

CP468677

## Getting Tipsy with Fusion 360

Scott Moyse  
CADPRO Systems

### Learning Objectives

- Learn how to implement a series of simple tips and tricks in the Design workspace which will immediately impact your daily productivity.
- Discover various methods to model and reuse similar geometry across the Design and Drawing workspaces.
- Learn how to create product animations that tell a useful story.
- Learn how to create rigs to generate beautiful renderings of glass and fluids within the Render Workspace.

### Description

How many tips do you think we can learn in 60 minutes while recreating the Engineers' Guide to Drinks within Fusion 360 software? The Engineers Guide to Drinks drawing was a test drawing originally created to test pen plotters back in 1972. Autodesk stalwart Shaan Hurley recreated it in AutoCAD software a number of years ago. It's since been replicated in a number of other applications. When coming up with a title for this tips and tricks class, the speaker thought about how much more sober everyone will be at Autodesk University this year given it's not being held in Las Vegas (in theory, at least). This thought immediately invoked the Engineers' Guide to Drinks and how it would be a fun way to cover a series of tips and tricks across sketching, modeling, assemblies, animation, rendering, and manufacturing. Let's do this!

## Speaker



Scott Moyse is a Manufacturing, CAM & Data Management Technical Specialist for CADPRO Systems in New Zealand.

He previously worked at SMI for over 9 years after moving from the UK while studying Motorsport Engineering. Starting out as design support, Scott quickly moved into programming their new CNC machine. Over the next 4 years he worked closely with both manufacturing & design to create & implement automated processes.

This provided him with an invaluable insight into both departments' operations. In 2008 he moved back into design full-time, resulting in him taking the Design Manager position in 2009. For the last 3 years of that position, he implemented & managed Autodesk Vault Professional, improving communication, work allocation, organization & increased control over the design review process.

Over the years he has been deeply involved in process formation, implementation & development in an ever changing environment. Since late 2013, Scott has been working for CADPRO Systems in New Zealand. He covers most of Autodesk's manufacturing focused design and engineering products. CAM is now his primary focus, supporting customers across New Zealand and Australia. Over the last few years, Scott has developed skills to write Post Processors for Fusion 360, HSMWorks and Inventor CAM as well as developing add-ins for Fusion 360.

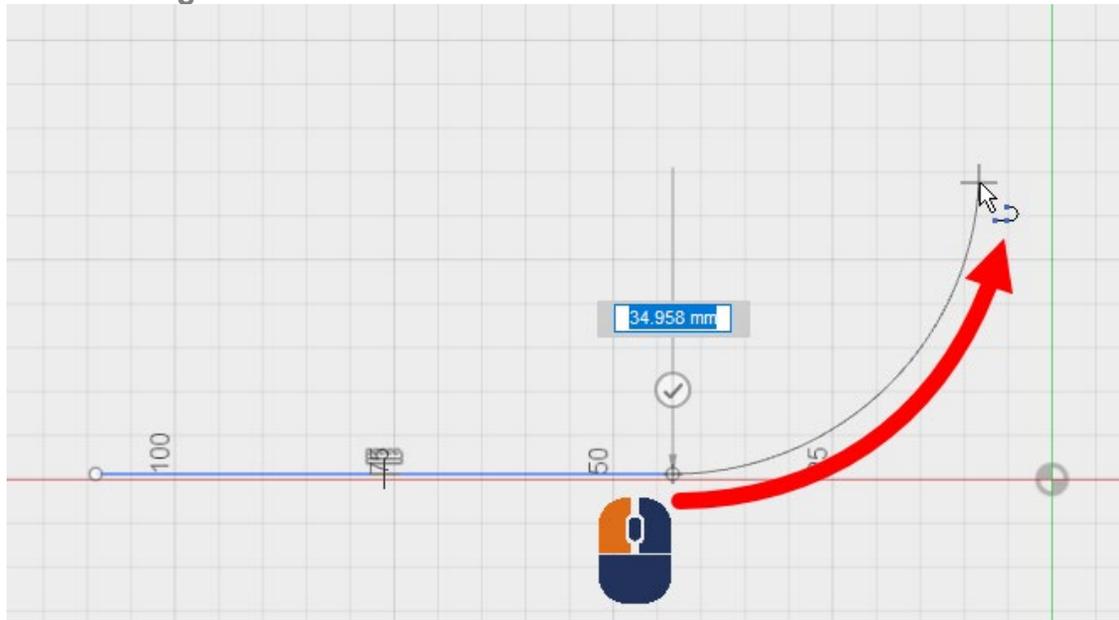
In his spare time, Scott runs [ChipX](#), a CNC tooling and workholding company selling products to machine shops and makers in New Zealand and Australia. His latest project is [Kiwiboard.IO](#), a collaboration between a few friends to design a completely custom, fully CNC machined low profile mechanical keyboard. The entire project including all the electronics is being designed and engineered in Fusion 360.

## Design Workspace Productivity Tips & Tricks

### Sketch

Since most CAD designers will spend a significant amount of time creating sketches, learning a few tips to speed up the process can have a huge impact on your workflow.

#### Click & Drag from Line to Arc



Fusion's Line command is actually a Polyline command, hence the icon. But to sketch out a polyline, you need to click twice to place the initial line, then click on the 2<sup>nd</sup> point of the line and drag away from it. In doing so you will see a tangent arc forming off the end of the line. Keep the left mouse button pressed until you are ready to place the end point of the arc. You can take advantage of snaps to reduce the amount of manipulation and constraints needed afterwards to make the arc fully defined.

#### Infer Sketch Constraints

Sometimes Fusion 360 wants to automatically apply/infer constraints to the incorrect adjacent sketch object, which can be annoying. However, you can influence which objects constraints are inferred between, by rubbing your cursor up against the object you want the current sketch object to be constrained to. This is something best shown in a [video](#).

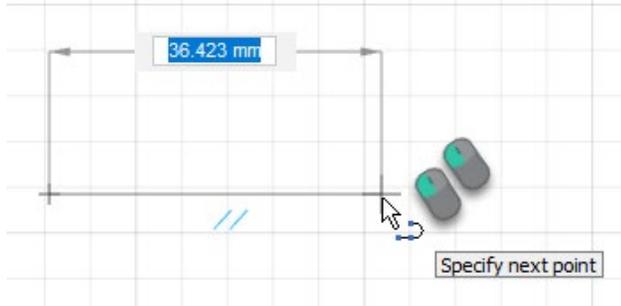
#### Ctrl / ⌘ to suppress constraint creation

By default Fusion 360 will try to project geometry through from the background into the current sketch to help define what you are currently sketching. While helpful, at times it's highly undesirable, especially when you have a complex sketch or model with a lot of features. The same is true with 'busy' sketches, Fusion may try to infer constraints to other sketch objects you don't want it to. The solution is to hold down the Ctrl / ⌘ key on your keyboard while sketching, this stops all constraints and projected constraints from being inferred/created. Check out [this video](#) for clarification.

### Double Click to end Line

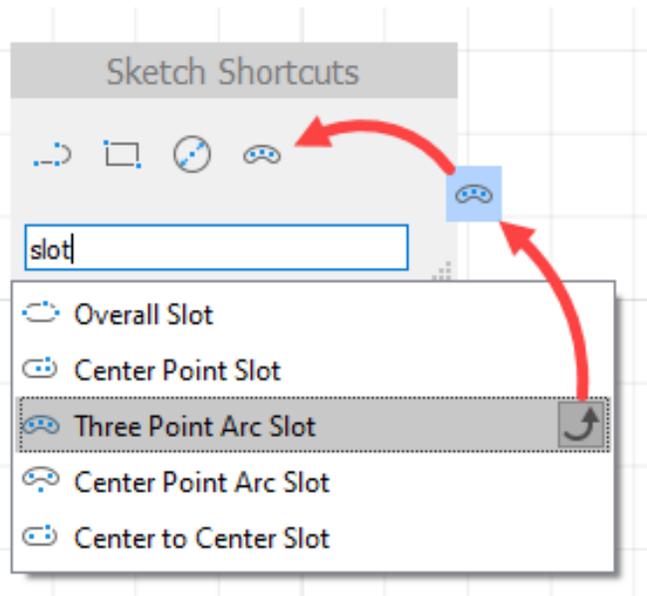


The green check mark is kind of annoying and often gets in the way, but there is a quicker way to end the current line without ending the Line command.

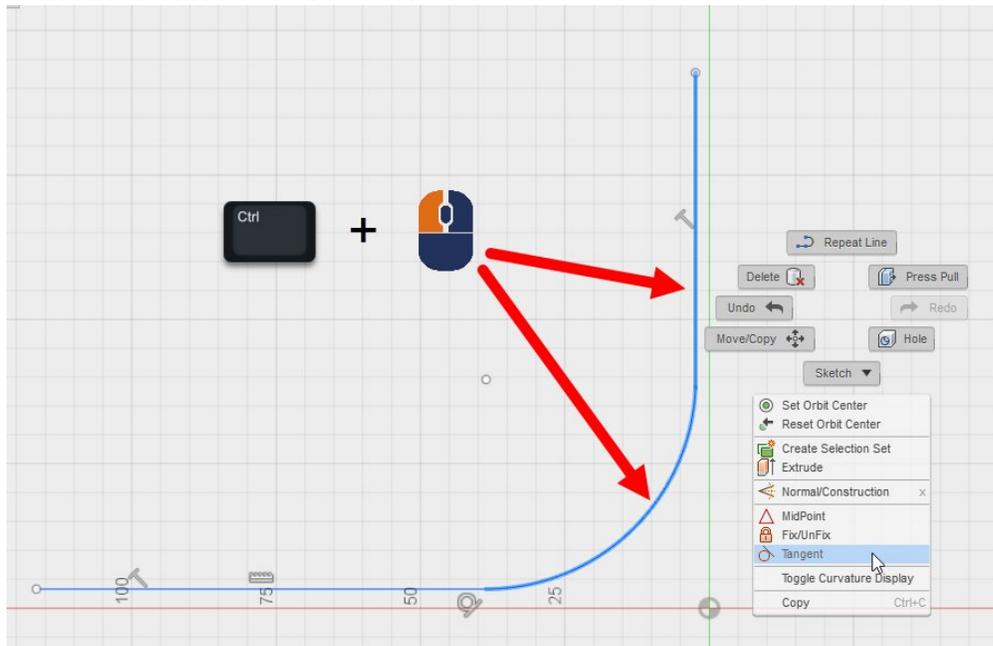


### S Key

The S key is the single most useful command in Fusion 360. Press it, and all of its commands are at your fingertips. Not only is it a filtering search tool, but it's also a contextual customisable toolbar. Meaning you can pin the commands you use the most in the Sketch environment, and they only show up while sketching. The same goes for each Workspace, they each have their own S key toolbox. To add commands, search for them, then click the curved arrow. To remove them, just drag them off the toolbox.



