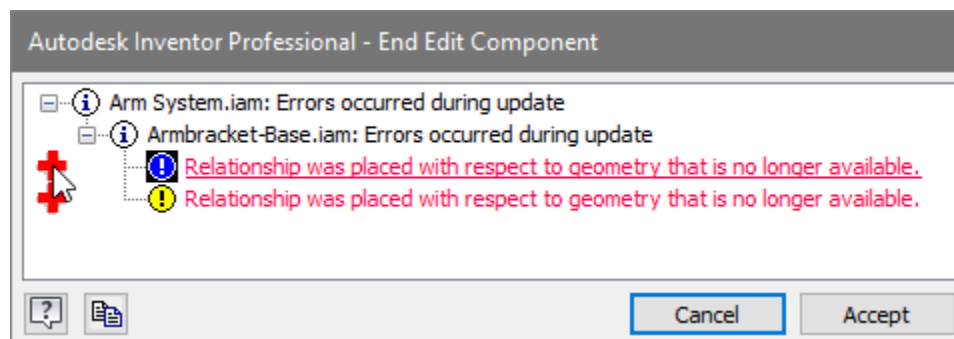
Click OK to complete the command and suppress or delete the problem Joints or Constraints.



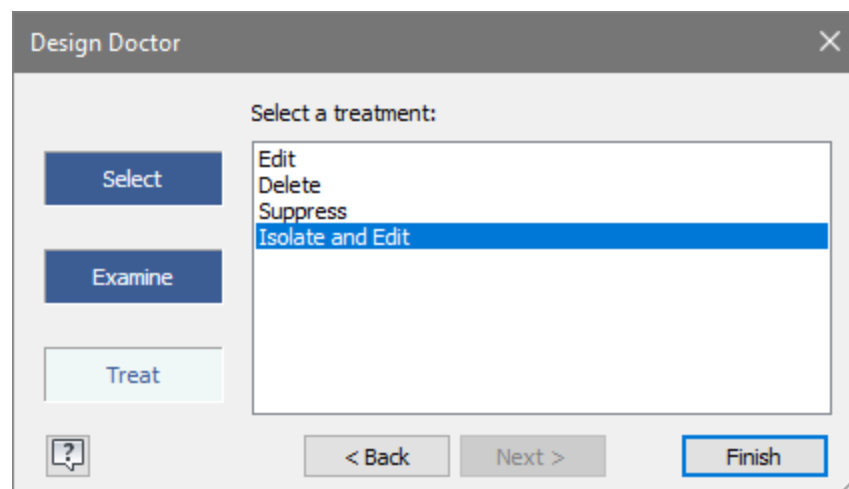
Fixing constraints that have lost their references

Sometimes, editing a feature on a part that is referenced by an assembly relationship can cause the relationship to 'Lose' the referenced geometry.

Click on the '+' symbols to expand the node and gather more information on the problem.

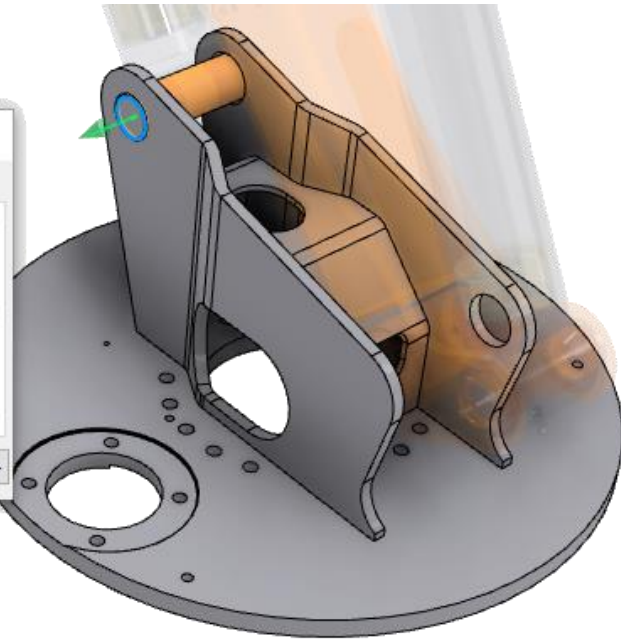
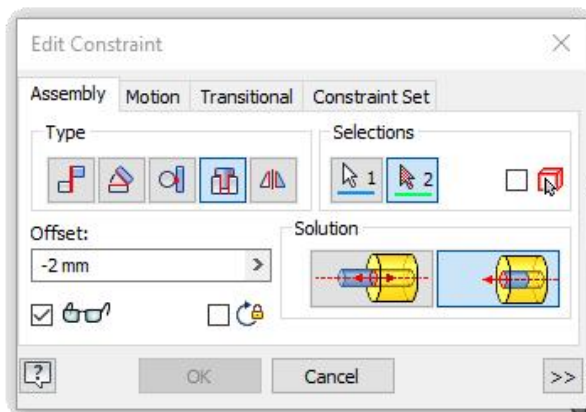


Click on the Red Cross to open the design doctor.



Click 'Next' to explore your options.

Clicking 'Isolate and Edit' will turn off the visibility of all other components and jump straight into the edit mode for the problem relationship, allowing you to re-select the problem geometry.

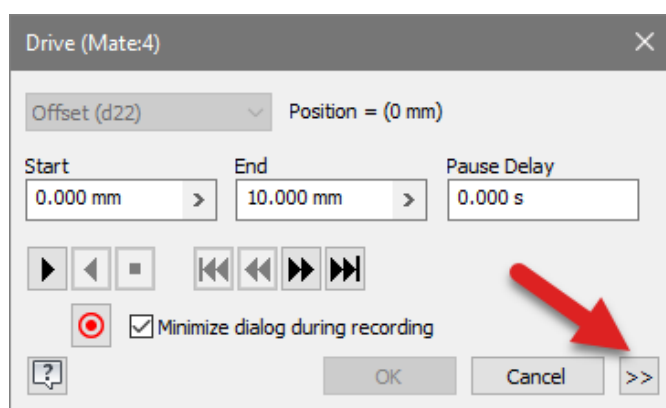


Driving constraints

If you've ever tried to describe complicated motion using assembly relationships - you'll know that clicking and dragging on a component to see how it moves can result in some unforeseen results.

However smoothly you drag your mouse, your PC and Inventor will interpret this as a series of stepped movements. At each movement it will try and calculate what relationships need to change. Sometimes it gets it wrong.

For a more predictable method use the 'Drive' tool to apply motion.



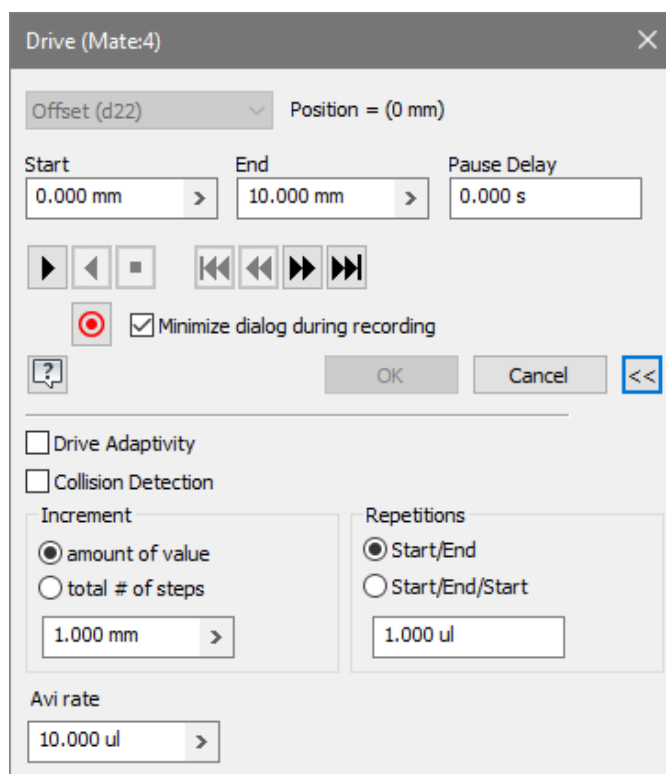
To drive a joint or constraint's movement, right click on the relationship and choose drive.

The drive constraint dialog will open, allowing you to add start and end values. Click on the black triangular 'Play' button to drive your constraint.

All constraints driven by the move will be honored and you should see the motion that you expected.

Tip: Click on the Red button to record your components being driven – you can share this video with your customers or colleagues.

Tip: Click on the double chevron '>>' button in the bottom right of the dialog to see more options – including 'Collision Detection' - the ability to stop the motion at the point that two components clash.



Reducing the number of constraints needed

We all learn to build models in Inventor from the 'Bottom up'. The bottom up method describes how we build each part file individually, before placing the parts into an assembly and constraining them in place.

This technique is great for small assemblies (100-500 unique parts) but can increase the overhead we are placing on Inventor and our hardware as our assemblies get bigger.

Relationships – constraints and joints – will always be needed when we want to simulate motion in our assemblies. However, most of the relationships we create are for assembly – they only need to relate the parts to one another.

So – what techniques do we have available to us that can reduce the number of constraints required?

Note: *These techniques are not mutually exclusive, you can combine them to manage your design intent.*

Derive workflow (Multibody or Skeletal modelling)

Derive workflows are commonly used for 'Top Down' modelling. This is where we model the system overall first, then create more detail as the design process continues.

A common 'Top Down' modelling technique is to use create multiple bodies inside a single part file. This is helpful, because we can easily create features that effect multiple parts (more on this later).

Each [body](#) will eventually become a part in our final design.

We use the 'Make components' tool, which will create a new part file from each body. Each part file contains a derived link back to the original [multibody](#) 'Master' part file.

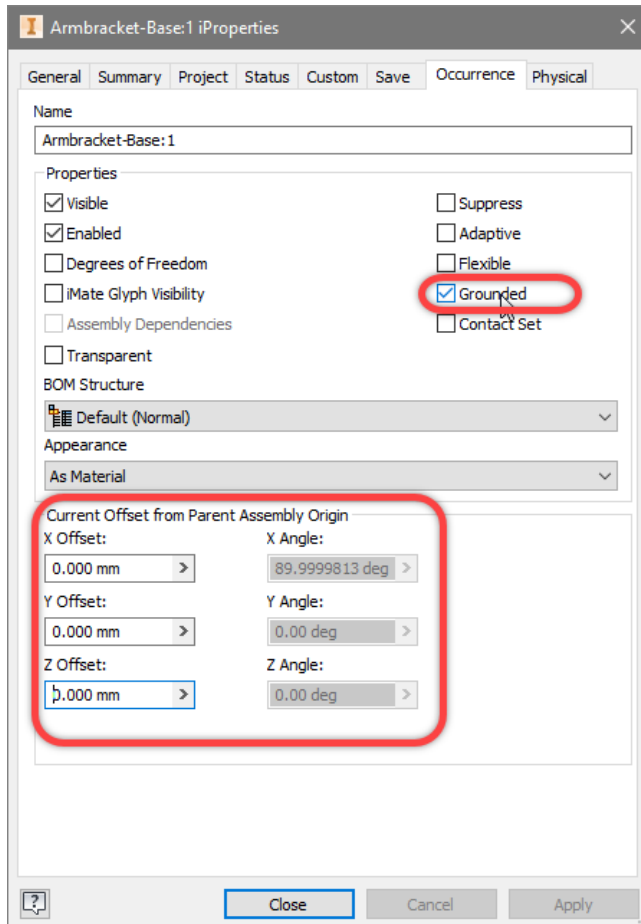
Because each body is modelled in its correct location, we don't need to use constraints to position the derived parts in the assembly.

Instead each part is grounded in its location*. To change the position of the part, we go back to the Multibody and edit it. Changes to the multibody Master part file will propagate through to the assembly.

**If we want to add mechanical motion, we can un-ground the components that move and add joint or constraints to define the movement.*

You will find more on [Top down Modelling](#) in the additional links at the end of this handout.

Place by Coordinates and Ground



This technique is useful when you have created sub-assemblies and you want to create a layout to show where the subassemblies will be placed on site.

This technique makes use of an iProperty that is only available for components that are placed inside an assembly.

In this method we ground the components to reduce the number of constraints required.

To make sure that the component is in the right location, we use the components 'occurrence' iProperties to set the X, Y and Z coordinate.

In an assembly file, right click on a component and choose 'iProperties'. Select the occurrence tab. Use the X, Y and Z input boxes to place your components.

Notice that you can toggle the 'Grounded' state from here.

Advantages: No constraints (No constraints to go wrong!)

Disadvantages: You can't set the rotation of components from the occurrence iProperties

Inventor in the construction Industry

(See next page) UCS Constraint sets are great if you are working on a construction project using a common origin. Often on a construction project, the site origin is miles away (or Kilometers away!). This is not helpful to Inventor, which prefers to model within a 99.9 meter cube around the Origin (0,0,0).

You can create a top-level assembly, which only has a UCS work feature in it, to represent the distance to our work package from the site origin. Next place a UCS in your assembly model at the corresponding location.

Place your assembly into your 'Site' assembly and constrain the two UCS's together. You can design within your assembly model, when you want to export the design for coordination open your 'Site' assembly to see your design in its coordinated location.

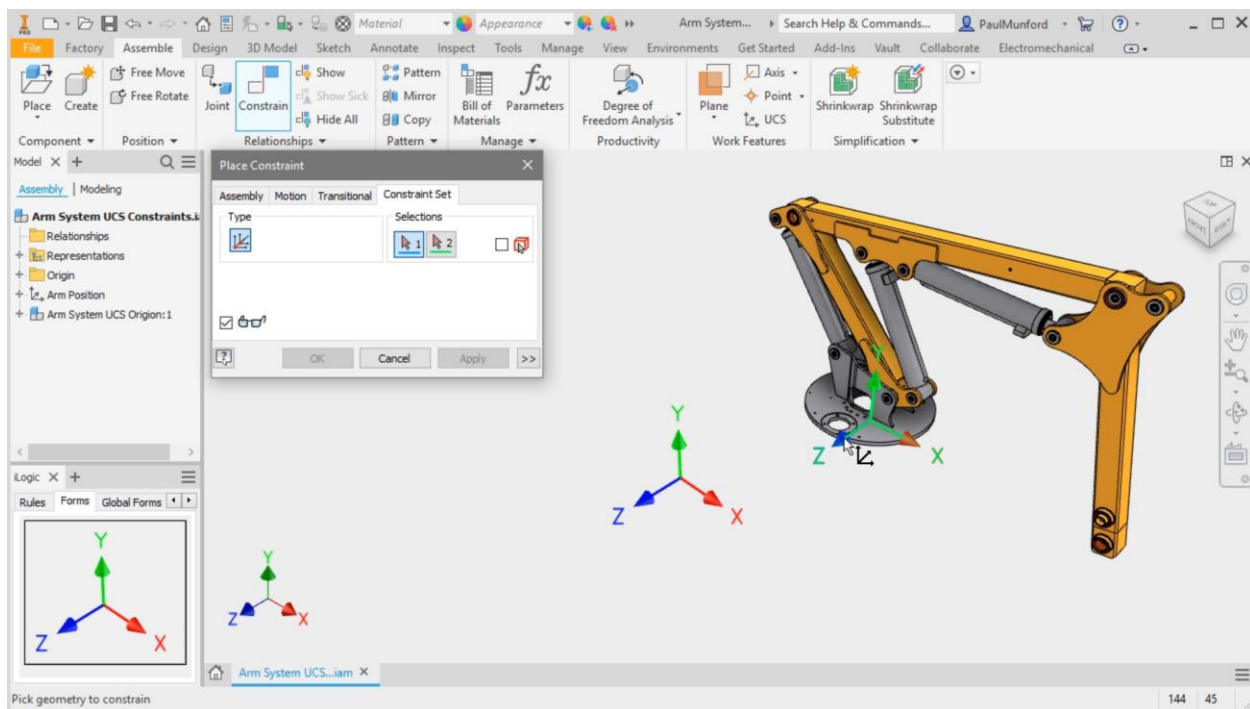
UCS and Constraint Sets

This technique is useful when you can coordinate your design around a common origin.