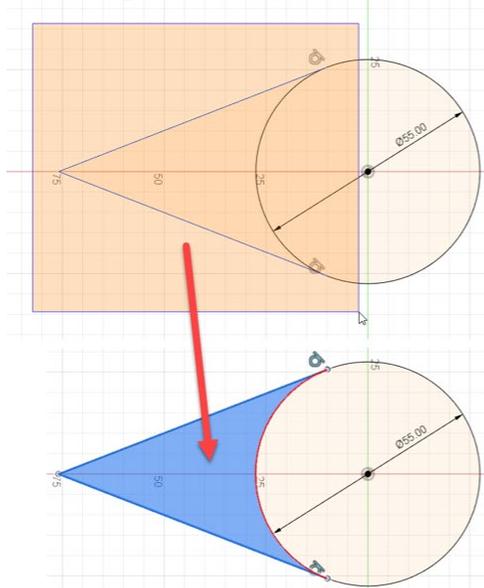
### Filtered Sketch Constraints



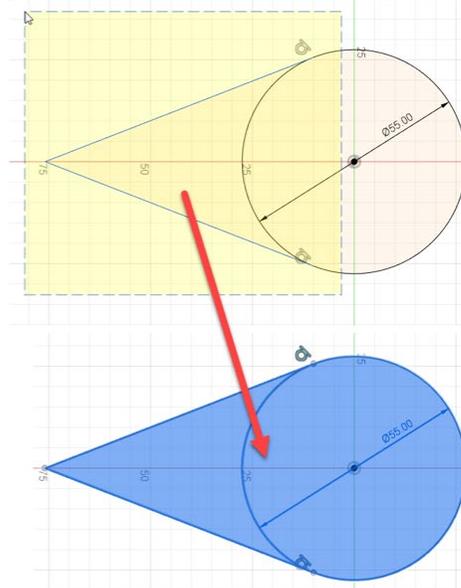
If you pre-select the sketch objects you want to constrain using the Ctrl / ⌘ key, then both the context menu & sketch pallet constraint list gets filtered to display only the valid constraints you could use for the current geometry selection.

### Window Selection

Wrapping your head around this tip will make your life a lot easier when it comes to selecting objects with finesse. When dragging a window from the top left to the bottom right, all selected objects will be fully enclosed within the window. Dragging from the bottom right to the top left will select all objects the window touches.



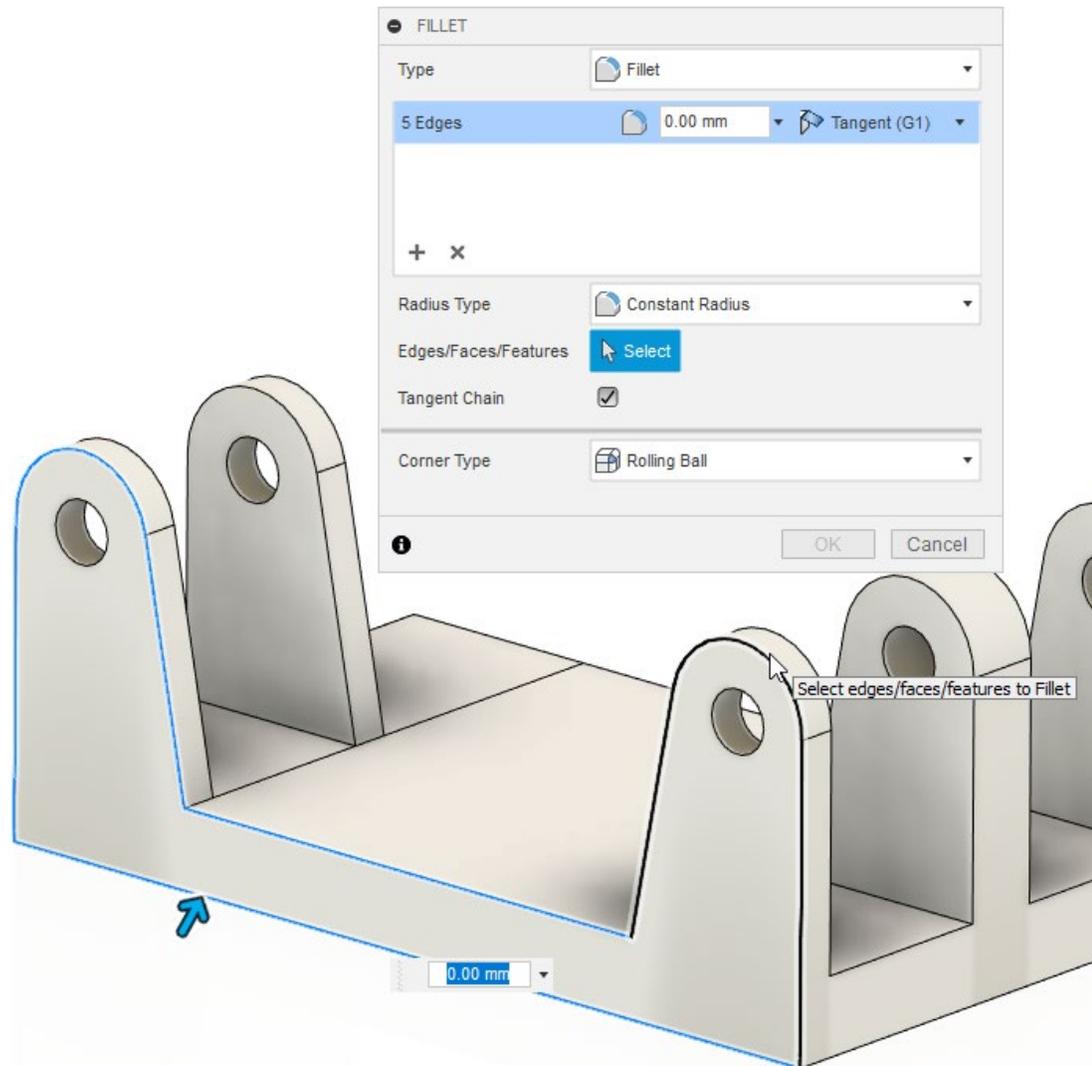
Top Left to Bottom Right / CW



Bottom Right to Top Left / CCW

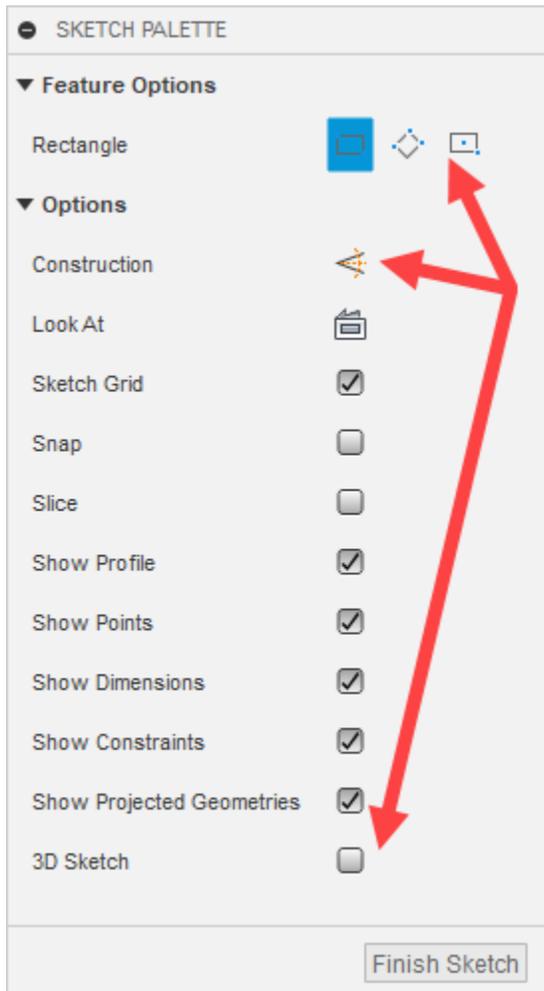
## Ctrl / ⌘ Selections

Using Ctrl on Windows and Command on a Mac is a very handy computing skill, but a lot of people either don't know about it, or don't realise they can use it in CAD as well. Their use in Fusion 360 is no exception. It can be used for selecting objects in the browser (Components or Toolpaths), the timeline, or in the graphics window.



Fusion 360 however does have a unique application for it in the Design Workspace. When editing an existing feature like a Fillet or Chamfer, you need to press and hold Ctrl/ ⌘ to add or remove additional selections.

In a sketching context, you would use the Ctrl / Command modifier to deselect or select additional objects the window selection method captured.



### Sketch Pallet Modifiers

The sketch pallet in Fusion 360 is a hybrid of the old school pre-ribbon & new school context sensitive ribbon user interfaces. AutoCAD also uses pallets extensively. But this sketch pallet in Fusion 360 is highly context aware, and constantly adjusts the options available based on the active command or objects selected in the graphics area.

The *Feature Options* section will allow you to switch between rectangle, arc, circle, slot & polygon types, without having to go and select it from the ribbon sub menu system or search for it via the *S* key.