A 'UCS' Work feature (User Coordinate System) is created in the Assembly file to define the common origin.

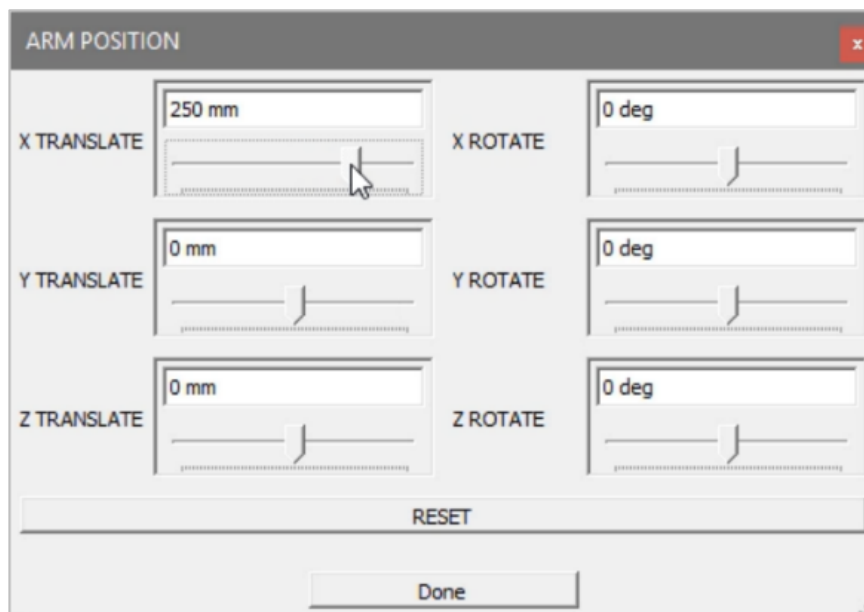
UCS's are placed in component files to define their relationship (distance and rotation) from the common origin.

When the component is placed into the top-level assembly, the UCS's can be 'Snapped' together using a 'Constraint set'.



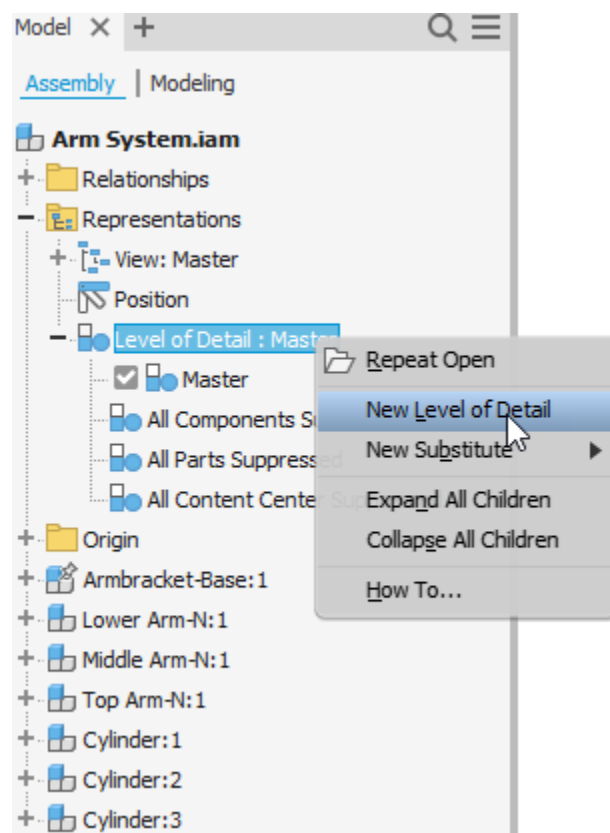
Tip: UCS work features work really nicely with iLogic forms ([click here for more on iLogic](#)).

1. Insert a UCS Work feature
2. Open the parameters manager
3. Rename the six parameters that have been created to define the UCS work features position – e.g:
 - a. X_t , Y_t , Z_t – Translate along the X,Y or Z Axis
 - b. X_r , Y_r , Z_r – Rotate around the X,Y or Z axis
4. Create a new iLogic form
5. Drag your renamed parameters onto the form
6. Set the control type to 'Slider'



The screenshot shows a software interface titled "ARM POSITION" with a close button in the top right corner. The interface is divided into two main columns of sliders. The left column contains three sliders labeled "X TRANSLATE", "Y TRANSLATE", and "Z TRANSLATE". The right column contains three sliders labeled "X ROTATE", "Y ROTATE", and "Z ROTATE". The "X TRANSLATE" slider is currently set to "250 mm", while the others are set to "0 mm" or "0 deg". Each slider has a visual representation of its range and a central knob. At the bottom of the form, there are two buttons: "RESET" and "Done".

Shrinkwrap LOD



Let's discuss a method of reducing file size in your assembly and minimising the number of relationships.

This method uses a feature in Autodesk Inventor called 'LOD' (Level of detail). You'll find the 'Level of Detail' control in the browser, in the 'Relationships' folder.

Before we learn about 'Shrinkwraps' let's refresh our knowledge of LOD's.

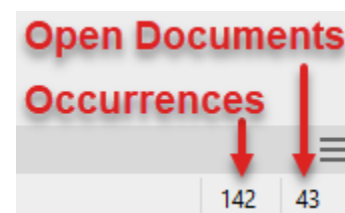
LOD Representations

An LOD representation saves the suppression state of the components in our design. When a component is suppressed, it is taken out of memory. This gives us the ability to control which components are loaded into memory at any one time.

You can keep an eye on how many components are loaded into memory at any one time by checking the indicator in the bottom right hand corner of Inventor's User Interface.

The first figure is 'Occurrences' (Copies), the second figure is 'Open Documents' (All open the documents in the current session, not just the assembly you currently have open).

Tip: If you'd like to try creating an LOD, I recommend that you create the LOD representation first, then start suppressing components.



You'll notice that the following LOD representations are in every Assembly file by default and cannot be deleted.

- **Master** (Everything loaded)
- **All Components suppressed** (Everything suppressed)
- **All Parts suppressed** (All Parts suppressed, all Assemblies Loaded)
- **All Content center suppressed** (It does what you think it does)

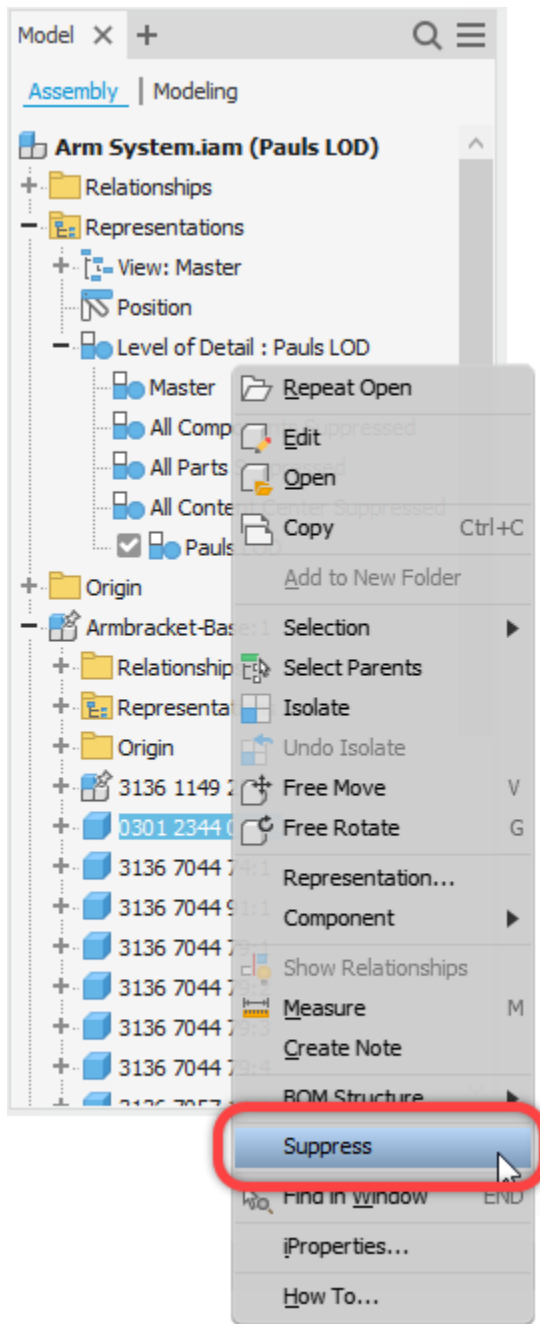
To create your own LOD rep', right click on the 'Level of Detail' node and pick 'New Level of detail'.

A new level of detail node will be added, and you can re-name it however you like.

Tip: To activate an LOD rep' double left click on the node's icon, or right click on the node and chose 'activate'.

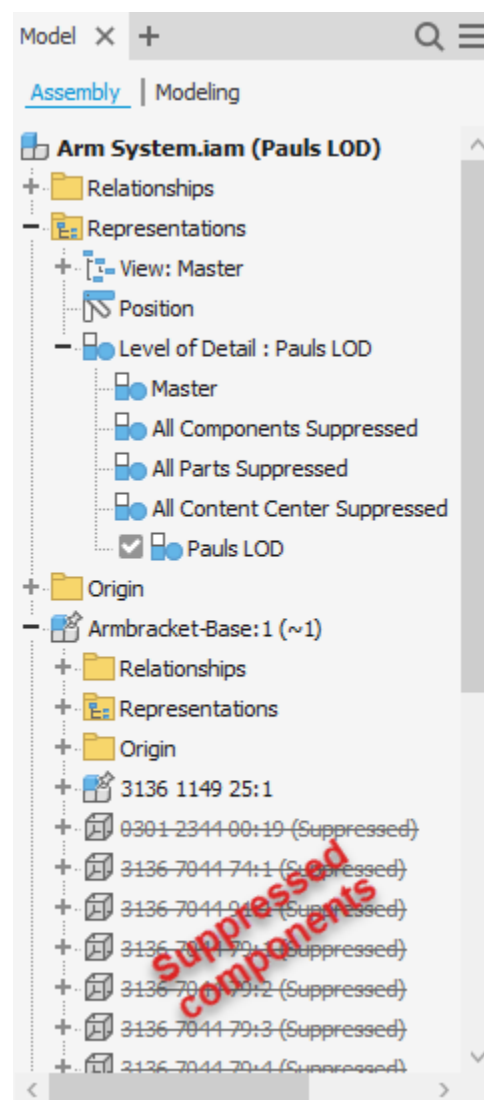
Activating different LOD reps will load different subsets of your design into memory.

Note: your BOM data and relationships will not be affected.

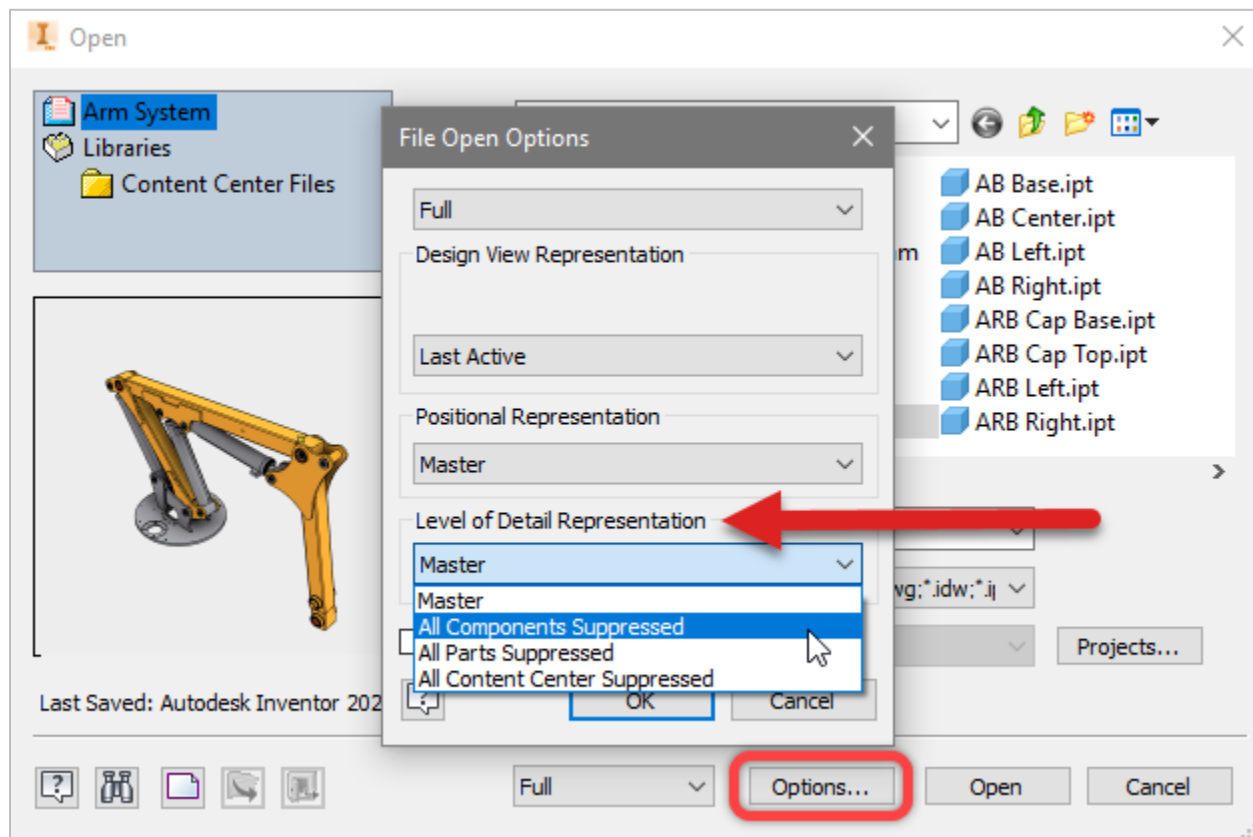


With your new LOD rep' active, right click on any component and chose 'Supress'. You'll notice that the 'Occurrence' and 'Open Document' counts will drop, and the component will disappear from the graphic window.

The components node will become grey, and there will be a line through the component's node name.



Tip: To open an Assembly with an LOD active, click on the 'Options' button in the Open dialog.



Note: NEVER reference more than one LOD representation into a drawing. From Inventor's point of view, each LOD rep is a separate design.

If you reference more than one LOD into a drawing – inventor may load components into memory twice! For drawings, it's recommended that you stick to View Representations instead.

Shrinkwrap LOD

So – LOD rep's (Representations) are a great tool for managing the number of components loaded into memory so that you can focus on a subset of components at a time.

How are Shrinkwrap LOD's different?