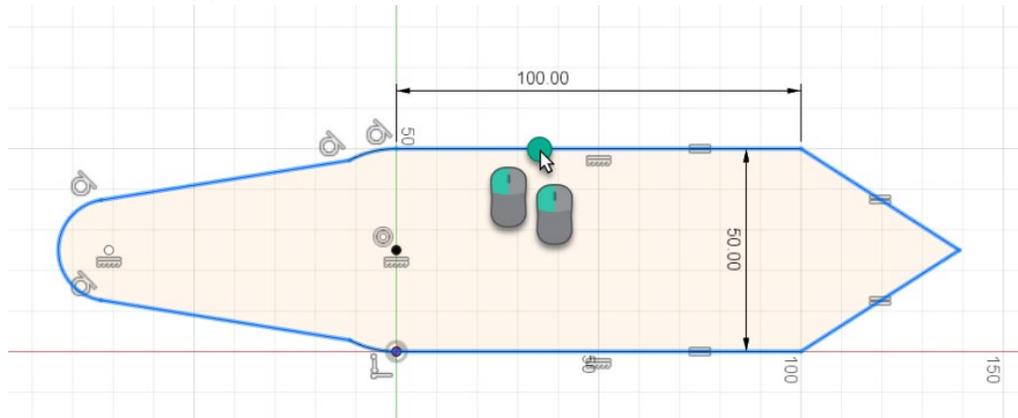
The *Options* section allows you to turn some very useful things on & off.... Is your sketch getting too busy? Then turn off all the Constraints. Setting sketch geometry as Construction is also something you should be making a habit of doing if you aren't already, since it makes your sketches more lightweight. Construction geometry isn't included in profile calculation for modelling commands, it also makes your sketches a lot easier to look at, allowing you to focus on the profile geometry which really matters.

### Midpoint Selection

If you want to dimension to the midpoint of a line/arc or create a constraint to the midpoint of a line/arc. Then you just need to hold down the SHIFT key to activate the midpoint snap. Once your cursor reaches the midpoint of the line or arc, a triangle icon with a coincident constraint X will appear. Click to create the constraint or dimension from that snap, once created a black point will appear at the midpoint.

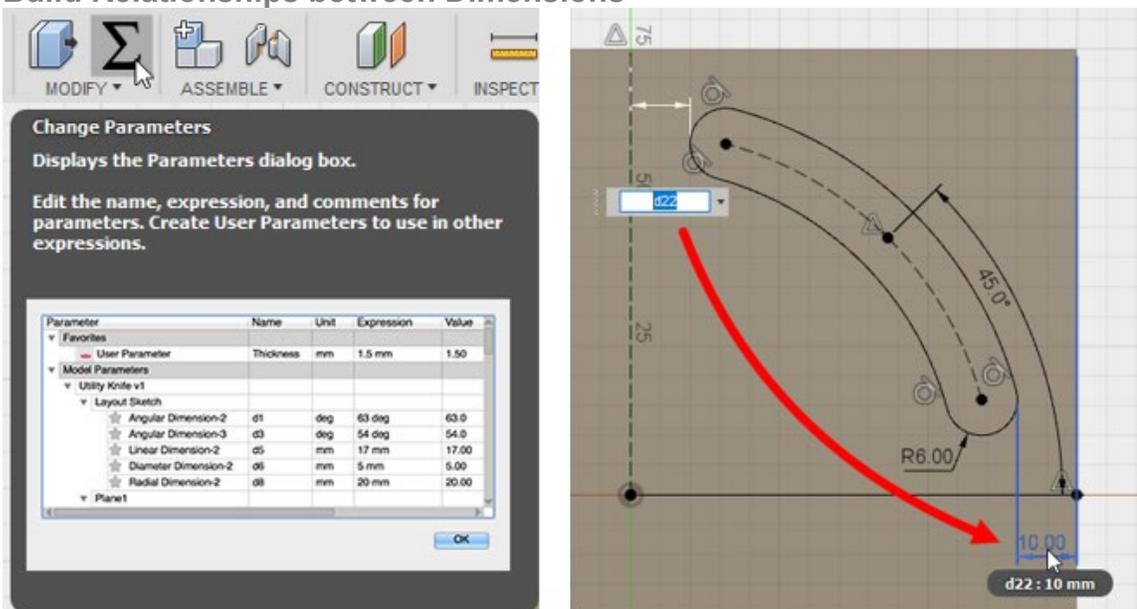


## Double Click to select Chain



With no command active, double click a sketch element, and Fusion will select the entire edge chain around the profile. Chained loops respect the line type, so making one of the sketch elements of a construction type will interrupt the loop. You can effectively 'group' sketch geometry for chain selection by using construction geometry.

## Build Relationships between Dimensions



**Change Parameters**  
Displays the Parameters dialog box.

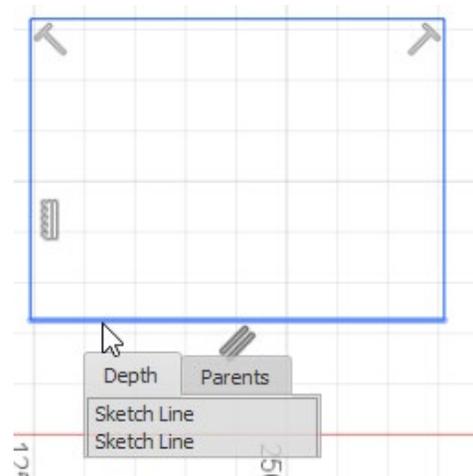
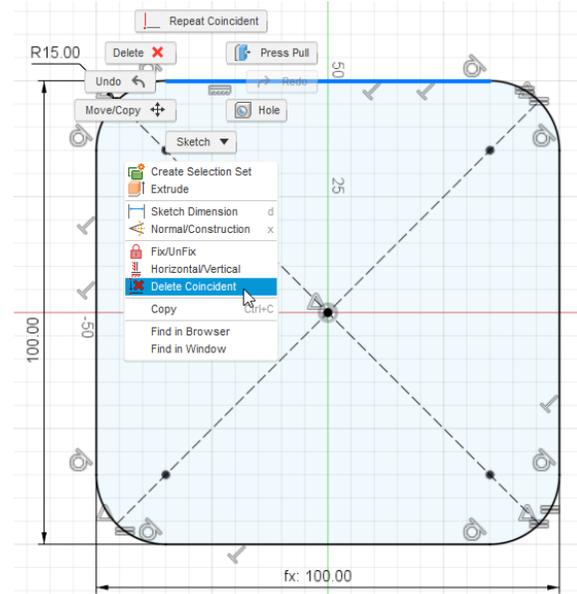
Edit the name, expression, and comments for parameters. Create User Parameters to use in other expressions.

| Parameter        | Name                 | Unit | Expression | Value |
|------------------|----------------------|------|------------|-------|
| Favorites        |                      |      |            |       |
| Model Parameters |                      |      |            |       |
| Utility Knife v1 |                      |      |            |       |
| Layout Sketch    |                      |      |            |       |
|                  | Angular Dimension-2  | deg  | 63 deg     | 63.0  |
|                  | Angular Dimension-3  | deg  | 54 deg     | 54.0  |
|                  | Linear Dimension-2   | mm   | 17 mm      | 17.00 |
|                  | Diameter Dimension-2 | mm   | 5 mm       | 5.00  |
|                  | Radial Dimension-2   | mm   | 20 mm      | 20.00 |
| Planet           |                      |      |            |       |

To build intelligent models in Fusion 360, you have to learn how to leverage parameters to their fullest extent. You can easily make one dimension/parameter equal to another, by selecting the dimension on screen that you want it to equal. This will use the name of the parameter driving that dimension, instead of having to type in a duplicate numeric value. What's more, you can use all those mad math skills you learnt in school! [Fusion supports a full complement of algebraic operators](#) for you to take advantage of. Think about design intent when doing this... do you always want one feature to be a 1/3 of the size of another. Yes? Then just type in  $parametername/3$ . Done! Be sure to check out all the functions you have available as well, because you can work out the min/max number of pattern instances you can fit within a certain distance etc... this stuff is SUPER powerful.

### Delete Coincident Constraint

If you want to detach the end of a line or arc from another sketch object, right click on the side of the object you would like to detach. In other words, if you want to detach the left hand side of the line from an arc, right click on the left hand half of the line. In the context menu, select **Delete Coincident**. Then left-click and drag the line/arc away from it's neighbouring object.



### White Points & Select Other

With the base sketch active, if you don't see any white points, then make sure the Show Points checkbox is active on the Sketch Pallet. This setting is sticky within each sketch, so it's possible for it to be ON in one sketch but OFF in another sketch. If you still don't see any white points, then it's possible you have two identical lines/arcs sitting on top of one another, and fully attached at both ends with coincident constraints. To check suspect geometry, click and hold it with your left mouse button so the Select Other dialog appears, then you can choose to delete one of the surplus elements.

### Black Points

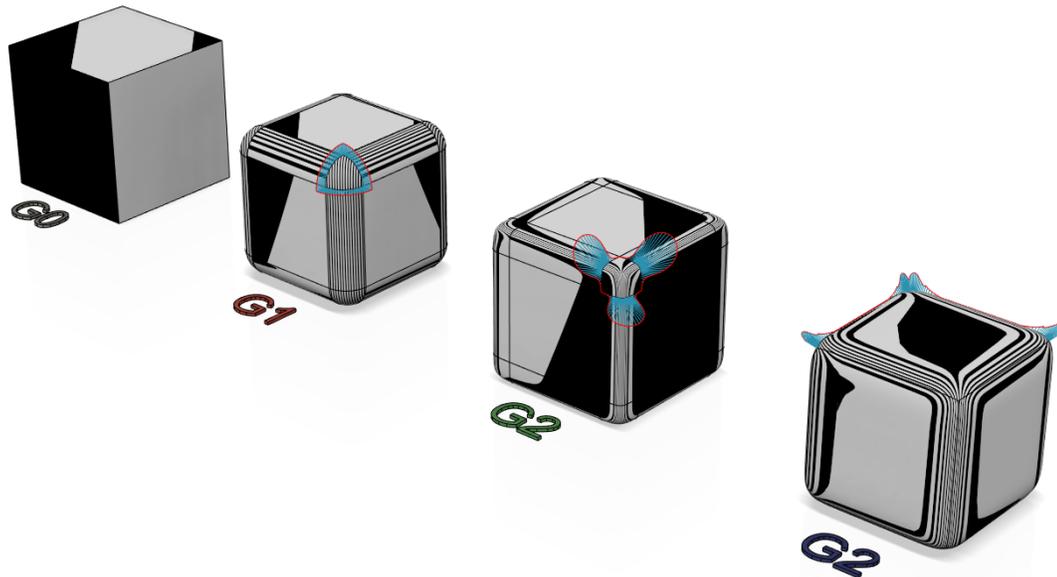
What if you see black points in your sketch? While white points show unconstrained points, black points can be fully constrained duplicate points. You might see this if you place a sketch point at an intersection of several lines/arcs, or you have created a relationship to the midpoint of a line/arc or if a placed sketch point is coincident with a line/arc. Two other places you will see black points is at the center of a fully constrained circle, or the control points on a fit point spline.

### Modelling

Solid modelling is perceived to be easier than surface modelling, but it really depends on the task at hand. For most people's workflows this will be true, it's not until you need absolute control over your geometry when G2 or G3 continuity conditions are called for do you need to start exploring surface modelling techniques. Nonetheless there are increasingly more tools and

controls being added to the solid modelling toolset in Fusion 360 which will take care of a lot of G2 scenarios.

### Wut up G?



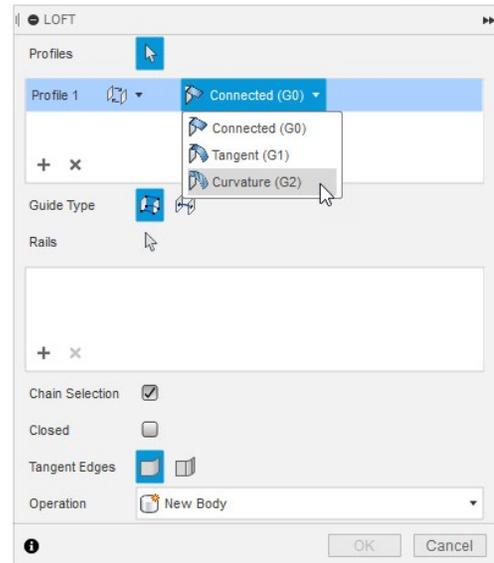
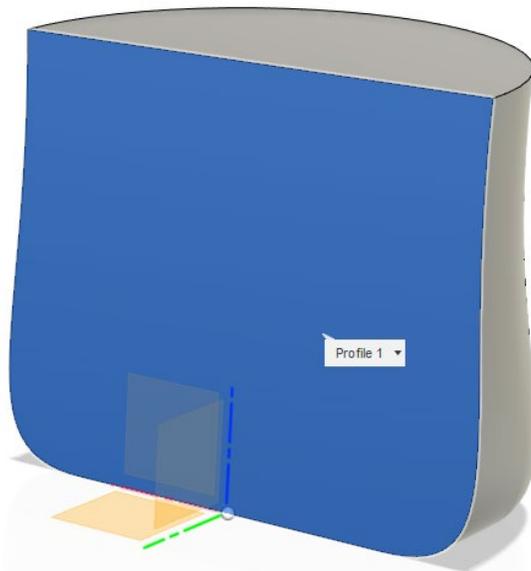
In the above example you can see 4 distinctly different ways to model a cube with varying levels of smoothness. The first 3 are normal cubes, one doesn't have a fillet (G0 Connected Faces), the next one has a standard fillet (G1 Tangent), the 3<sup>rd</sup> cube along has a curvature continuous fillet (G2 Curvature), then the last cube is a T-Spline body where curvature continuity flows across and between all faces.

Paul Munford does a much better job than I could of explaining this in his 2017 Fusion 360 Autodesk University Class, [Complex Topology and Class-A Surface Modeling with Fusion 360](#). Make sure you go and check it out.

Why do we care? Because in nature it's rare that G0 & G1 conditions exist. G2 & G3 would be far more common and they show up quite obviously when light strikes those surfaces. So when it comes to rendering realistic images, modelling with G2 curvature continuity in mind will help out tremendously. In my context it's very useful in making sure my ice cubes look more realistic.

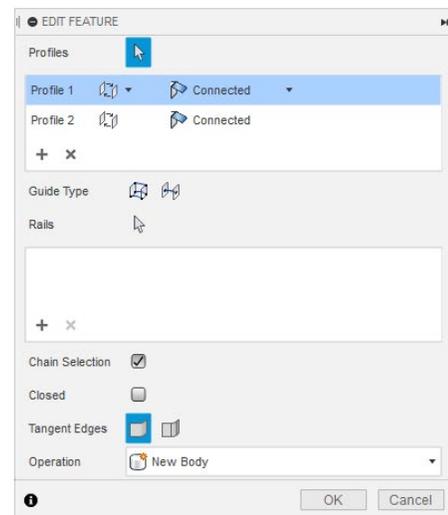
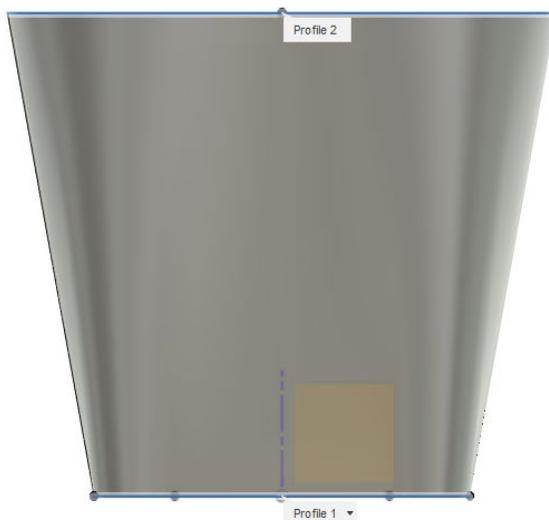
### Lofty Potential

Loft is an extremely powerful modelling command with a lot of options, some hidden away until you make just the right selections. The G2 curvature condition won't appear unless the profile is based on valid geometry, since a G2 condition can't be calculated unless there are adjacent faces to do so.



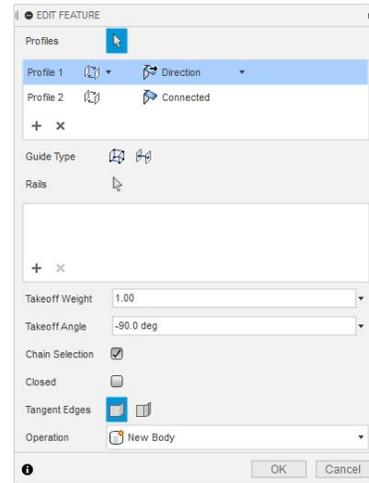
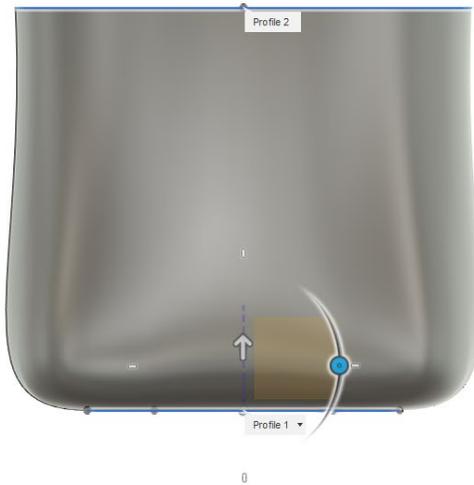
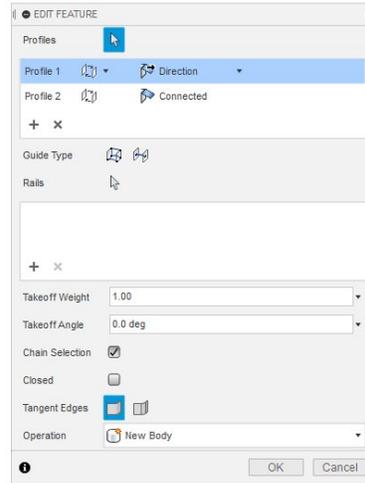
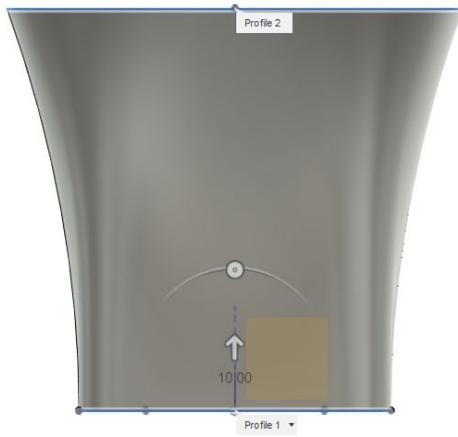
For example selecting the face of a solid for one of the profiles activates the G2 Curvature option. The same is also true for the G0 & G1 conditions.