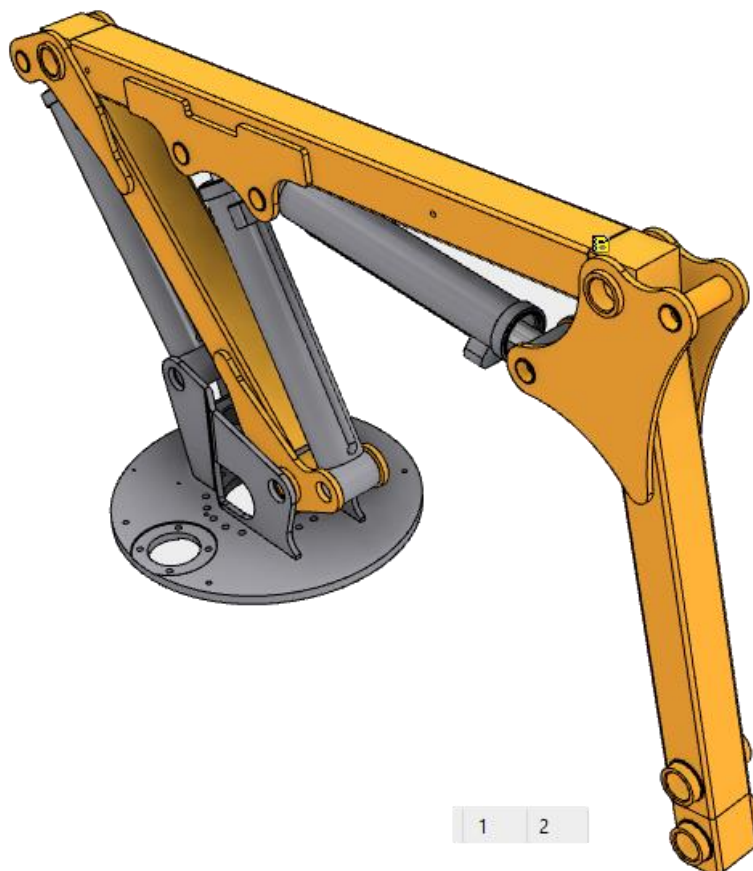
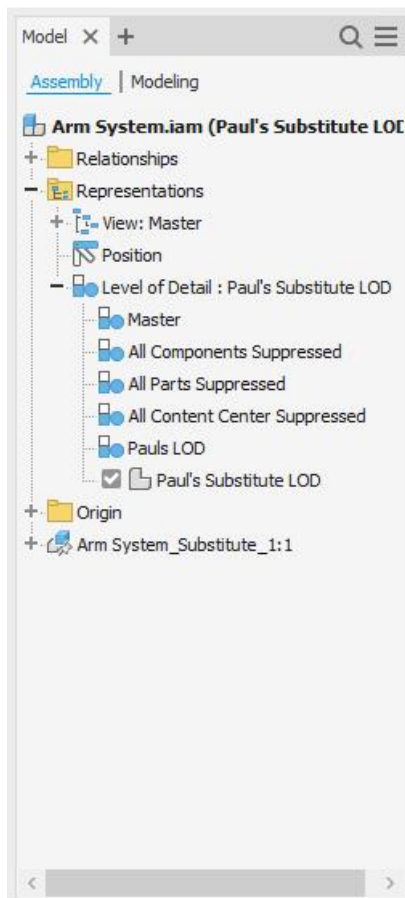
A Shrinkwrap LOD uses the derive toolset to create a part file which contains a reference to the Assembly. Each component in the Assembly becomes a body in the part file.

This works in almost exactly the opposite way to the [Top-Down, 'Multi-Body, Master Part' technique](#).

The components in the assembly will be replaced by a single component know as a 'Substitute'. The substitute component is the 'Shrinkwrapped' version of the assembly.

You have now reduced your open files to two (The substitute part file, and the assembly file), but you retain the geometry for visualization purposes.

Note: *your BOM data and relationships will not be affected.*



Because the geometry in your substitute component is derived, it will update when you update any of the components in the assembly.

Activate the Master LOD to reload all components into memory.

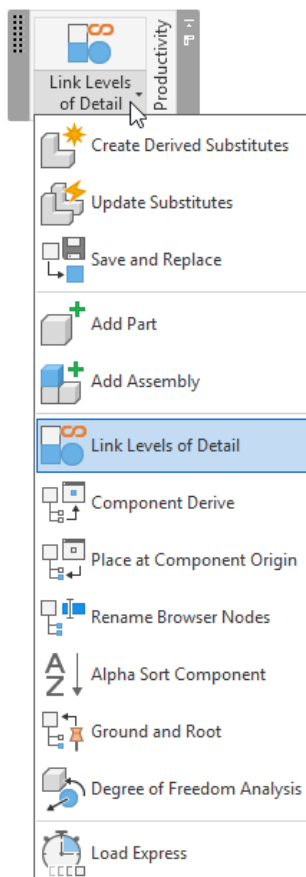
To create a Shrinkwrap LOD, right click over the Representations > Level of detail: Node and chose 'New Substitute' > Shrinkwrap.

Follow the prompts to simplify your shrinkwrap (derived) substitute component.

Link LOD

Shrinkwrap LOD's are extremely useful when you are creating Layouts. Instead of having to insert a 10,000 component subassembly into your design each time, you can insert the Shrinkwrap LOD version containing one component.

Your 10,000 component version is still available to you for manufacturing data, and any changes you make will update the substitute component referenced by the shrinkwrap LOD.



If you are intending to use shrinkwap LOD as part of your workflow, I'd like to show you one last tool. This tool is found in the Assembly productivity tools. Navigate to:

'Assemble' (Tab) > 'Productivity' (Panel) > Click on the black drop-down arrow > Link Levels of Detail.

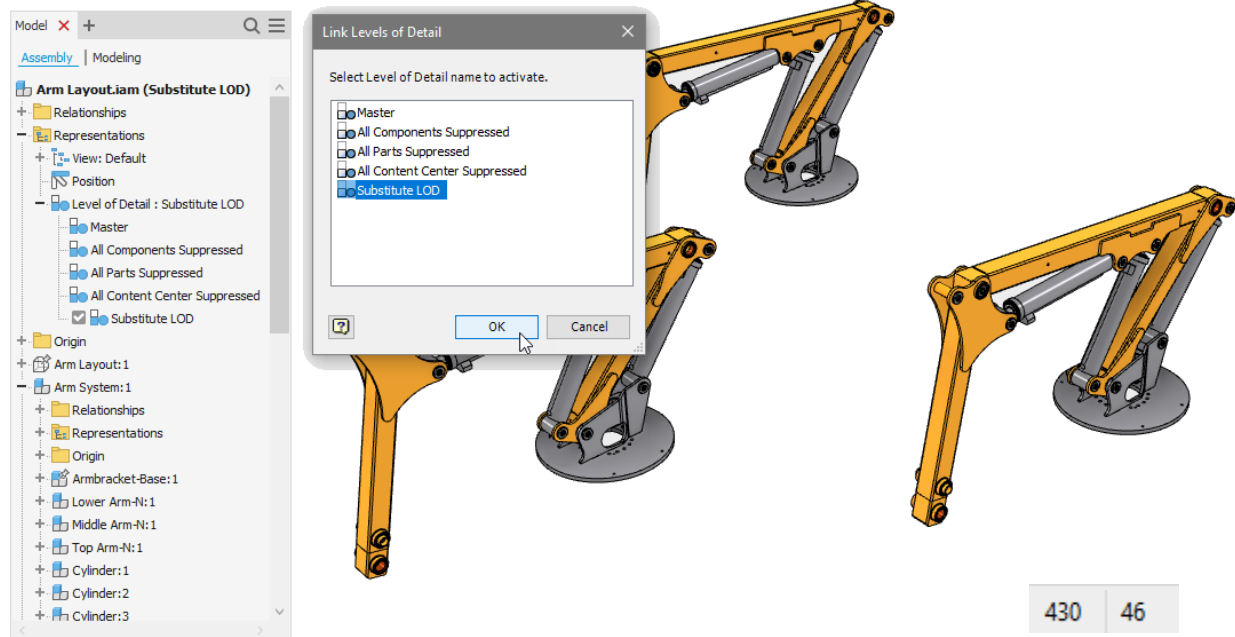
This tool will link an LOD in the top-level Assembly (It can be a regular LOD representation, it doesn't have to be a shrinkwrap) with any LOD's contained in the sub-assemblies that has the same name.

Therefore, activating your top level LOD will trigger a switch in your sub components, switching out all the parts and components for your shrinkwrap substitutes...

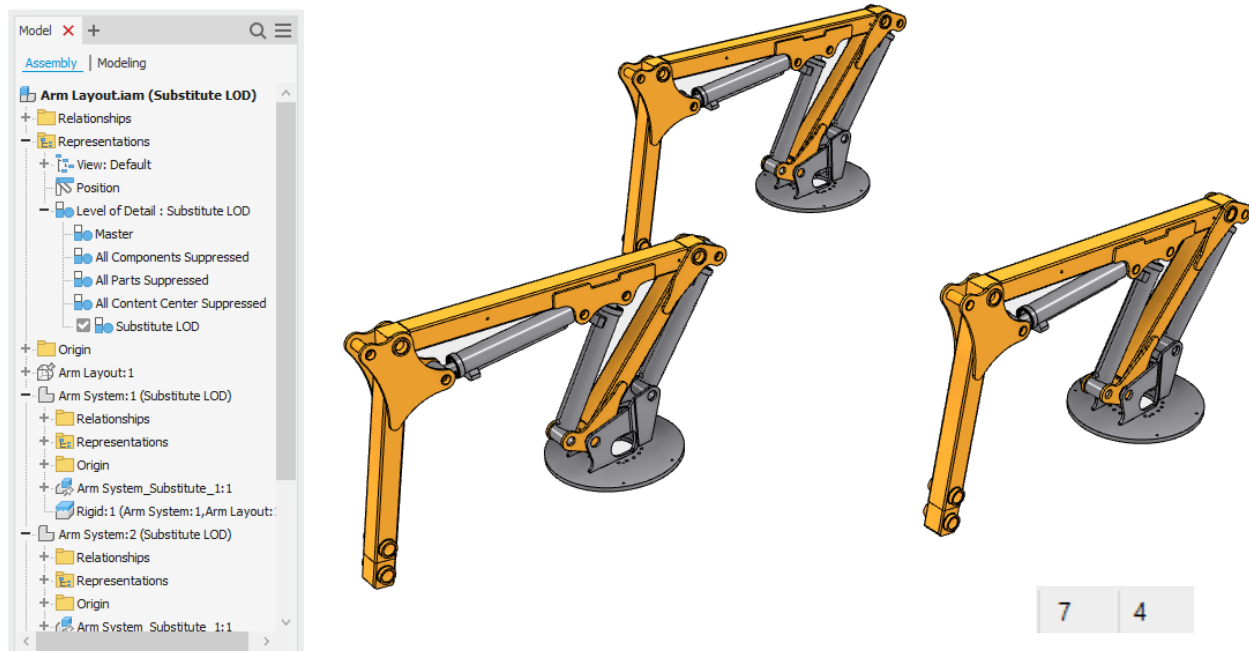
It's a beautiful thing.

Activate the Master LOD to load all components back into memory.

Before Link Level of detail'.



After Link Level of detail' and LOD rep activated.



Feature Relationships

A big advantage of using a parametric CAD system is the ability to have a feature in one part drive the dimensions of a feature in another part.

This helps bake our design intent into the model and helps prevent errors which arise when changes of size are made to the design.

Adaptivity

Adaptivity is a toolset inside Autodesk Inventor that allows us to create part files in context and relate features on those parts to existing features inside the assembly.

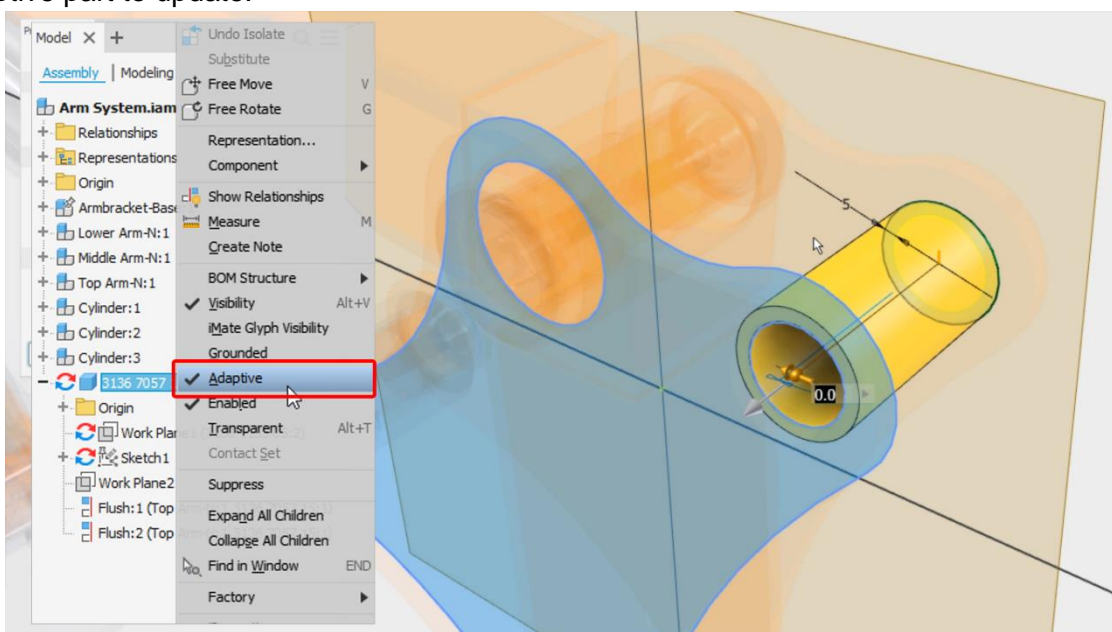
Technically – this could be called ‘Middle out’ modelling, because we aren’t driving the design from the bottom (individual files) or from the top (one master file).

Adaptive modelling is very useful when you need to model quickly and intuitively within the context of the assembly. It is great for early stage work or when you are prototyping your designs.

If you’ve tried using Adaptivity in your designs, you’ll know that adaptivity can ‘fight’ (create contradictions) with constraints, causing some unexpected behaviour.

So – if you want to progress your model from prototype to production model, I recommend that you (temporarily) turn adaptivity off.

You can turn Adaptivity off by right clicking on the Adaptive component(s) and un-ticking ‘Adaptive’. If you need it – you can turn Adaptivity back on in the same way, which will allow the Adaptive part to update.



Linking Parameters

Have you ever noticed the Button in the Parameters manager that is marked 'Link'?

This button allows you to link to parameters in an external file such as an Inventor Assembly of part file, or an *Excel spreadsheet.

You can link a part file to a part file or an assembly file to a part file, but you can't link a part file to an assembly file – because this might create circular references.

For this reason, [Parameter](#) Linking can't be used to drive 'Top Down' design (but we can do this with iLogic instead).

**Back in the day, we used Excel spreadsheets A LOT to drive configurations, because you can create far more complex equations in a spreadsheet than you can in Inventor's parameters manager.*

BUT - linking to Excel does come with a performance hit. To reference the Excel file, Inventor needs to open Excel in the background and load your spreadsheet (How long does it take to open your spreadsheet when you open it in Excel? Inventor will take at least this long!).

These days we recommend iLogic to drive your configurations, because it works in-process with Inventor and doesn't require opening up another application.

Derive (Skeletal or Multibody modelling)

Earlier, we discussed the derive tools for reducing the number of constraints required in our assembly.

We mentioned that the derive tools can also be used to coordinate the size of features that effect multiple components.

These techniques involve creating our design intent in a 'Master' file, before deriving out components so that we can create a BOM and individual part drawings.

Skeletal modelling

In this technique, we create all the geometry we need in one file, then derive the geometry out into multiple part files using the 'Make component' or 'Make Part' tools. We create the 3D geometry in the part files.

This technique can also be referred to as 'Layout' modelling and works really well with Inventors Sketch Blocks.

A lot of people find that conceptualising their design in 2D before moving to 3D helps them to plan their design before committing to a high level of detail.

Multibody modelling

In this technique, we model the majority of or design (Excluding Library and Supplier components) in a single part file.

Each part in the design is modelled as a separate body. When we are done modelling, we use the 'Make Components' tool to create part files. Each part file contains a derived link back to the 'Master' part. Making a change to the Master part will update the derived parts.

This technique is helpful when conceptualising complex geometry and has the advantage that it is easy to create features that effect more than one body/component.

Derive workflow

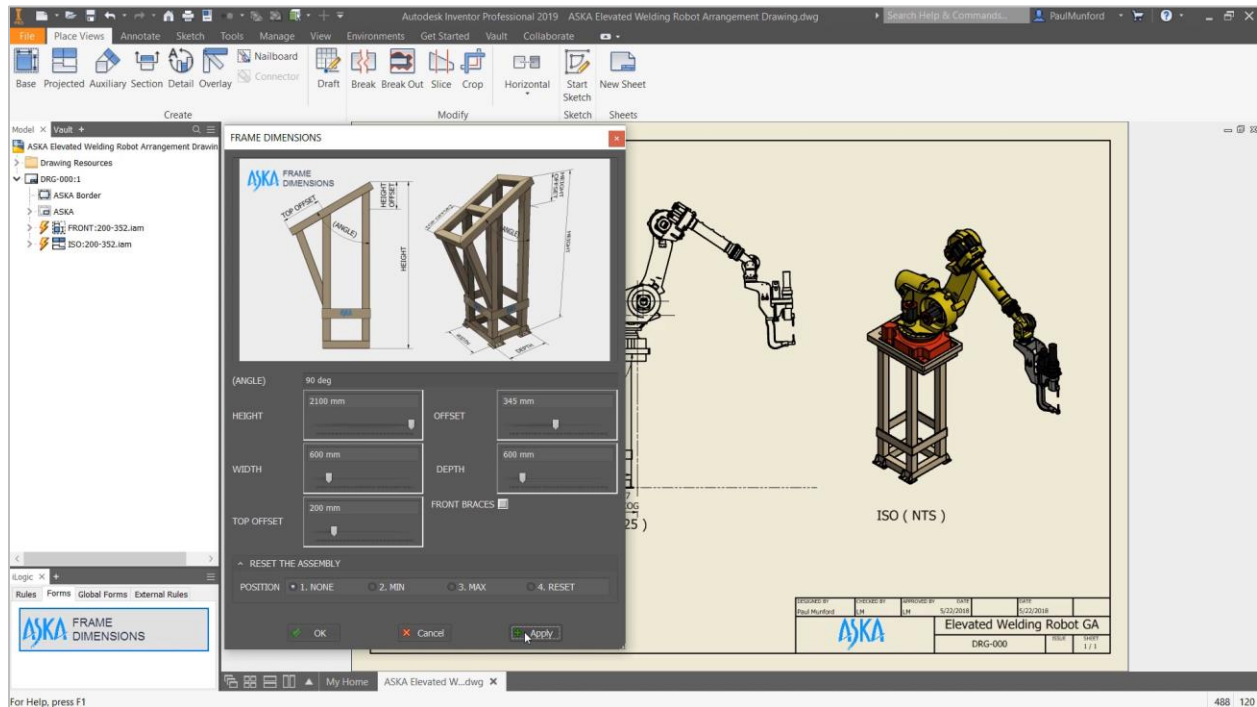
It's worth noting that the derive workflows can come with a performance hit. You'll notice that each part containing a derived reference is the size of the part plus the size of the master part that it is referencing!

You can suppress the link to the master part to help with this. You can use iLogic to automate the suppress and unsuppressed all links to derived components in your design.

Here is a link to a post on the Autodesk Inventor Customisation forum that has code which you can play with:

<https://forums.autodesk.com/t5/inventor-customization/howto-suppress-link-to-derived-parts-inside-assm/m-p/2130712#M23066>

iLogic



iLogic is a true 'Top Down' technique for creating relationships between components, because it allows us to pass parameter values down from the Top-Level Assembly to the subcomponents.

It has become my 'Go To' method of assembly design for configurators, because it doesn't have the performance overhead of derived components.

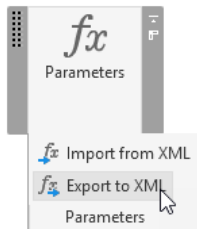
Using iLogic, we can create a form at the top level of the design (or even in a drawing) that describes our design intent. Our users will find it very clear to understand what they need to do to change the design.

The form edits the parameter values and an iLogic rule passes the parameter values down to the components.

iLogic top down workflow

1. Create Parameters in part file (This could be a template file)
2. Export parameters
3. Import parameters into the assembly file
4. Create an iLogic rule
5. Insert the named parameters into the rule
6. Copy and paste until the rule reads:
Assembly parameter = part parameter
7. (Optional) [Create an iLogic form](#)

Import and Export parameters

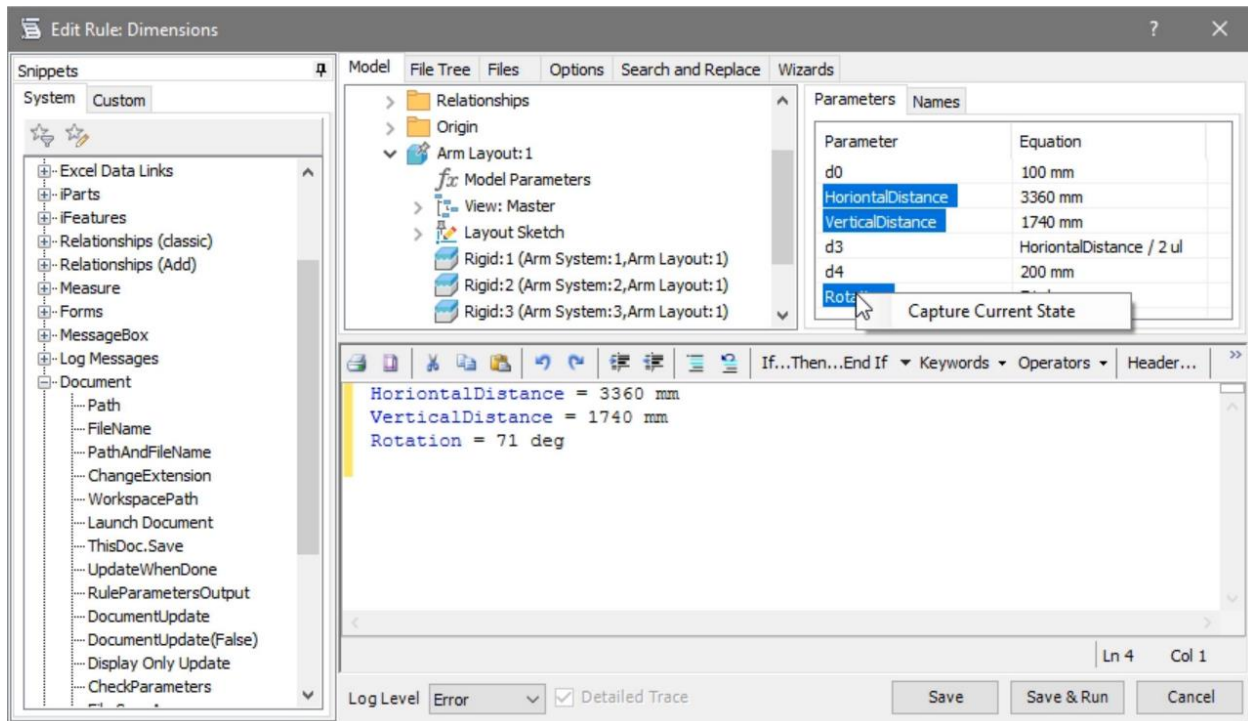


You will find the 'Import parameters' and 'Export Parameters' tool hidden under the parameters manager button.

Tip: Tick the parameters you want to export as 'Key' before you export them. In the Export dialog box, click on the 'Options' button and tick the 'Key Parameters only' option.

The Logic Rule

Use 'Capture current state' to copy the assembly and part parameters into your rule.



Tip: Any Parameter name written in **BLUE** will be constantly monitored. If the value of this parameter changes - the iLogic rule will run.

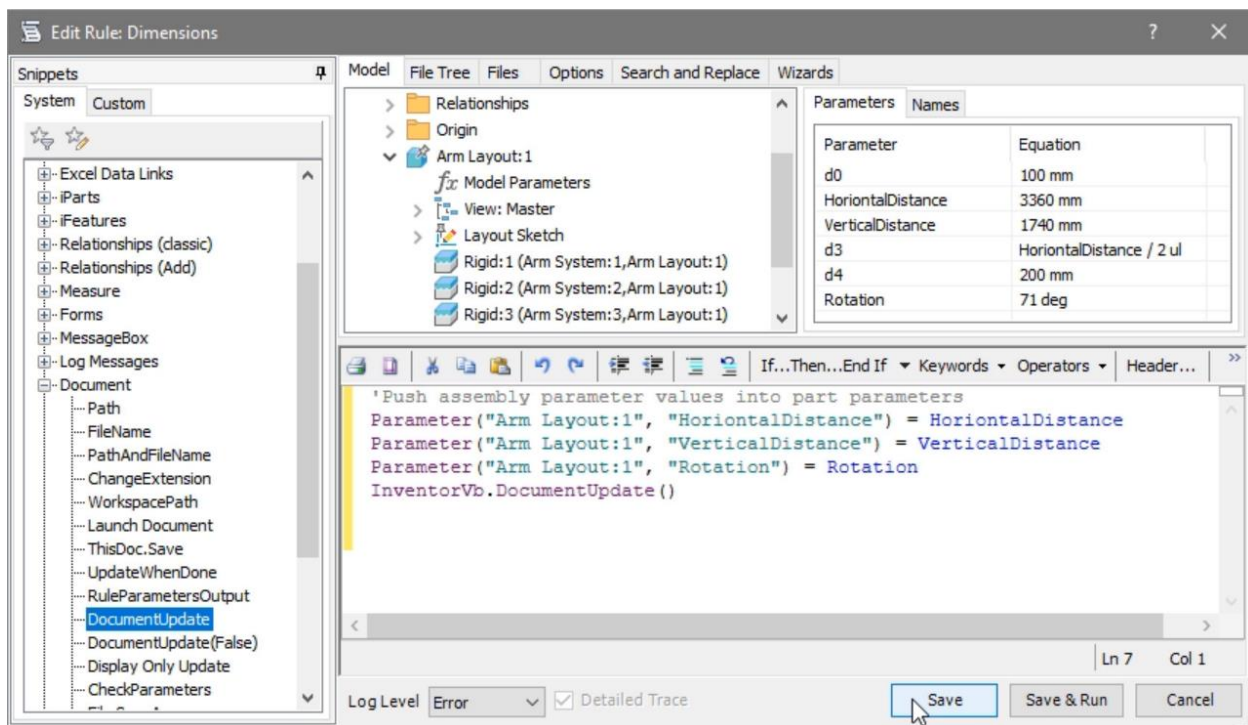
The rule reads:

'Keep an eye on the assembly parameter called "HorizontalDistance". If the value of this parameter changes, look for a part file called "Arm Layout". In that part file, look for a parameter also called "HorizontalDistance". Copy the value from the assembly parameter into the part parameter so that both are equal.'

The line:

`InventorVb.DocumentUpdate()`

Is the same as clicking 'Global Update' in the QAT.



If you'd like to know more about Top Down design – check out this class from Autodesk University Online:

Driving Inventor with the top down – Alternative assembly modeling techniques.

<http://au.autodesk.com/au-online/classes-on-demand/class-catalog/classes/year-2013/product-design-suite/ma2604#chapter=0>

Model template checklist

Use this checklist to make sure that you have a robust template for parts, assemblies, and presentations.

General

- ☐ Create a part naming schema
- ☐ Create an assembly naming schema
- ☐ Create a Parameter naming schema
- ☐ Create a Feature naming schema

Application options

- ☐ Turn 'Show Extended Name' on (Parts)
- ☐ Turn 'Display Component Names after relationship names' on (Assemblies)

Part template (And Sheet Metal Template)

- ☐ Set the Viewcube orientation
- ☐ Set the default view
- ☐ Re-Name origin Planes

Optional

- ☐ Edit Body and Surface prefixes
- ☐ Create a UCS base feature
- ☐ Create named parameters
- ☐ Create a Layout sketch

Assembly template (and Weldment Template)

- ☐ Set the Viewcube orientation
- ☐ Set the default view
- ☐ Re-Name origin Planes
- ☐ Set up BOM columns

Optional

- ☐ Set default material
- ☐ Set default standard for 3D annotation
- ☐ Create a UCS base feature
- ☐ Create named parameters
- ☐ Create Representations (LOD, View and Positional)

Presentation Template

- ☐ Set the Viewcube orientation
- ☐ Set the default view

Modelling checklist

Use this checklist to ensure that you are approving your design in a methodical manner.

Planning

- ☐ How will you name your files?
- ☐ Where will you save your files?
- ☐ What is the design intent? i.e. What parameters will drive your model?
- ☐ In which orientation will you create your model?
- ☐ Where would you like the origin (0,0,0) to end up when your model is finished?
- ☐ How will you name your part features and bodies?

Modelling

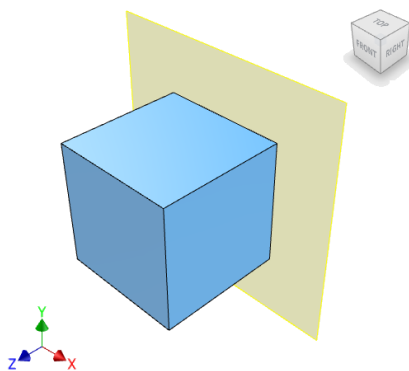
- ☐ How will you manage relationships between part features?
 - Parameter linking?
 - Adaptivity?
 - Derive (Multibody or Skeleton modelling)?
- ☐ What is your relationship strategy?
 - Constraints, Joints or Ground and position manually?
- ☐ Create Named Parameters
 - Use formulas to add design intent
 - Add a comment to describe what the parameter does
 - Use Multi-Value parameters where possible
 - Rename other important parameters as you go
- ☐ Create Layout Sketches
 - Define the overall size of the design
 - Define key datum points or lines
- ☐ Create Datums
 - Create UCS or Work features to create relationships with.
- ☐ Use iLogic to map assembly parameter values to part parameter values.
- ☐ Create an iLogic form to communicate design intent.
- ☐ *Flex!*
- ☐ Re-order components in the browser if necessary
- ☐ Group component using sub-assemblies or assembly folders.
- ☐ Document design intent
 - Parameter comments
 - Rename part nodes and Relationships
 - Add Engineering notes or 3D Annotations
- ☐ *Flex!*

Modelling Standards

You don't have to agree with me on standards such as file naming or model orientation, but you do need to follow your industry standards, or at least get everyone in your office to agree to work the same way!