Compare this to lofting from sketch profiles:

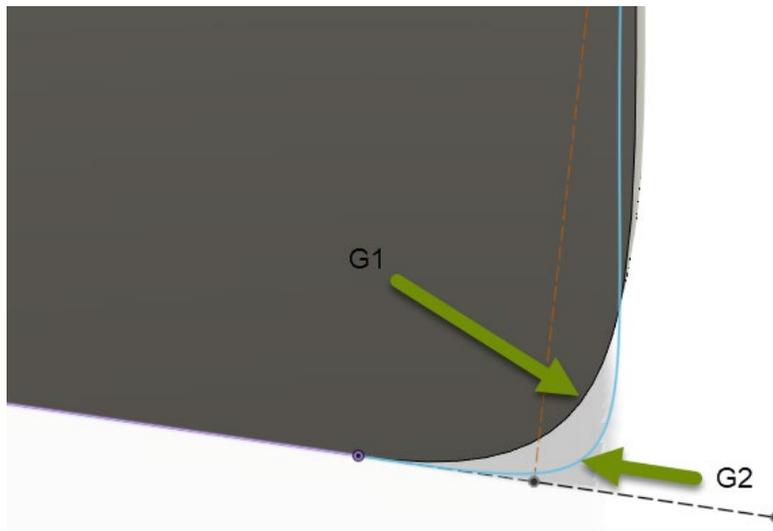


The default option is Connected, but not G0, because the profile must be connected to the resulting faces created by the loft.

However, you can also select Direction as the termination method for the face where it meets the selected profile. When you couple this with the Takeoff Angle value, you can dramatically alter the final shape of your Loft.



With exactly the same input profiles, we get two totally different looking tumblers.

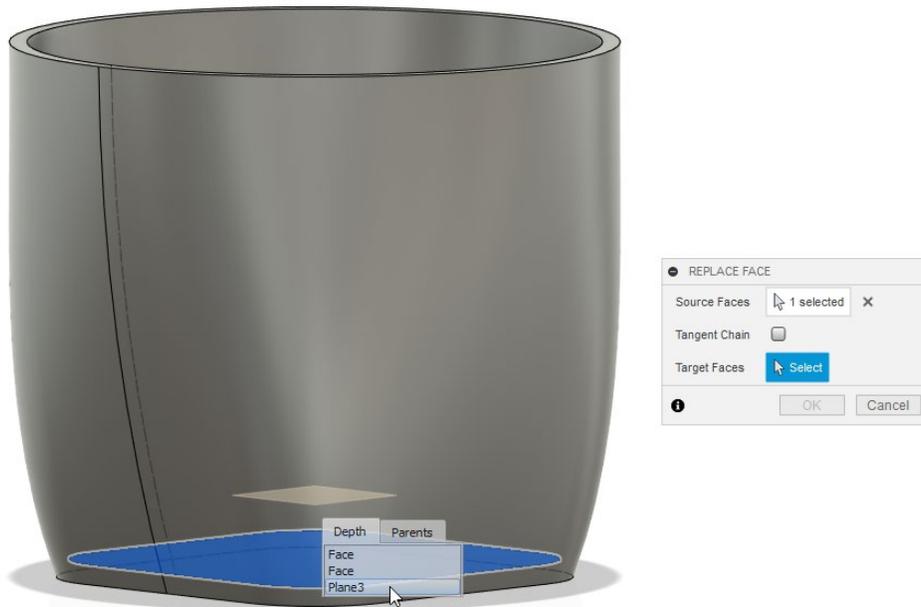


It's tempting to think that at a 90 degree takeoff angle, that the result is a G2 transition, but it is only a G1.

I've applied a G2 Curvature sketch constraint to the blue spline so you can see the difference.

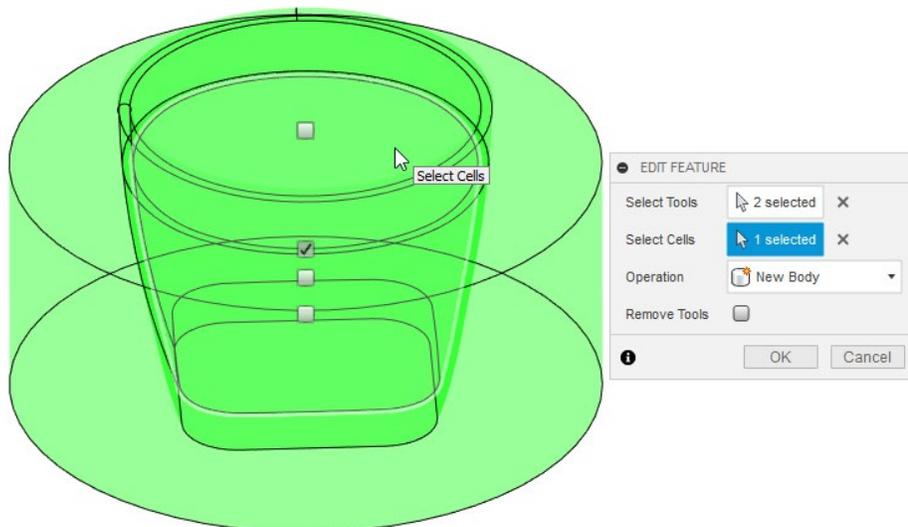
### Replace Face with Plane

You don't have to have another Face modelled to replace and existing face with. You can simply use a plane. In this case the face also gets larger as it projects up the instead of the glass.



Offset face would also work here, but I wanted to parametrically drive the overall thickness of the glass' base, which would have been a little harder to do.

### Boundary Fill using Solids

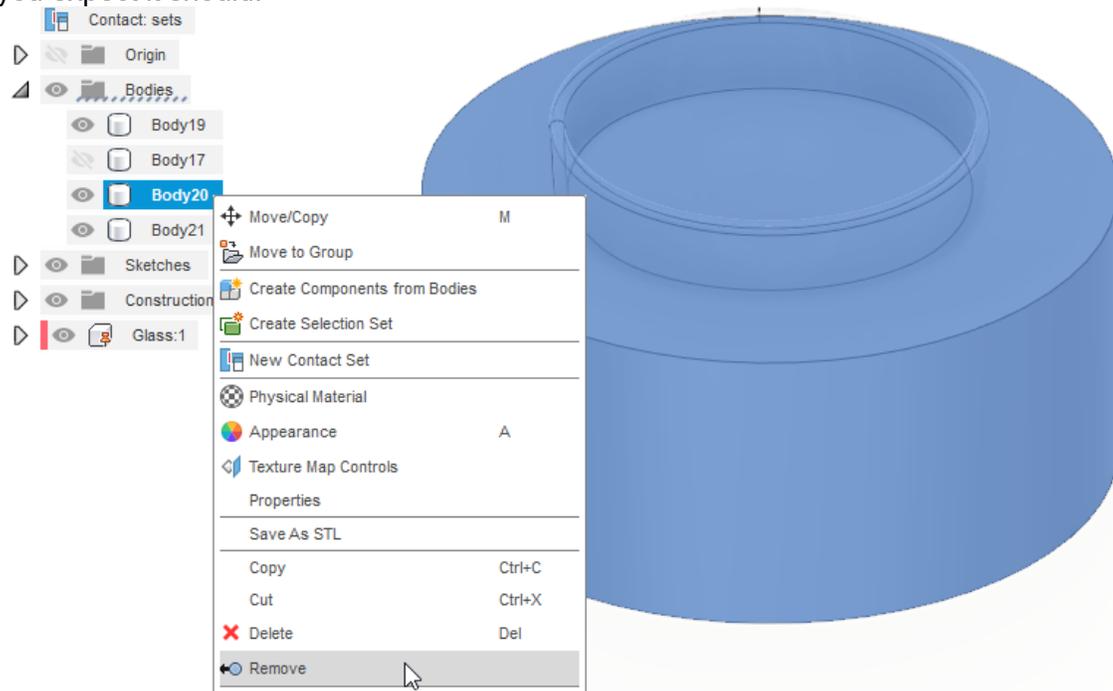


Basically, you don't have to use a group of surfaces to use boundary fill, two intersecting solids will give you what you need... top tip, it's easier to directly select or use window

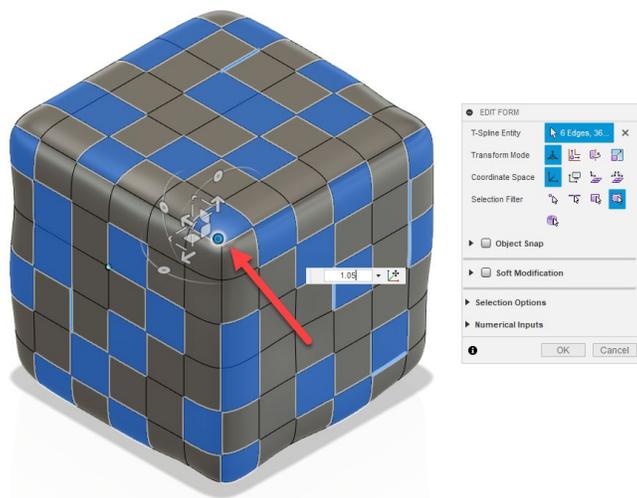
selection to specify which bounded volumes you want to keep. The checkboxes aren't easy to use and you can often have several of them on top of one another without you realising that's the case.

### Remove Body

It's weird... but if you want to delete a body that's surplus to requirements, the only option in the Ribbon to do so is the Delete command in the Modify panel. The trouble is that really messes things up in the Timeline because it doesn't respect history, deleting the body will pull the rug out from under any features which use the body... cue the Remove command in the bodies context menu... use this and everything will work as you expect it should.



### Random T-Spline Body Scaling

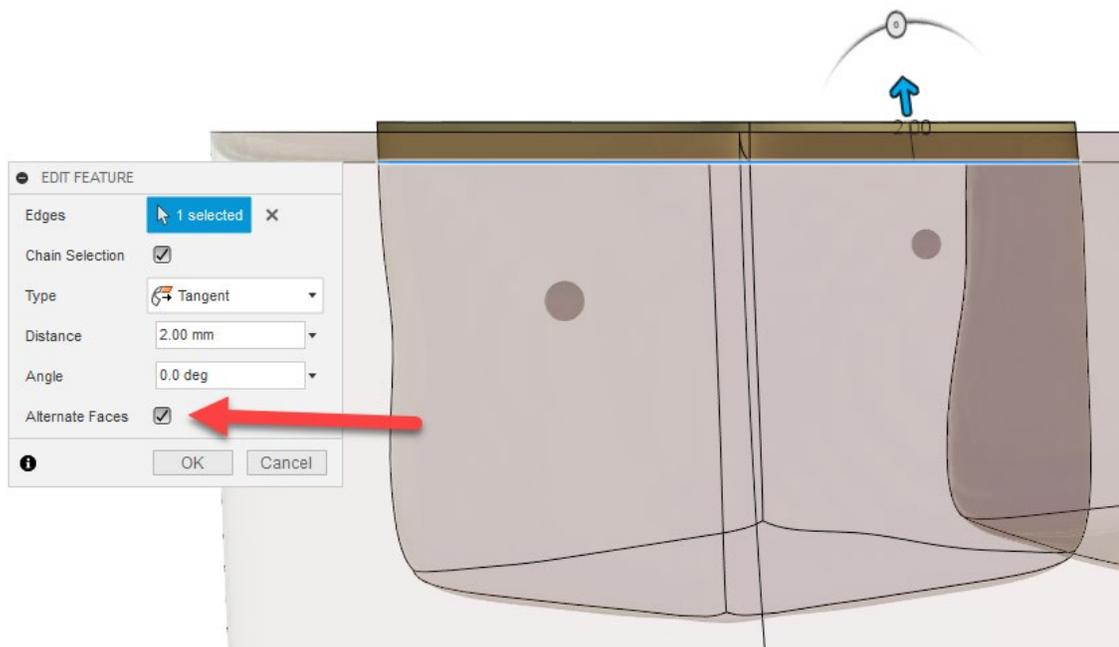


If you randomly select faces, edges and vertices on a T-Spline body, then select the Manipulator origin. This will apply your chosen scaling value to every object in all directions about the Manipulator origin, creating a degree of randomness in the resulting geometry. In the case of an ice cube it's quite effective.

### Alternate Faces Ruled Surface

To create a realistic top surface of fluid in a glass, you have to model the meniscus. This is relatively easy around the perimeter of the glass, but is harder where the fluid meets the organic surfaces of the ice cube. The ice cubes are also sitting in the fluid at a compound angle to add realism. This presents a challenge for creating a fillet of some kind. But you can't model a fillet between something and nothing. I decided to overmodel some surfaces and create a G2 fillet between them.

Ruled surfaces are perfect for this, I had a hard edge on the top of my fluid where I had subtracted the ice cubes. Which meant I could 'pull' off a surface. The trouble was when I tried it was filling up the hole left by the ice cube.. selecting Alternate Faces saved the day.



## Components

### Pattern Joint Origins

Joint Origins can't be patterned unfortunately. But sketch points can... since patterns can generate a large number of objects very quickly, placing Joints on each sketch point would be laborious. In comes the Fusion 360 API... using a Python Script you can automate the placement of a Joint at every sketch point in a selection.

I knew I could write a script to do it, but Patrick Rainsberry asked me for a tip to put into his Fusion 360 API Tips class, so I asked him if he had a script lying around I could use... because he's a machine, he banged one out in no time. You can download it from [here](#) or in the additional materials section of my AU class listing.

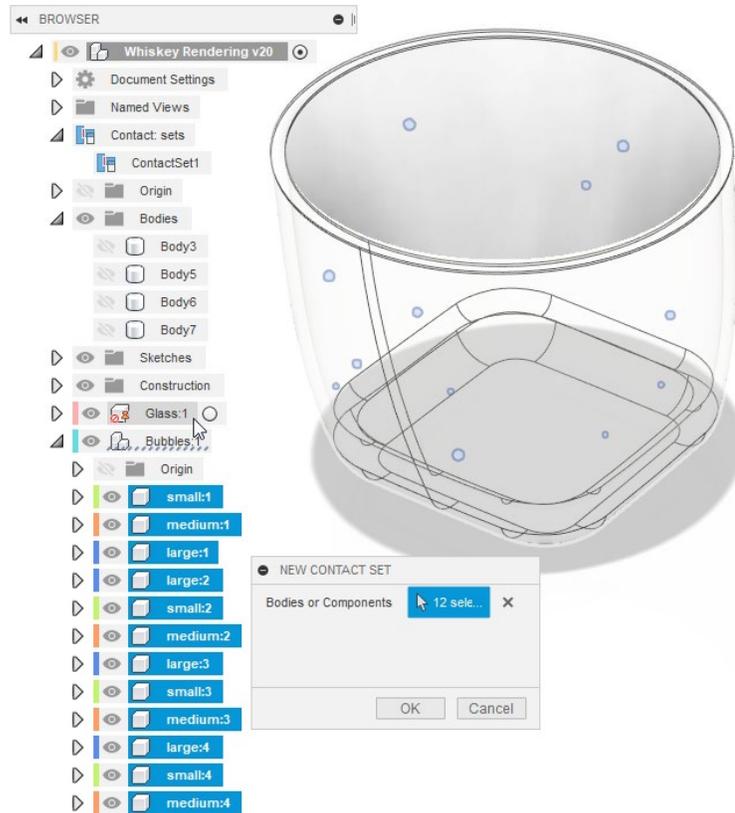
To use it, just pre-select all the points you would like to place Joints onto, then run the script.

### Unconventional use of Contact Sets

Typically Contact Sets are used to limit the motion and check mechanical behaviour or clearances. For my renderings I wanted to include some air bubbles inside my fluid, typically bubbles attach themselves to the side of the glass or ice cubes. Precisely and randomly positioning spheres around my assembly would be extremely tricky to achieve manually. So I decided to try and use Contact Sets... and it worked a treat.

I created an individual component for each bubble, and placed them all within a parent component. This meant I could select all the bubble components at once by selecting the parent 'bubble' component.

The Contact Set was made up of the glass and all the individual bubble components. From there I enabled Contact sets, then dragged out all the bubbles from their modelled position until they contacted with the glass.



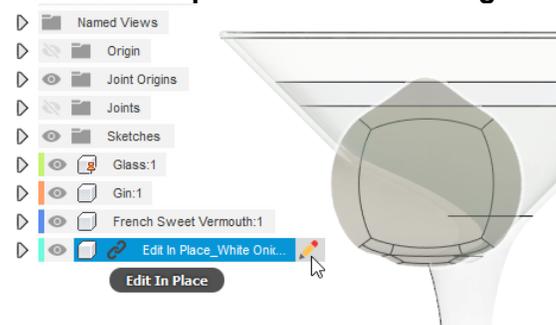
### Edit In Place (EIP) / Assembly Context

This handy new feature delivered by the development team earlier in 2020, makes it extremely easy to make modifications to standalone Components/Designs in the context of their placement with a parent assembly. I used it twice in my Cocktail assembly

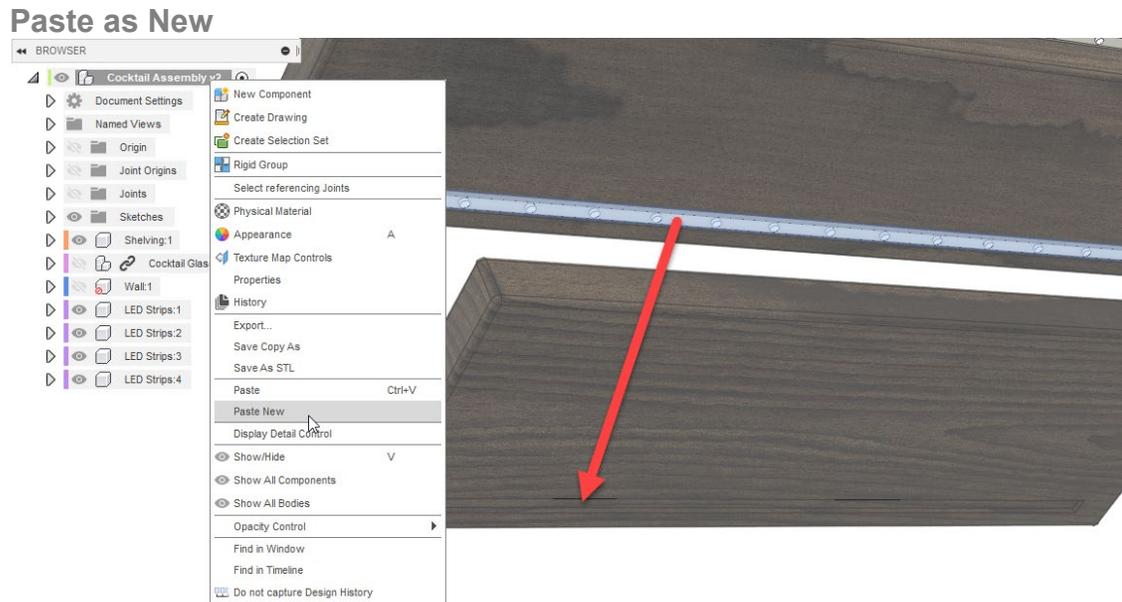
#### To cutout the glass from the lemon



#### Scale the T-Spline onion to fit the glass



## Reusing your models and drawings



If you Copy a Component, then Paste it, you create a second instance of that Component in your Design. Effectively resulting in a quantity of 2 in your Bill of Materials for that Component. As a result, if you modify anything in either of those instances, the rest of them will also update.

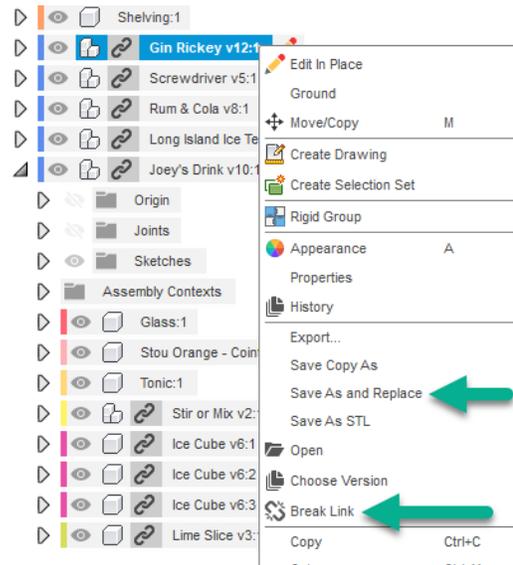
However, if you right-click on on a Component and select Paste New from the context menu, it will create a new independent copy. Upon inspection of the Timeline (if you are Capture History), you will see the new Component has full set of features all of it's own. Meaning if you make a change to the original Component, the changes won't propogate to this new Component.