Here are some topics that you'll need to consider. Please feel free to copy and paste from this section into your CAD manual.

Orientation



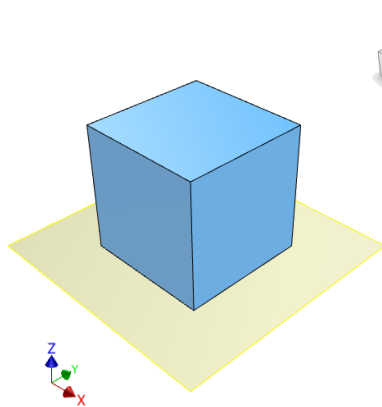
Traditionally **Engineers** have always drawn components from the front. This makes the coordinate system:

X = Left/Right

Y = Up/Down

Z = In/Out

This is how Inventor is set up.



Traditionally **Architects** have always drawn their building layouts from the top. This makes the coordinate system:

X = Left/Right

Y = Forward/Backward

Z = Up/Down

This is how AutoCAD is set up.

Engineering = Y Up

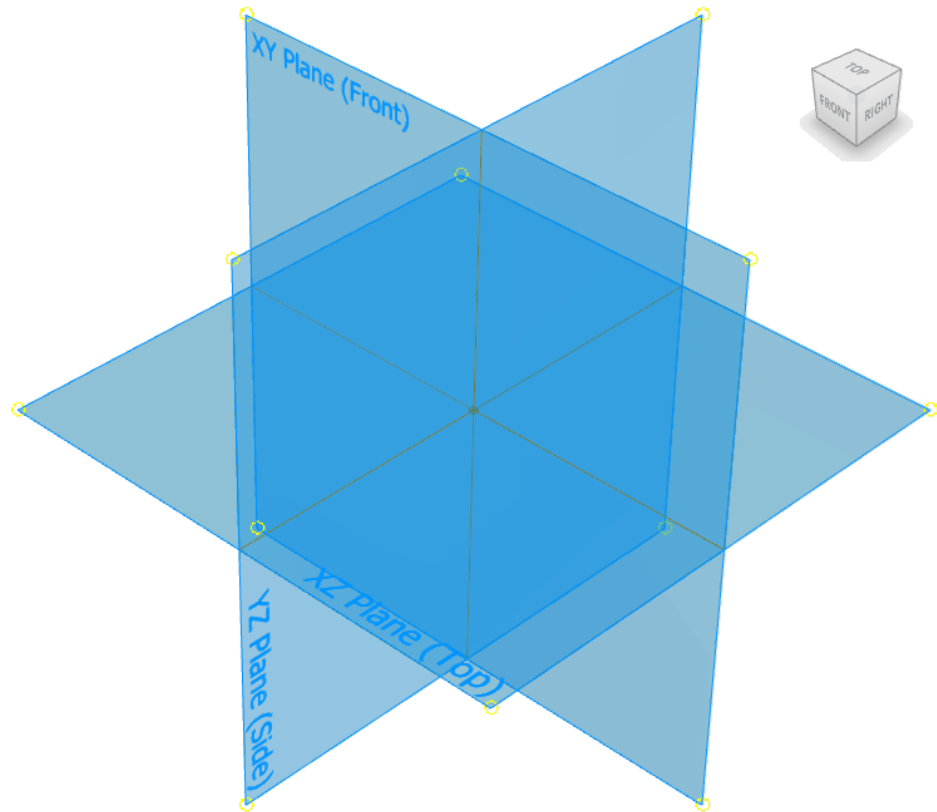
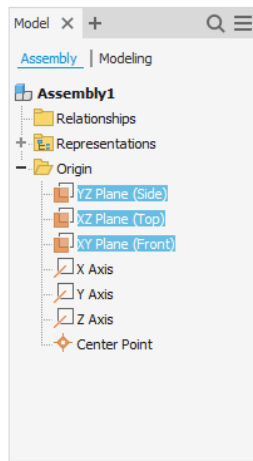
Architecture = Z Up

If you work with AutoCAD or Revit users, or you often share files with colleagues in the Architecture/Construction, you may want to model with 'Z up'.

You can set your Inventor template to use 'Z up' by adjusting your view cube orientation and saving your file as the 'Standard' template (remember to do this for your Part, assembly and presentation templates).

Origin Planes

You can't edit or delete the Origin planes, but you can re-name them. Rename the Origin planes in your template files to match your view cube orientation to subtly reinforce your modelling orientation standard.



File naming

Here are some file and folder naming tips you may want to consider when putting your strategy together:

Consider

- Drawings and Assemblies are unique to a project, so can probably contain a project number as a unique identifier.
- Part files can be project specific, so could be appended with a project number.
- Library components that belong to your company will definitely be used across multiple projects and should be unique.
- Library files from your suppliers could be named using your suppliers naming convention.
- Output files (PDF, DWF and Image formats) are not 'live data' – they are a snapshot of data in time and can have revision or date information in the file name.
- Output files may need to be saved in a location where non-cad users can access them. CAD-data can be locked down to CAD-users.

Avoid

- Windows has character limit of 252 characters – so keep your folder structure flat and you file names short*
- Avoid adding 'transient' data like reason or design state, this guarantees that you will have to re-name the file later.
- Avoid adding metadata, such as description, vendor's stock number, material, appearance to your file name. Each field can be extracted to a separate column in your parts list/BOM.
- Avoid calculated data, such as dimensions, volume, mass – this will change.

**260 character limit minus 'C:\' Minus the file extension minus backslashes for subfolders minus a hidden null terminator...*

Parameter naming

Parameters in Inventor represent placeholders for values that can change. Parameter names must follow some rules set down by Inventor.

To improve communication between users, it's a good idea to agree on some conventions for parameter naming.

You might also want to consider if there are any standard named parameters that should be in every model file? If so – include them in your template (and should your standard parameters be set to 'export' so that they can be read into the BOM/Parts List?).

Shorter parameter names are easier to remember and to type as you are modelling but can get obscure. All User Parameters should have a comment, to explain their function.

Parameter naming rules:

1. Parameter names are case sensitive (length & LENGTH are both acceptable).
2. Parameter names must start with a letter (but can include numbers).
3. Parameter names cannot contain spaces.
4. Parameter names can only include the Underscore '_' and Colon ':' characters.

Some parameter names are reserved by Inventor and cannot be used. [Please see the appendix for details.](#)

Examples

OverallWidth
Overall_Width
OAwidth
OA:Width

Tip: Don't forget that you can rename parameters on the fly. Type the formula:

'Parameter name = Parameter Value'

into any input box to rename the parameter as it is created.

Component Data

There are Four places that we can enter data into a Part or Assembly file. They are:

1. Materials and Appearances
2. Parameters
3. BOM Settings
4. iProperties

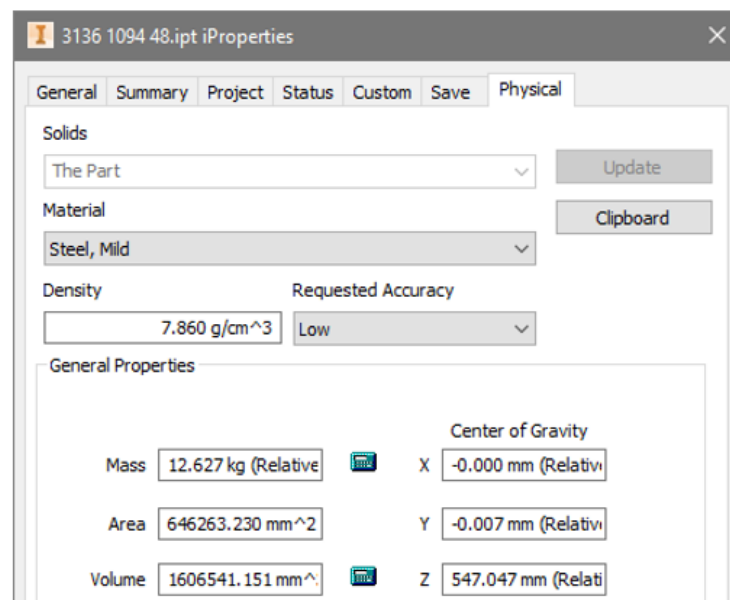
Note: You don't have to enter data in all these places, but you do need to be consistent. A database is only as good as the data entered it.

You can't create an automated BOM and Parts List if you didn't enter the data into the parts. It can be a real pain to have to go back and enter all the data at the end of the project.

Materials and Appearances

Each part can only contain one material. This property is used to calculate the mass of the part and can be used downstream for FEA and simulation.

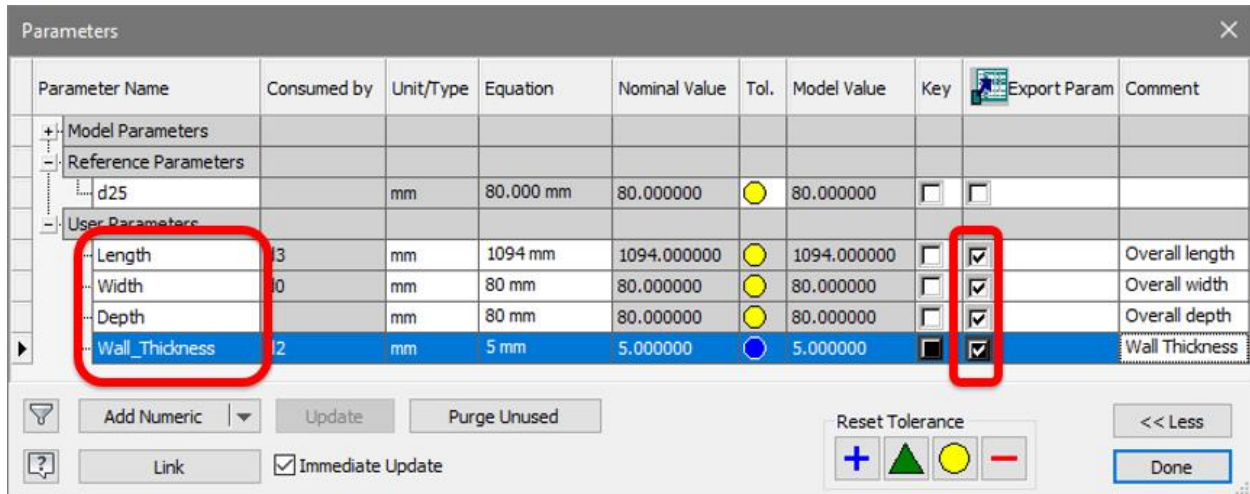
Part files can contain multiple appearances, but only the default appearance is read into the BOM. This can be used to represent coatings or other finish information.



Parameters

Parameters can be exported and read into the BOM. This can be helpful for capturing dynamic property values such as the dimensions of a part.

To read a parameter value into the BOM, tick the 'Export' box in the parameter manager. The Parameter will become available as a custom iProperty.



Parameter Name	Consumed by	Unit/Type	Equation	Nominal Value	Tol.	Model Value	Key	Export Param	Comment
Model Parameters									
Reference Parameters									
d25		mm	80.000 mm	80.000000	Yellow	80.000000			
User Parameters									
Length	3	mm	1094 mm	1094.000000	Yellow	1094.000000		<input checked="" type="checkbox"/>	Overall length
Width	0	mm	80 mm	80.000000	Yellow	80.000000		<input checked="" type="checkbox"/>	Overall width
Depth		mm	80 mm	80.000000	Yellow	80.000000		<input checked="" type="checkbox"/>	Overall depth
Wall_Thickness	2	mm	5 mm	5.000000	Blue	5.000000		<input checked="" type="checkbox"/>	Wall Thickness

Buttons: Add Numeric, Update, Purge Unused, Link, Immediate Update, Reset Tolerance, Done.

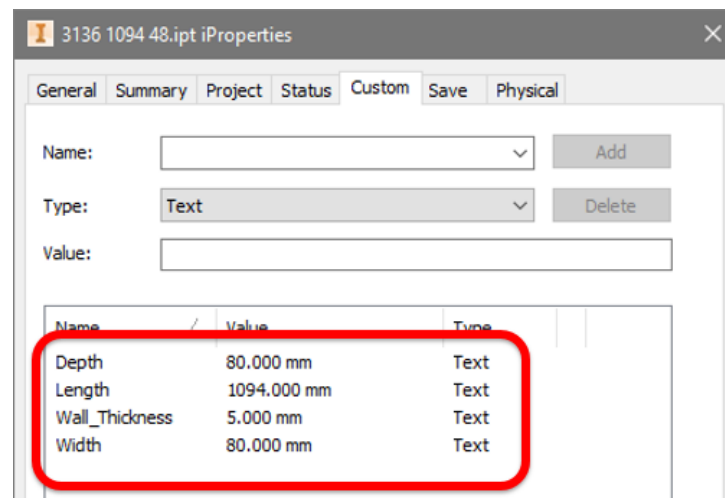
iProperties

I hope that you are all familiar with iProperties? iProperties are the component's meta-data (Literally 'Data about the data').

The iProperties manager contains text-based data, such as 'Company', 'Designer', 'Checked by' and so on.

You can also create custom iProperties to capture any data that isn't listed in the standard iProperty fields.

You don't have to fill out all the iProperty fields in *every* component, but you do need to be consistent!



Parameters that are ticked for export (see previous image) will show up as custom iProperties and can be referenced in your BOM and parts list. Their values will update when the part updates.

BOM Settings

To find your BOM settings, navigate to:

'Tools' (Tab) > 'Settings' (Panel) > 'Document Settings' (Button) > 'Bill of Materials' (Tab)

The BOM settings allow us to do two things – first we can set the BOM structure for the part. Typically, this is:

- **Normal** a fabricated component
- **Purchased** a purchased component

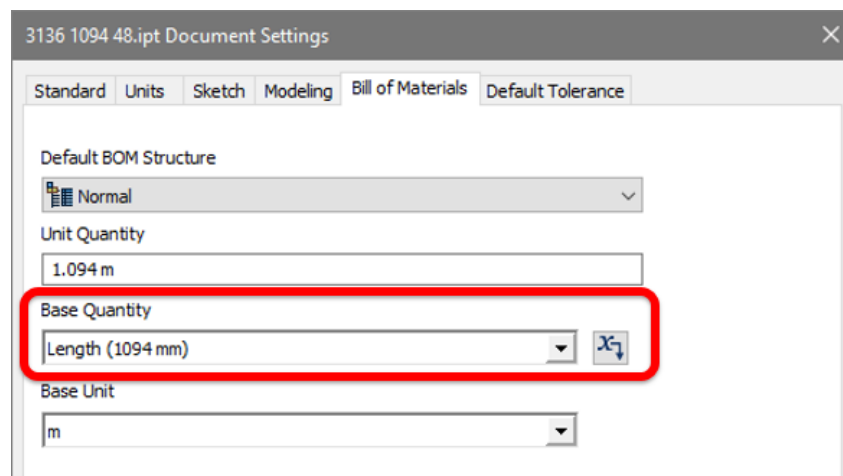
This allows us to filter out part list between fabricated and purchased items.

In addition, you could choose:

- **Phantom** a component that will not be listed in the BOM.
- **Reference** a component that will not be listed in the BOM and will only appear as a dotted outline in drawings.
- **Inseparable** an assembly that is listed In the BOM as one item.

Secondly – we can set the base quantity for the component. This allows us to quantify the component by its dimensions (Such as volume or weight), rather than simply the number of components required.