Paste New is a quick way to create lots of similar Components in your Design.

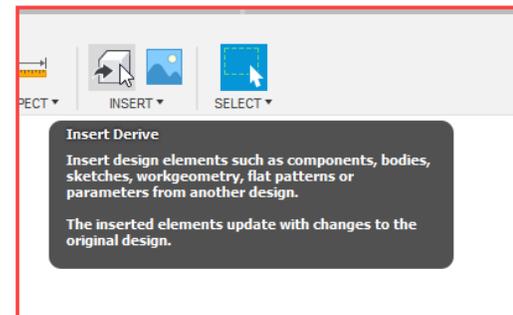
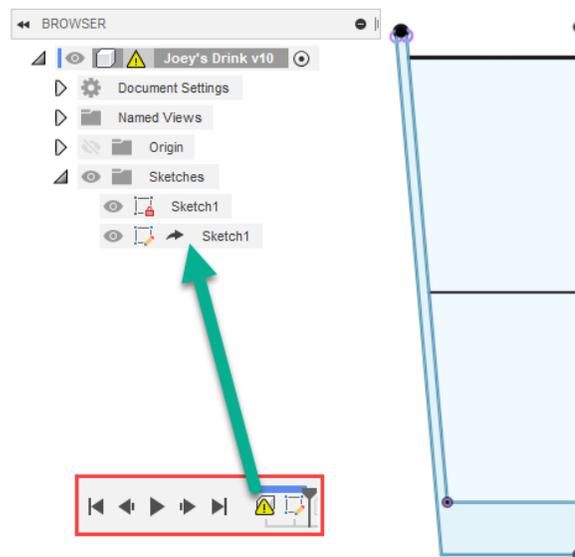
## Referenced / XREF / Distributed Design

There are a few workflows you can use for manipulating Components inserted into a Design document.

- Maintaining links mean any updates to the source Design are pushed into any Designs that use it.
- Breaking the links copies the component and all of it's features into the current Design.
- Save As & Replace well maintain a link but create a copy allowing a variation to be created



## Insert Derive

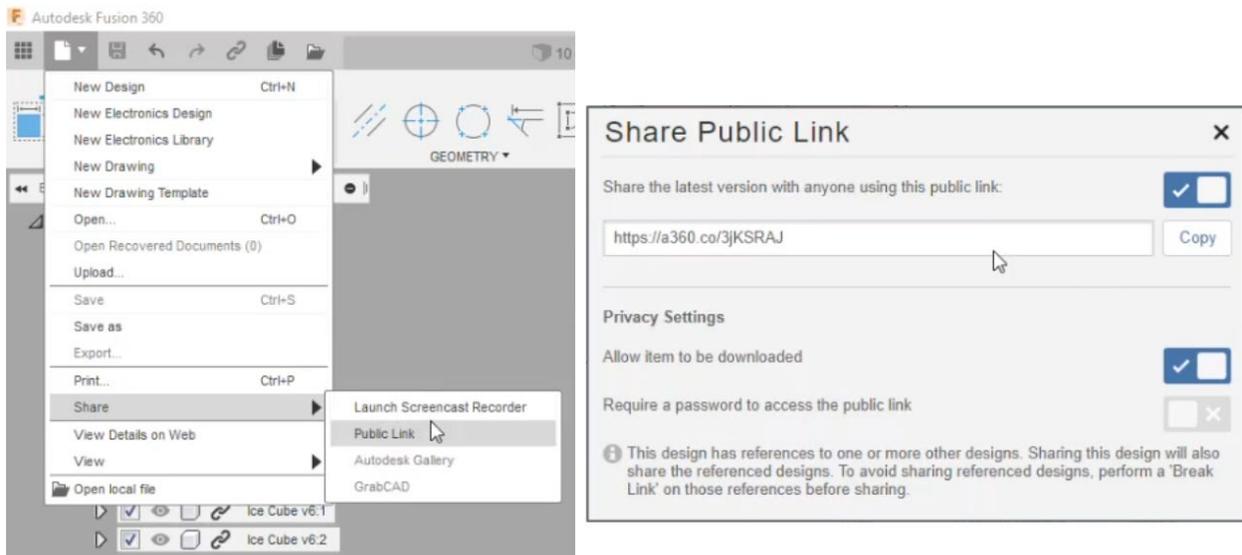


Derive allows you to tie a series of Designs together to a source set of design parameters or features, but still provides flexibility for variation across Designs. In my example I've pulled through a base sketch to use for revolving the Glass in multiple Design files. This means I can control the size, shape and volume of the glass in each Cocktail design from a single source. But it still allows me to parametrically control the height of each fluid ingredient so we have the correct amount of each fluid for each Cocktail recipe. Effectively enabling 'mass customisation' from a core set of standards.

## Copying Drawings with your 3D Model

If you have created drawings for your model and you would prefer not to have to recreate those over and over again for each design variation. Then you have two options.

1. Reduce the drawing effort by using Smart Templates, you can refer to my AU Class from 2018 if you would like to learn more about those.
2. Copy your drawing by creating a public link and downloading a full dataset, then re-upload it via Fusion 360.
  - a. Create a public link and allow downloads

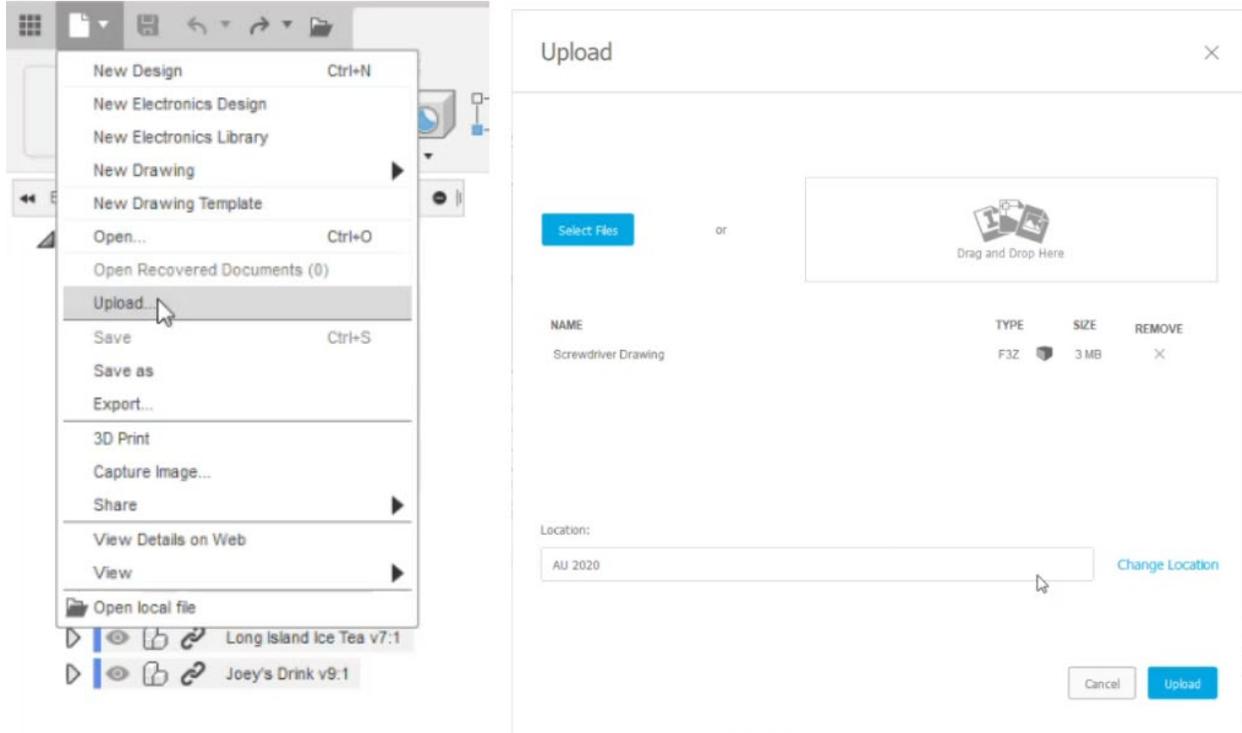


- b. Open the link in your browser and request a Fusion 360 archive download link to be emailed to you.

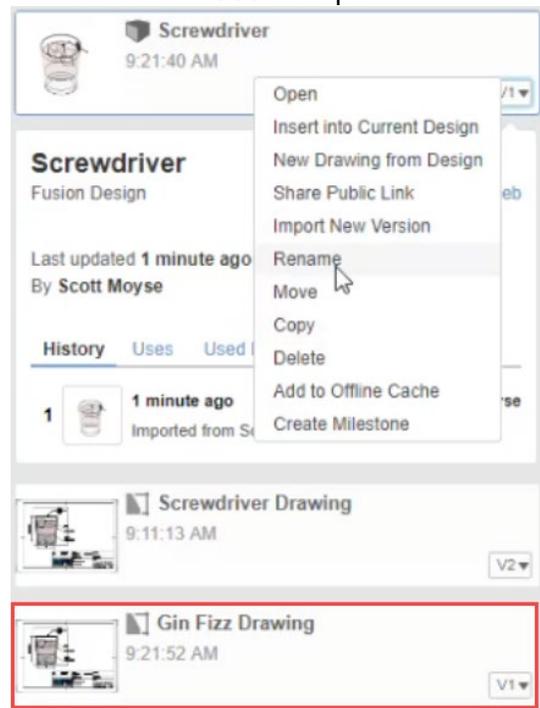


- c. Use the link in your email to download the archive file

d. Upload the archive file via Fusion 360 to the Project and folder of your choice.



e. Rename the Drawing & any associated files you would like to rename from the Fusion 360 data panel.

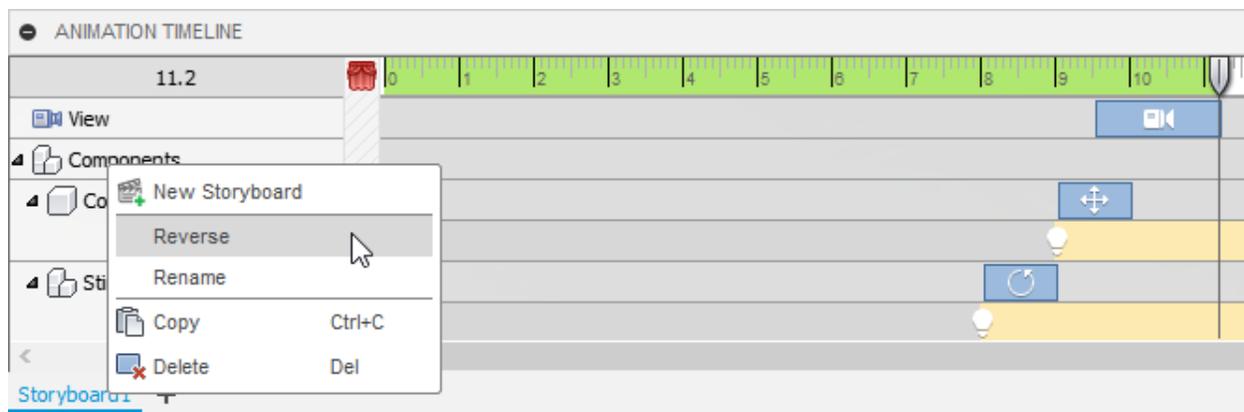


## Tell a story in the Animation Workspace

The animation workspace is used to create exploded views to place onto your drawings, so you can then balloon them for parts list identification of your parts. Typically this process is effectively disassembling your Design.

### Reverse

You can also save out the animation as a video, allowing you to publish some maintenance instructions onto a website somewhere. But what if you want to show someone how to assemble your Design? Well there's a handy Reverse command in the context menu for each storyboard.



## Pouring transparency into your renderings.

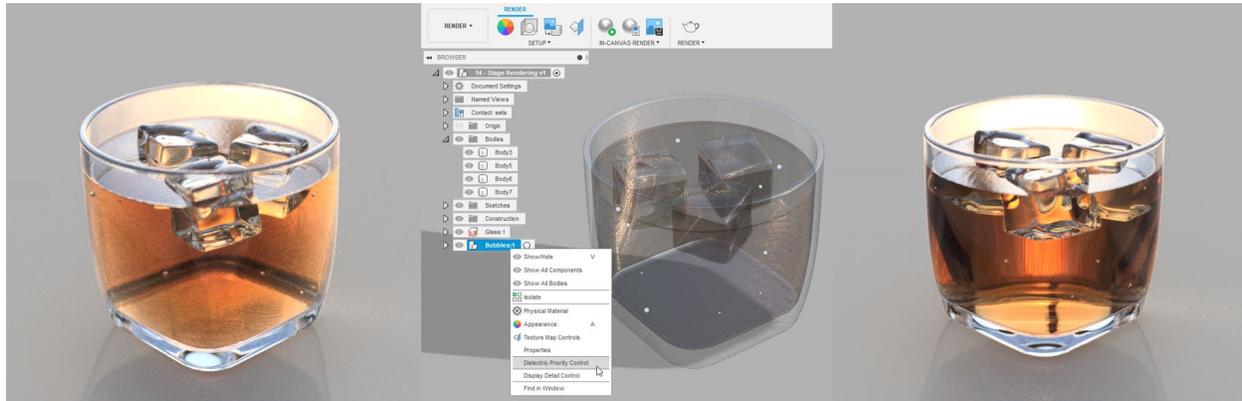
### Intersecting liquid and glass



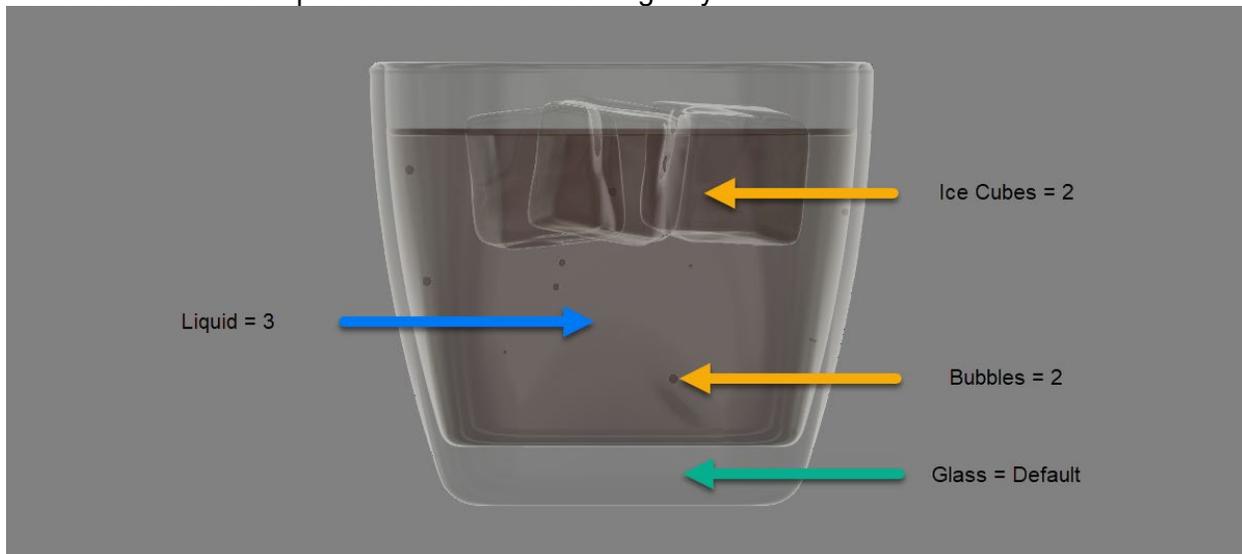
There's a conflict between engineering modelling and modelling for renders. Generally we avoid interferences in engineering design. But if the components use transparent materials, then when we render them artifacts appear. Pressing Q to launch the Press Pull command, is a simple way to add some additional material to one of the bodies to create interference. The result of doing so can be compared between the image on the left vs the right.

If you have to do this with an engineered design and creating an interference is going to cause problems. Then use the Derive command to push your Components through to a new Design, where you will be able to make modifications for rendering and staging purposes.

## Nested dielectrics



Dielectric priority determines the rendering order from an optical perspective. The image on the left has all components and bodies set to the default order. As a result the bubble on the left hand side of the glass looks like it's on the outside of the glass, and the ice cubes aren't being distorted by the glass and liquid when they should be. Whereas, the right hand rendering looks more realistic with the priorities set in the following way:



Aradhana Vaidya wrote a post on the Autodesk Fusion 360 blog 3 years ago, explaining [how to create realistic renderings with transparent materials](#). In the post he goes into more detail about why rendering with dielectric priorities set can be beneficial.

### Stage the product

Generally, engineered models aren't that visually interesting. They lack the detail our mind's eye needs to convince us they are real. Just like how real estate agents like to have your house staged for their photoshoots, you also need to give your designs a fighting chance by staging them in a good light. Add additional detail to your models that you normally wouldn't take the time to do.

- If your design contains liquids, make sure you model the meniscus. Surface tension on the top of the fluid will either result in the fluid being pulled up or down the side of the vessel or objects inserted into it. Some fluids have a positive meniscus and others have a negative one. The light in the rendering will catch this detail, making the appearance of the fluid more believable.
- Would there be bubbles in your liquid? If so, make sure you add some. I didn't add any in my ice cubes and it shows. They look too perfect.



- Add some props. Would there be furniture nearby? What about windows or tables? Pens, pencils, paper and books etc.

## Lighting



- You can add additional lights, which can also include bodies with emissive appearances applied. The default lighting in the Fusion 360 Rendering workspace comes from the environment lighting. This often isn't enough, since in real life there are normally multiple light sources. Especially if a product is staged by a photographer, then they will use artificial lighting to set the mood and present the product in its best light. You can replicate that in Fusion 360 by accurately modelling additional light sources and emitting light from them.
- Fusion 360 has some built in HDRI backgrounds, these provide the aforementioned environment lighting, but can also provide a backdrop to set the scene. It's possible to use your own HDRI backgrounds, but they have to be the correct format. Here are a couple of great resources to read through to learn about how to convert some free HDRI backgrounds into formats supported by Fusion 360.

[Creating Custom Environment for Rendering in Fusion 360](#)

[Paul Debevec – Light Probe Gallery](#)

[HDR Shop – Free for Non-Commercial use](#)

## Additional References

I can't claim that all these ideas are mine. I've picked up all sorts over the years from various different people and sources. I thank everyone who has helped me in the past and in the future. There are a couple of videos which really helped me when trying to figure out a few things with this class, so here are some links if you are interested:

<https://www.youtube.com/watch?v=qh5MwJzxlGE>

<https://www.youtube.com/watch?v=n6ZsTApTRK4>

Then here are some additional links which contained interesting information which I didn't get to include.

<https://www.youtube.com/watch?v=et5K9G216uo>

<https://www.autodesk.com/products/fusion-360/blog/camera-settings-make-your-renderings-look-more-realistic/>

Paul Sohi provides from insight into how some of the appearance properties work in this blog post:

<https://www.autodesk.com/products/fusion-360/blog/10-things-you-didnt-know-fusion-360-can-do/>

And if you want to check out the dataset... you can download a full copy of it using this link, it might be a familiar workflow to you having watched this class 😊:

<https://a360.co/3opWGih>