This allows us to calculate how much material is required to make our components.

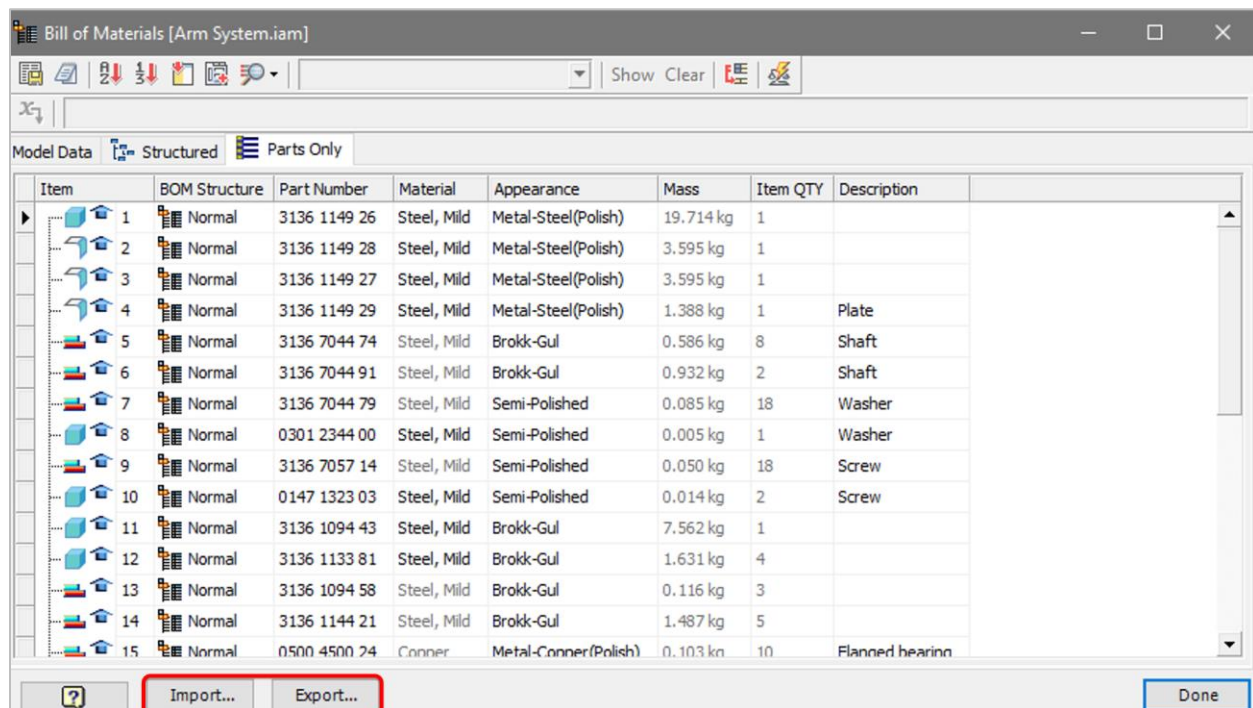


BOM Layout

Do you ever open the BOM your assembly files and spend time adjusting which columns are shown? If so **STOP!** Don't do this anymore. We can get this set up in your template. This will save you time and improve consistency as your team collaborate on projects.

Navigate to your assembly template, open it up and configure the columns the way you like them (according to your CAD standard, obviously)

Do you have a BOM that is already set up the way you like it?



Item	BOM Structure	Part Number	Material	Appearance	Mass	Item QTY	Description
1	Normal	3136 1149 26	Steel, Mild	Metal-Steel(Polish)	19.714 kg	1	
2	Normal	3136 1149 28	Steel, Mild	Metal-Steel(Polish)	3.595 kg	1	
3	Normal	3136 1149 27	Steel, Mild	Metal-Steel(Polish)	3.595 kg	1	
4	Normal	3136 1149 29	Steel, Mild	Metal-Steel(Polish)	1.388 kg	1	Plate
5	Normal	3136 7044 74	Steel, Mild	Brokk-Gul	0.586 kg	8	Shaft
6	Normal	3136 7044 91	Steel, Mild	Brokk-Gul	0.932 kg	2	Shaft
7	Normal	3136 7044 79	Steel, Mild	Semi-Polished	0.085 kg	18	Washer
8	Normal	0301 2344 00	Steel, Mild	Semi-Polished	0.005 kg	1	Washer
9	Normal	3136 7057 14	Steel, Mild	Semi-Polished	0.050 kg	18	Screw
10	Normal	0147 1323 03	Steel, Mild	Semi-Polished	0.014 kg	2	Screw
11	Normal	3136 1094 43	Steel, Mild	Brokk-Gul	7.562 kg	1	
12	Normal	3136 1133 81	Steel, Mild	Brokk-Gul	1.631 kg	4	
13	Normal	3136 1094 58	Steel, Mild	Brokk-Gul	0.116 kg	3	
14	Normal	3136 1144 21	Steel, Mild	Brokk-Gul	1.487 kg	5	
15	Normal	0500 4500 74	Conner	Metal-Conner(Polish)	0.103 kg	10	Flanged bearing

TIP: Use the 'Import' and 'Export' buttons to export out your columns and import them into your template.

Don't forget to standardise the column layout for the 'Model Data', 'Structured' and 'Parts only' tabs.

Don't forget to apply the same edits your Weldment template.

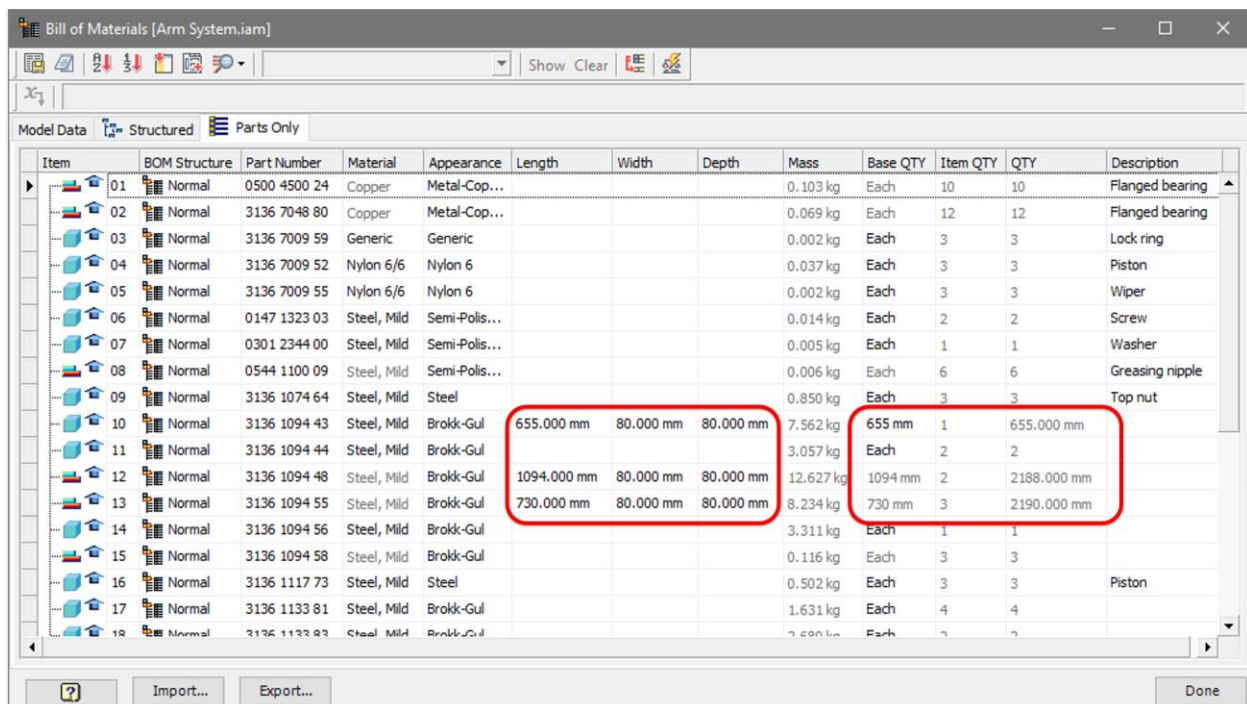
BOM layout example

In this example, the Parameter 'Length' has been exported as a custom parameter – which is read into the Bom using a custom parameter column.

In addition, the quantity of the part has also been set to equal the value of 'Length'.

In the BOM, we can see that 2190mm of material is required to fabricate the three components of this type that we have specified.

Note: We can't calculate a total quantity of material required for all components of the same specification in the BOM. But we can do this step in a parts list.



Item	BOM Structure	Part Number	Material	Appearance	Length	Width	Depth	Mass	Base QTY	Item QTY	QTY	Description
01	Normal	0500 4500 24	Copper	Metal-Cop...				0.103 kg	Each	10	10	Flanged bearing
02	Normal	3136 7048 80	Copper	Metal-Cop...				0.069 kg	Each	12	12	Flanged bearing
03	Normal	3136 7009 59	Generic	Generic				0.002 kg	Each	3	3	Lock ring
04	Normal	3136 7009 52	Nylon 6/6	Nylon 6				0.037 kg	Each	3	3	Piston
05	Normal	3136 7009 55	Nylon 6/6	Nylon 6				0.002 kg	Each	3	3	Wiper
06	Normal	0147 1323 03	Steel, Mild	Semi-Polis...				0.014 kg	Each	2	2	Screw
07	Normal	0301 2344 00	Steel, Mild	Semi-Polis...				0.005 kg	Each	1	1	Washer
08	Normal	0544 1100 09	Steel, Mild	Semi-Polis...				0.006 kg	Each	6	6	Greasing nipple
09	Normal	3136 1074 64	Steel, Mild	Steel				0.850 kg	Each	3	3	Top nut
10	Normal	3136 1094 43	Steel, Mild	Brokk-Gul	655.000 mm	80.000 mm	80.000 mm	7.562 kg	Each	1	655.000 mm	
11	Normal	3136 1094 44	Steel, Mild	Brokk-Gul	1094.000 mm	80.000 mm	80.000 mm	3.057 kg	Each	2	2	
12	Normal	3136 1094 48	Steel, Mild	Brokk-Gul	730.000 mm	80.000 mm	80.000 mm	12.627 kg	Each	2	2188.000 mm	
13	Normal	3136 1094 55	Steel, Mild	Brokk-Gul				8.234 kg	Each	3	2190.000 mm	
14	Normal	3136 1094 56	Steel, Mild	Brokk-Gul				3.311 kg	Each	1	1	
15	Normal	3136 1094 58	Steel, Mild	Brokk-Gul				0.116 kg	Each	3	3	
16	Normal	3136 1117 73	Steel, Mild	Steel				0.502 kg	Each	3	3	Piston
17	Normal	3136 1133 81	Steel, Mild	Brokk-Gul				1.631 kg	Each	4	4	
18	Normal	3136 1133 83	Steel, Mild	Brokk-Gul				1.600 kg	Each	3	3	

Tip: The BOM manager is a great tool for checking that all the data you require has been added. If there are any blank cells shown in the BOM manager – you are missing data!

In most cases you can enter the missing data in the BOM manager. When you save the assembly, the data will be 'pushed' down into the components.

Documenting Design Intent

So – you’ve built a [component](#) with solid design intent. Simply by changing a few parameters your component updates in a predictable fashion. Well done you!

Now, how are your team going to know which buttons to push to update your component?

Communication of your Design Intent is super important if you want your colleagues to know just how hard you worked to build them a useful resource.

So – how to we document our design intent?

Tip: *Don’t forget that it might be YOU who must come back and edit this design in three months. Will you remember how to edit your own model?!*

Parameter comments

The first thing we want anyone to do upon opening our model – is to open the parameters manager and look for useful parameters.

If we have created a named parameter called ‘Length’ and our colleague wants to change the length of the component – the design intent is obvious. Your colleague will thank you for making their life so easy!

It's often not possible to capture the entire design intent in a parameter name, so add a comment to clarify what the parameter is controlling.

Tip: Create user parameters that capture your design intent before you model anything. If you can't think of them all up front, don't worry – just come back and add more user parameters as you need them.

If you want to keep track of a parameter (for example, in an equation), but it won't be a driving parameter. Use the ‘Parameter name = Parameter value’ formula to rename parameters as you go along.

Parameters									
Parameter Name	Consumed by	Unit/Type	Equation	Nominal Value	Tol.	Model Value	Key	Expo	Comment
Model Parameters									
Reference Parameters									
User Parameters									
HoleDistHoriz	d0	mm	40 mm	40.000000	●	40.000000	✓	<input type="checkbox"/>	Horizontal distance between holes
HoleDistVert	d2	mm	40 mm	40.000000	●	40.000000	✓	<input type="checkbox"/>	Vertical distance between holes
BossDia	d10, d4, d3	mm	8 mm	8.000000	●	8.000000	✓	<input type="checkbox"/>	Diameter of the Radiused bosses
BaseDepth	d14, d12, d8	mm	10 mm	10.000000	●	10.000000	✓	<input type="checkbox"/>	Depth of the base feature
BossHoleDepth	d12	mm	5 mm	5.000000	●	5.000000	✓	<input type="checkbox"/>	Depth of the Radiused bosses
BossCenterDepth	d14	mm	12 mm	12.000000	●	12.000000	✓	<input type="checkbox"/>	Depth of the Circular feature
HexDepth	d16	mm	14 mm	14.000000	●	14.000000	✓	<input type="checkbox"/>	Depth of the Hexagonal feature
FrontConnectorDepth	d17	mm	12 mm	12.000000	●	12.000000	✓	<input type="checkbox"/>	Depth of the front connector
TopConnectorDepth		mm	15 mm	15.000000	●	15.000000	✓	<input type="checkbox"/>	Depth of the top connector
FlangeWidth	d60	mm	37.5 mm	37.500000	●	37.500000	✓	<input type="checkbox"/>	Width of the triangular flange
BossCenterDia	d18	mm	30 mm	30.000000	●	30.000000	✓	<input type="checkbox"/>	Diameter of the Circular feature
HexAcrossFlats	d24, d19	mm	20 mm	20.000000	●	20.000000	✓	<input type="checkbox"/>	Distance across flats of the Hexagonal feature
FrontConnectorDia	d20	mm	16 mm	16.000000	●	16.000000	✓	<input type="checkbox"/>	Diameter of the front connector
FlangeThick	d30	mm	8 mm	8.000000	●	8.000000	✓	<input type="checkbox"/>	Thickness of the triangular flange

☒ Immediate Update

Reset Tolerance

Adding Notes

The more information you can embed in your design, the more likely that your colleagues will be able to figure out your design intent.

Want to leave a note in your file to say:

'I'm not finished with this model yet!'

Or

'Use the imbedded iLogic form to edit this model'

We have several options available. I suggest that you discuss this with your team and pick one that everyone can use consistently.

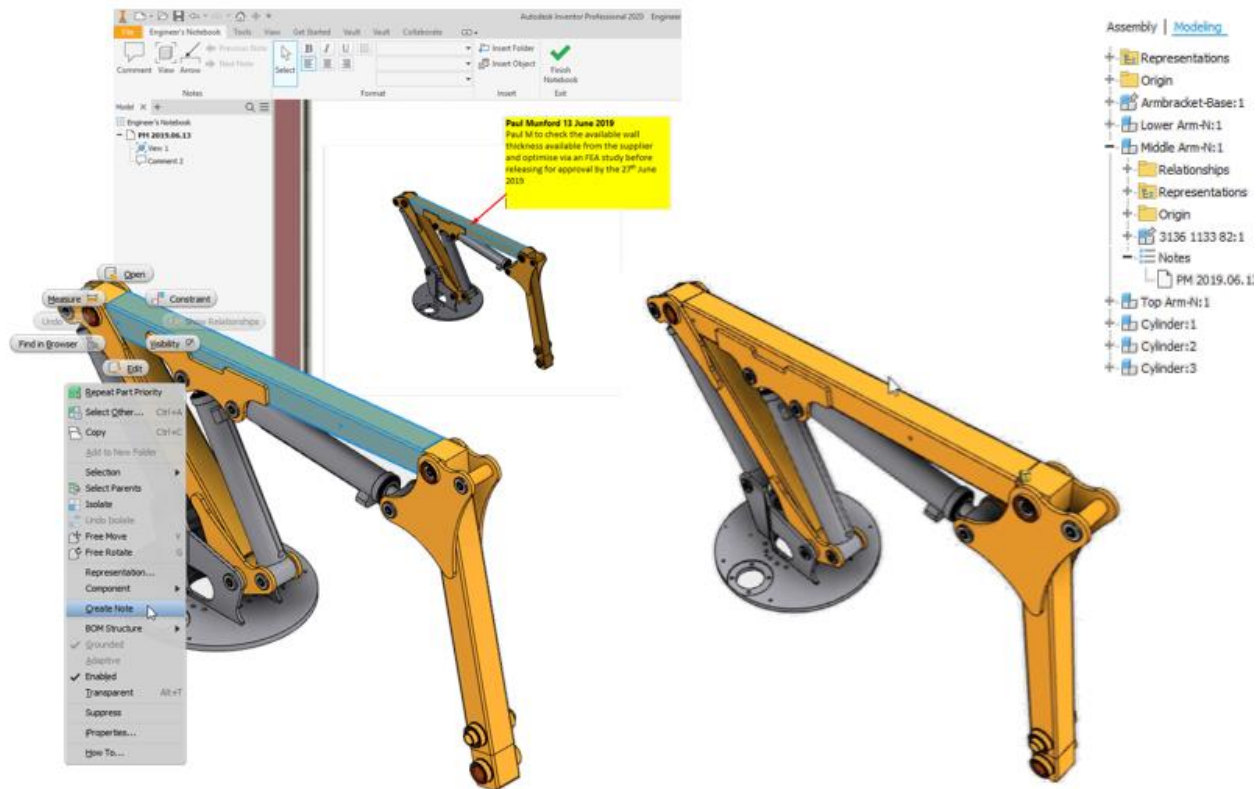
Engineers Notebook

Right click on any part file in an assembly, or any sketch edge of feature in a part file and pick 'Create Note' from the context menu.

You will automatically be taken to the Engineers notebook, where you will find a screen shot of your component, and a text note for you to add your information to.

More information in the online help:

<https://help.autodesk.com/view/INVENTOR/2019/ENU/?guid=GUID-19FA6297-F980-456D-A53F-226DA9870B73>



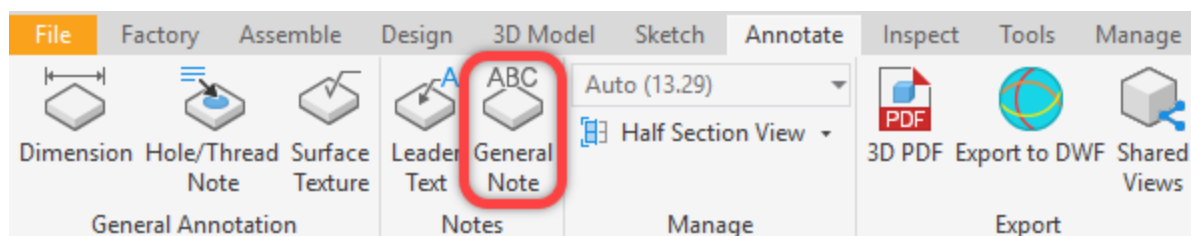
Model based Annotation – General note

Model based annotation is relatively new. If you haven't tried it out yet – give it a go.

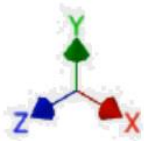
My favourite is the 'General Note' tool, which creates a note which sits in the corner of your screen and remains there – even if you rotate the model.

Navigate to:

'Annotate' (Tab) > 'Notes' (Panel) > 'General Note' (Button).



2019.06.13 PM to check wall thickness from supplier

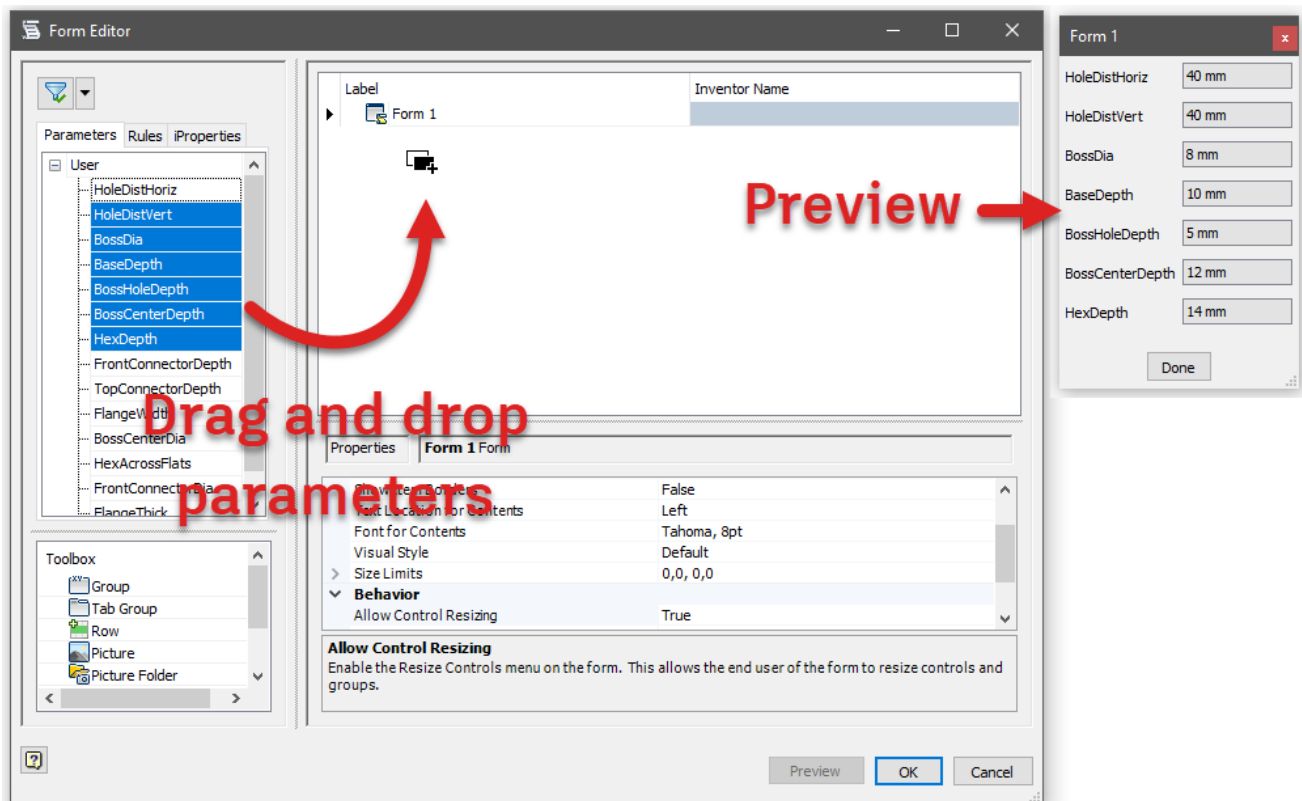
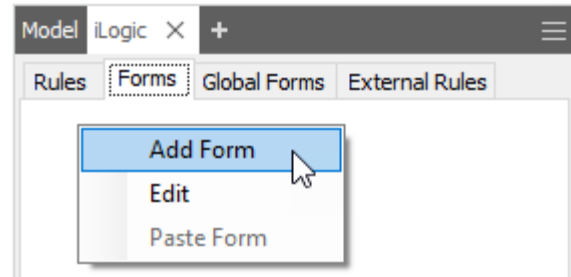
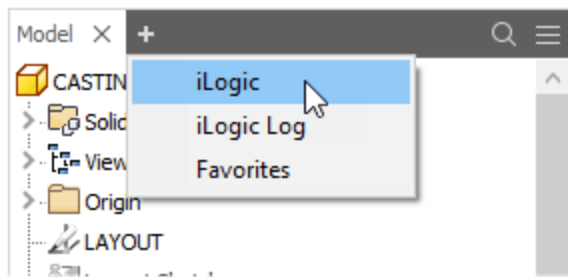


iLogic forms to communicate design intent

Even if you take the time to name and comment your key parameters, it can be confusing for your colleagues to trigger the change they need.

A really simple way to communicate design intent is to use an iLogic form to control the part.

1. Open the iLogic browser
2. Create an iLogic form
3. Drag and drop parameters
4. Click on the button to trigger the form.



iLogic form pro-tips

Limit choice to the available sizes by making parameters multi value.

Parameters								
	Parameter Name	Consumed by	Unit/Type	Equation	Nominal Value	Tol.	Model Value	Key
+	Model Parameters							
+	Reference Parameters							
-	User Parameters							
▶	HoleDistHoriz	d0	mm	40 mm	40.000000	●	40.000000	
	HoleDistVert	d2	mm	40 mm				
	BossDia	d10, d4, d3	mm	8 mm				
	BaseDepth	d14, d12, d8	mm	10 mm	10.000000	●	10.000000	<input checked="" type="checkbox"/>

Copy To User Parameter

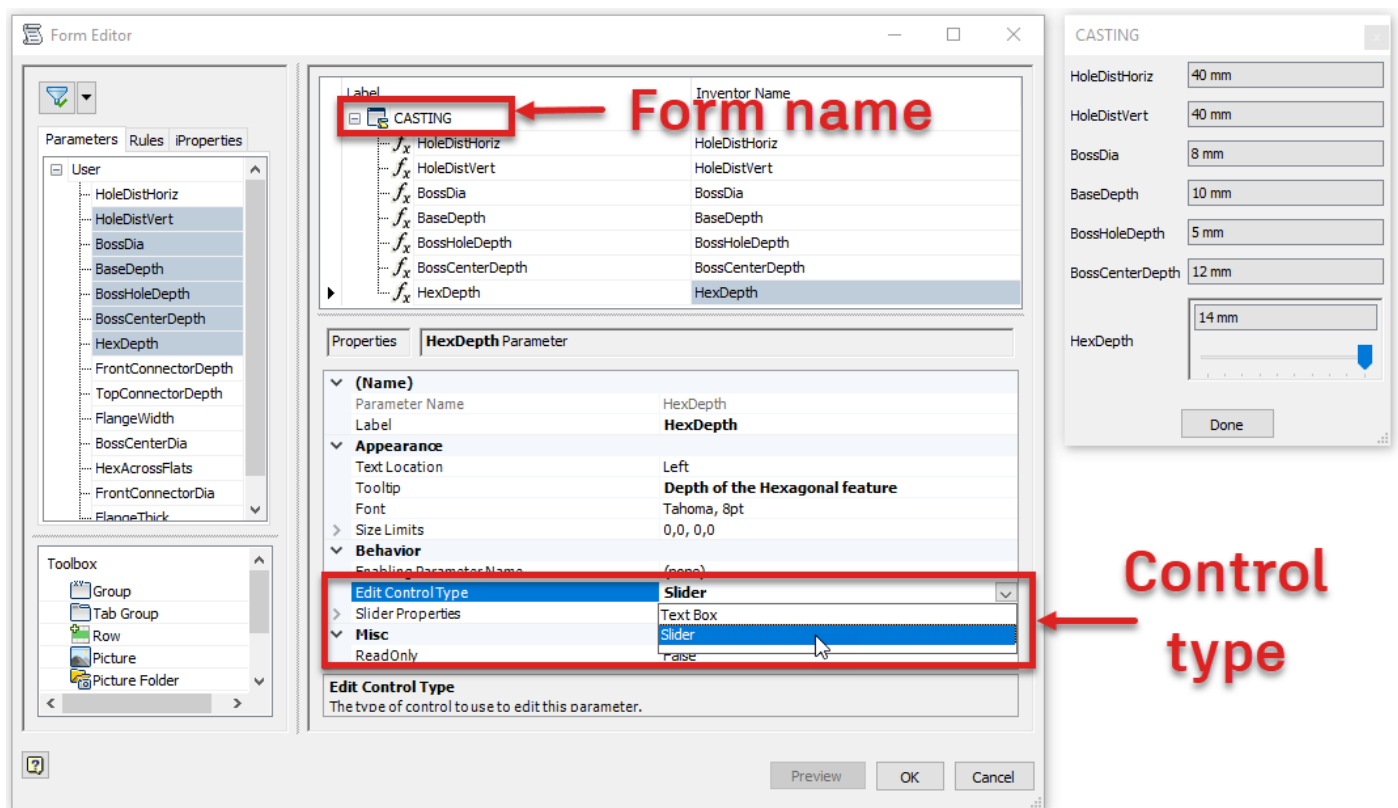
Make Multi-Value

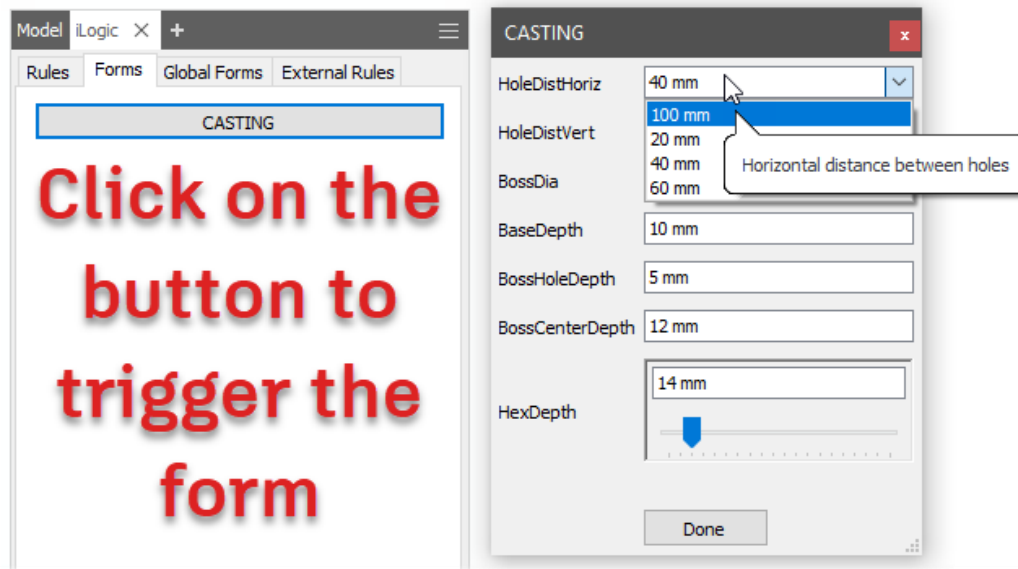
Parameters								
	Parameter Name	Consumed by	Unit/Type	Equation	Nominal Value	Tol.	Model Value	Key
+	Model Parameters							
+	Reference Parameters							
-	User Parameters							
▶	HoleDistHoriz	d0	mm	40 mm	40.000000	●	40.000000	<input checked="" type="checkbox"/>
	HoleDistVert	d2	mm	100 mm	40.000000	●	40.000000	<input checked="" type="checkbox"/>
	BossDia	d10, d4, d3	mm	20 mm	8.000000	●	8.000000	<input checked="" type="checkbox"/>
				40 mm				
	BaseDepth	d14, d12, d8	mm	60 mm	10.000000	●	10.000000	<input checked="" type="checkbox"/>

Edit the name of your form. The name of your form can be seen on the button that triggers your form to appear.

Edit the control type to 'slider'. A slider control has minimum, maximum and increment properties. This will limit users from picking a value which is out of range.

▼ Slider Properties	5; 50; 5
Minimum Value	5
Maximum Value	50
Step Size	5





Note: Your parameter comments become tool tips – no effort is wasted!

Using iLogic to prevent user error

Nothing is foolproof! Even though we went to all this effort to make the Design Intent of this component clear, we may still want to protect our model from the unexpected.

‘Don’t let a management problem become a technology problem’

Mark Kiker

Although I prefer good communication over complex technical solutions, iLogic can be helpful in this situation.

To create a new rule, right click in the iLogic browser > Rules tab, and choose ‘Add Rule’.

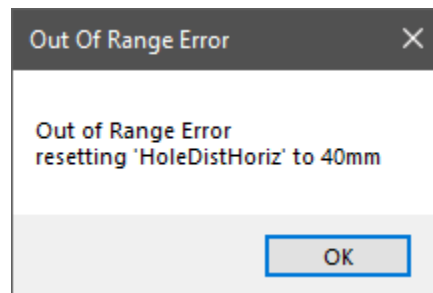
Now copy and paste the following code, replacing [Parameter_name] with the name of the parameter you want to control.

```
If [Parameter_Name] < 40 mm Then

MessageBox.Show("Out of Range Error" & vbLf & "resetting 'Parameter_Name' to 40mm", _
"Out Of Range Error")

[Parameter_Name] = 40 mm

End If
```



This code reads:

If [Parameter_Name] is less than 40mm then, show a message box – with the title

‘Out of Ranger Error’

which includes the message

‘Out of Ranger error, resetting the parameter named ‘Your parameter name’ to 40mm’

And then set the parameter [Parameter_Name] to 40mm.

iLogic can help you to prevent your colleagues changing values which will break your design intent.

Maybe this is slightly harsh?

Instead of resetting a value, maybe you could change the appearance of a component to indicate an error?

I'm sure that you can think of many ways to use iLogic to prevent your designs from becoming broken!

To learn iLogic:

Start with this introduction tutorial on the Autodesk Knowledge Network website:

<https://knowledge.autodesk.com/support/inventor-products/getting-started/caas/CloudHelp/cloudhelp/2015/ENU/Inventor-Tutorial/files/GUID-1414459D-3CF8-4719-8DDF-1AC17CE184F1-htm.html>

Check out these free to watch classes on the Autodesk University online website:

<http://au.autodesk.com/au-online/classes-on-demand/search?full-text=ilogic>

Check out Curtis Waguespack's blog 'From the Trenches with Autodesk Inventor:

<http://inventortrenches.blogspot.com/2013/10/ilogic-how-to-learn-inventors.html>

Flex your design

Test your designs! If you have invested effort building a model which can change over time – don't simply assume that your model will update predictably!

Don't forget, it could be *you* who must perform a quick Friday afternoon update to a model. No-one wants to be dealing with an exploding model when they have a deadline to meet.

I've stolen a term from Revit designers here, they call this testing 'Flexing' the model.

Flex early, flex often, flush any unpredictable behaviour out of your design while the design intent is fresh in your memory.

It's far more difficult to correct mistakes when you are revisiting a component after a few months working on other projects.

Conclusion

Simple designs require simple solutions. 3D modelling with Autodesk Inventor needn't be restrictive – it's fun!

When you see the opportunity to build designs that will pay you back for your planning and forethought, I hope that you will find the strategies described in the class useful.

Glossary

Base Feature – The first feature in your model (the one at the top of the browser).

Base Component – The first component you place in an Assembly. The base component is usually placed at the origin and grounded. All other components are constrained to the base component.

Parent – An item that is referenced by another item.

Child – An item which is referencing, or dependent upon, another item.

Parent Child Relationship – Describes the way parametric CAD can build one-way relationships between items. The actions of the parent change the child. The actions of the child do not change the parent.

Parameter – A placeholder for a value.

Design Intent – The desired outcome when a parameter is changed and the model updates.

Feature – Geometry that creates or modifies a component.

Body – A collection of features in a part file that comprise a solid (Solids and bodies are often used interchangeably).

Multibody – A part file that contains multiple bodies. In a top down modelling workflow, one body represents one part. Bodies are subsequently derived to form an assembly.

Solid – An operation in a part file that creates a new body (Solids and bodies are often used interchangeably).

Component – A generic term for parts or assemblies.

Design – a generic term for the collection of documents including parts, assemblies, presentation files, drawings and additional supporting data.

Appendix

Good modelling technique references

B.O.R.N – Base Orphan Reference Node

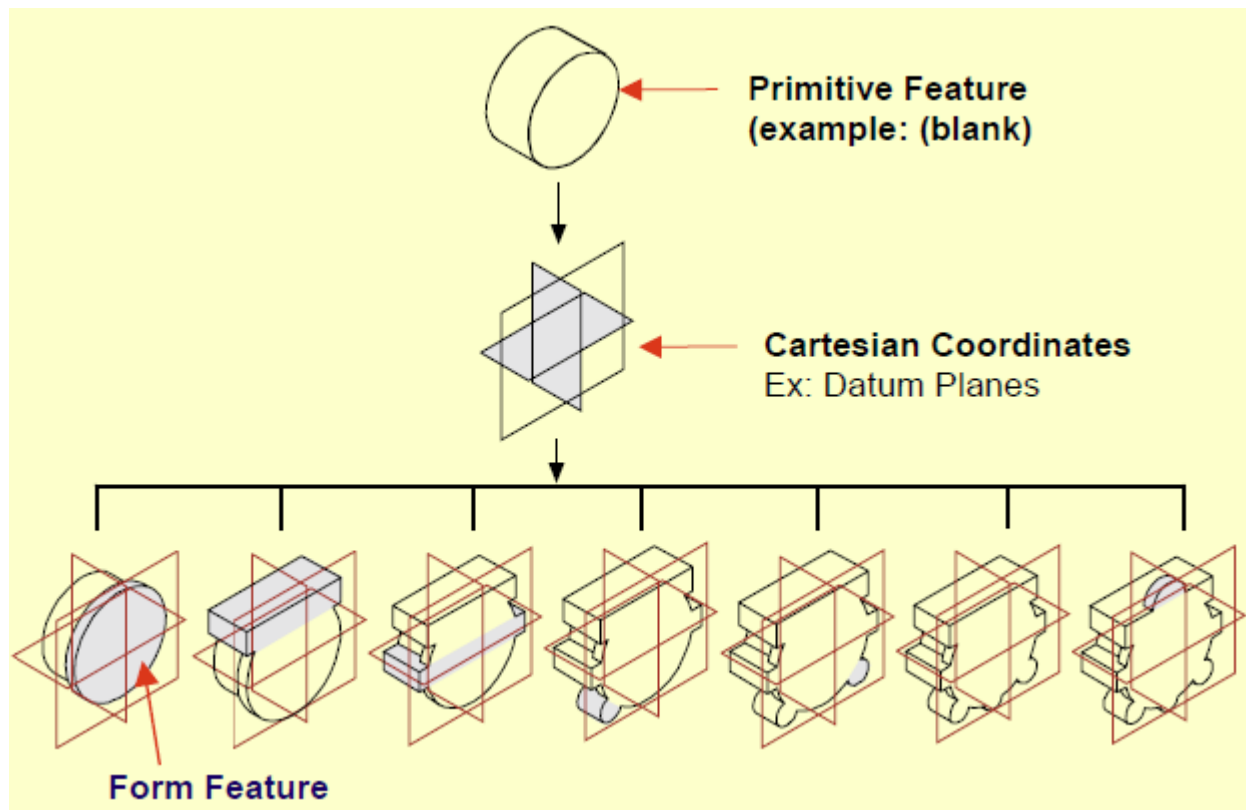
Limit relationships between features by creating all sketches on the origin planes and referencing all sketched geometry to the center point (Node).

The Center Point and Origin Planes are the default base feature in any Inventor modelling file. They don't have a '[Parent](#)', and so are considered 'Orphaned'.

An advanced technique is to create a UCS (Cartesian coordinate system) as your Base feature. This can be moved and rotated independently of the Center point and origin planes. Limit the number of relationships you create in your model, by referencing every sketch back to this 'Reference node'.

<http://blogs.rand.com/files/the-born-technique.pdf>

Horizontal modelling



All features are placed on datums. The only parent/[child](#) relationship is between the feature and its datum. By limiting the parent/child relationships in the model, features can easily be added, reordered or removed.

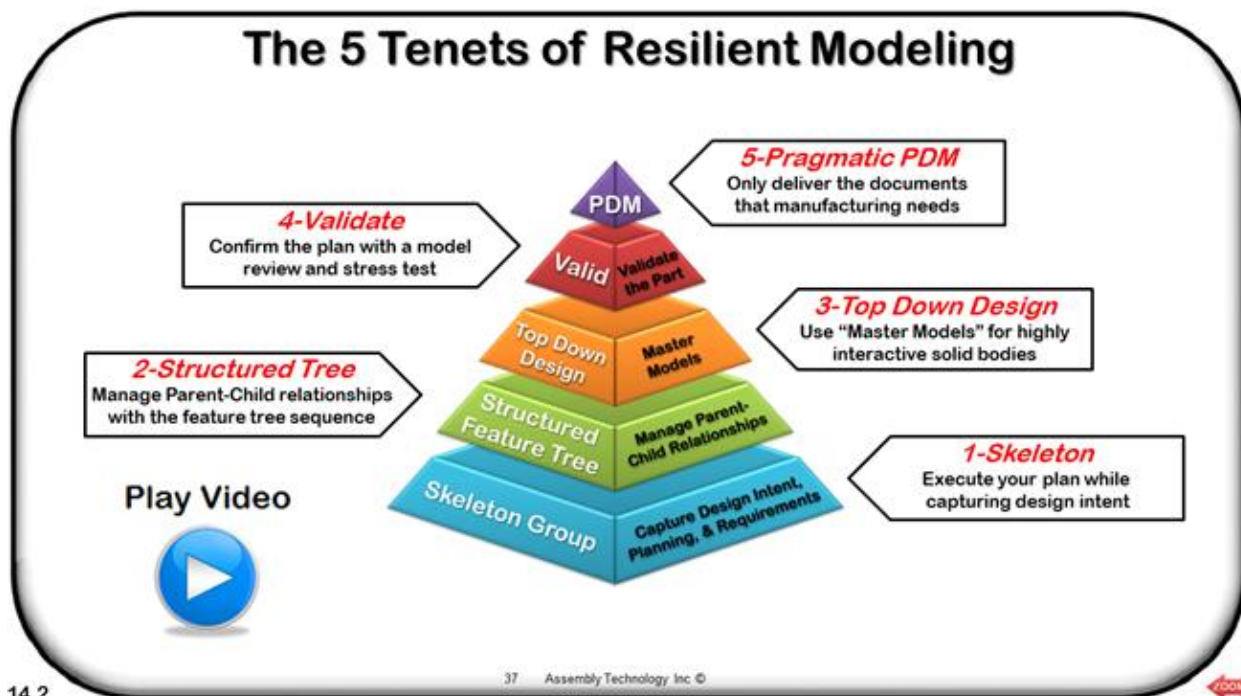
<https://patents.google.com/patent/US7472044>

The Resilient Modelling Strategy (RMS)

Dick Gebhard pulls together best practices from 'B.O.R.N', 'Horizontal Modelling' and many others into one comprehensive, and well thought out CAD Neutral modelling strategy.

"All models are resilient until you try to edit them."

Dick Gebhard



<http://learnrms.com/index.html>

Other good resources

'Reliable Modelling Techniques for Complex Part Design with Inventor'

<https://www.autodesk.com/autodesk-university/class/Reliable-Modeling-Techniques-Complex-Part-Design-Inventor-2018>

Autodesk Inventor large assembly best practices

<https://knowledge.autodesk.com/support/inventor-products/learn-explore/caas/CloudHelp/cloudhelp/2019/ENU/Inventor-Help/files/GUID-564D4AF1-E566-490F-9491-BAAFD208F1BC-htm.html>

Driving Inventor with the top down – Alternative assembly modeling techniques.

<http://au.autodesk.com/au-online/classes-on-demand/class-catalog/classes/year-2013/product-design-suite/ma2604#chapter=0>

'The failed Promise of Parametric CAD'

<https://www.3dcadworld.com/the-failed-promise-of-parametric-cad/>

Parametric CAD Modeling: An Analysis of Strategies for Design Reusability

<https://core.ac.uk/download/pdf/61471988.pdf>

Get smart & The Seven deadly sins of 3D Part Modelling

<http://au.autodesk.com/au-online/classes-on-demand/class-catalog/classes/year-2017/inventor/cp122682#chapter=0>

Complex topology and Class A Surface modelling with Autodesk Inventor

<http://au.autodesk.com/au-online/classes-on-demand/class-catalog/classes/year-2015/inventor/cp10847#chapter=0>

Workspace Envelope Modelling

<https://knowledge.autodesk.com/support/inventor-products/learn-explore/caas/CloudHelp/cloudhelp/2017/ENU/Inventor-Help/files/GUID-E240CC6A-0D19-4C09-9DF3-F9E70BF1B892-htm.html>

Common Origin Modelling

<https://knowledge.autodesk.com/support/inventor-products/learn-explore/caas/CloudHelp/cloudhelp/2017/ENU/Inventor-Help/files/GUID-D89AE80F-6AB7-4ACF-A015-47E3FA512DFE-htm.html>

Reserved Parameter Names

The following Parameter names are reserved by Inventor and cannot be used when custom naming Parameters.

Source: <http://blogs.rand.com/files/reserved-inventor-parameter-names.pdf>

°	dyne	Gravity	ln	nauticalMile	siemens
A	E	h	log	newton	sign
abs	erg	H	lumen	oersted	sin
acos	exp	henry	lux	ohm	sinh
acre	f	hertz	lx	ounceforce	slug
ampere	F	horsepower	m	ouncemass	SpeedOfLight
asin	farad	hour	max	ouncevolume	sqrt
atan	fahrenheit	hp	maxwell	Pa	sr
btu	FeetPerSecond	hr	mega	pascal	steradian
c	d	Hz	meter	peta	switch
C	femto	if	MetersPerSecond	PI	T
cal	fl_oz	in	ond	pint	tan
calorie	floor	inch	mho	pow	tanh
candela	foot	isolate	mi	psi	tesla
cd	fps	J	micron	pt	ul
ceil	ft	joule	mil	qt	unitless
celsius	g	K	mile	quart	V
centi	gal	kelvin	MilesPerHour	rad	volt
circular_mil	gallon	kilo	min	radian	W
cos	gamma	ksi	minute	random	watt
cosh	gauss	l	mole	round	Wb
coulomb	giga	lbforce	mph	rpm	weber
cup	grad	lbmass	mps	s	yard
deg	gradient	liter	N	S	yd
degree	gram	lm	nano	second	