AUTODESK. UNIVERSITY

CP226398

101 Fusion 360 Tips & Tricks

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Learning Objectives

- Acquire additional modeling skills specifically related to “The Fusion 360 Way”
- Learn how to develop a better understanding of Fusion 360 shortcuts and keyboard modifiers
- Learn the nuts and bolts of Fusion 360 assemblies
- Learn how to configure Fusion 360 to suit you and your workflow

Description

Everyone loves a quick-fire class full of productivity and power-user tips and tricks. Fusion 360 software is a big product, with lots of environments to master. No matter if you are a beginner or a seasoned veteran, there are always new tips to learn from different perspectives. Come along for an action-packed 90 minutes of learning and accelerate the process of mastering Fusion 360.

Speaker(s)

CAD Manager/Applications Engineer with a reputation of continuous improvement and innovation via a thorough understanding of manufacturing design processes, software and team management. Expertise in 3D CAD technologies, PDM & PLM, CAM programming, CNC machine operation, process documentation, technical training/support and R&D. A team leader who’s very active with self-improvement via online learning and collaborating with his peers. Well known for sharing his knowledge effectively via multiple forms of media. Experienced with managing diverse teams with an International project portfolio. A decade of experience working on luxury projects for High Net Worth Individuals. Hugely passionate about Formula One both as a sport and an engineering discipline.

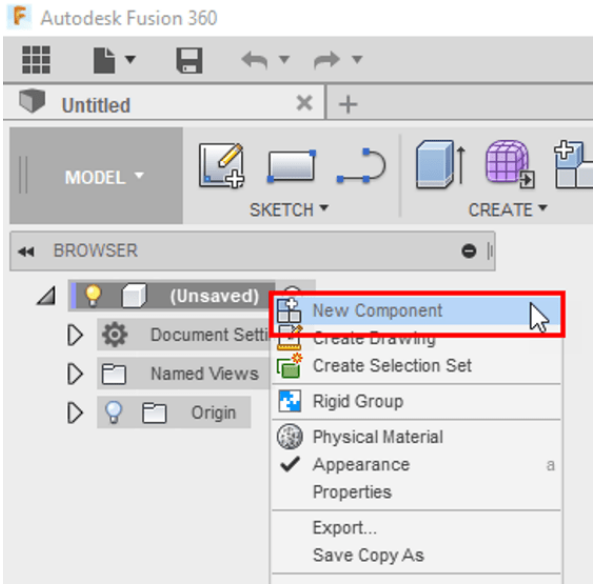
General & User Interface Fusion 360 Tips

This section will cover tips which will help you throughout Fusion 360.

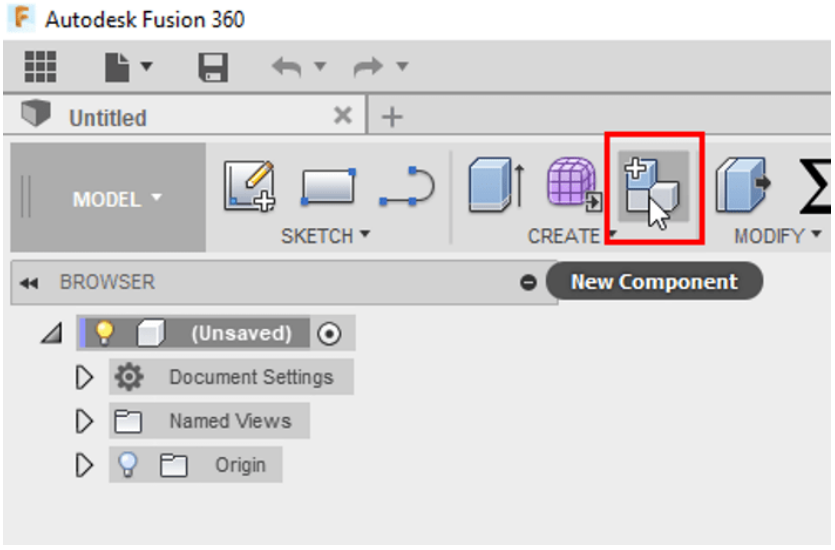
1. New Component

One of the most important tips you can know when starting out with Fusion 360, is to create a New Component before you do anything else. This is true 95% of the time. If you are only designing a single component, like a bolt, then there is no need. But if you wanted to model a nut, bolt and a washer, you need to create one component per item. Doing it first as a matter of habit, will save you a lot of extra work later on, and result in a far more organised design. Fusion 360's new component command can be accessed two different ways:

1. Right Click on Top Level Node in the browser



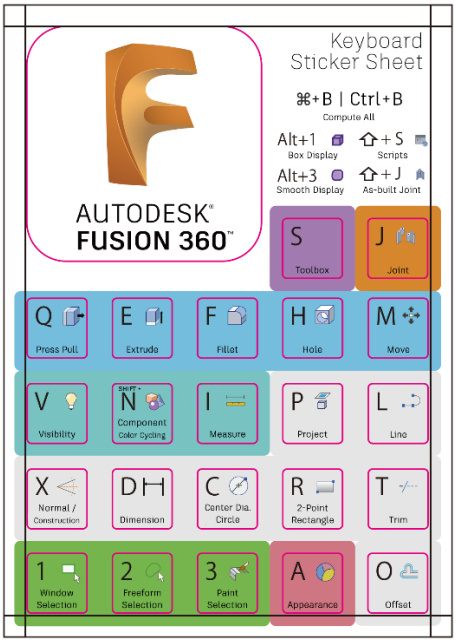
2. Select the command from the Ribbon

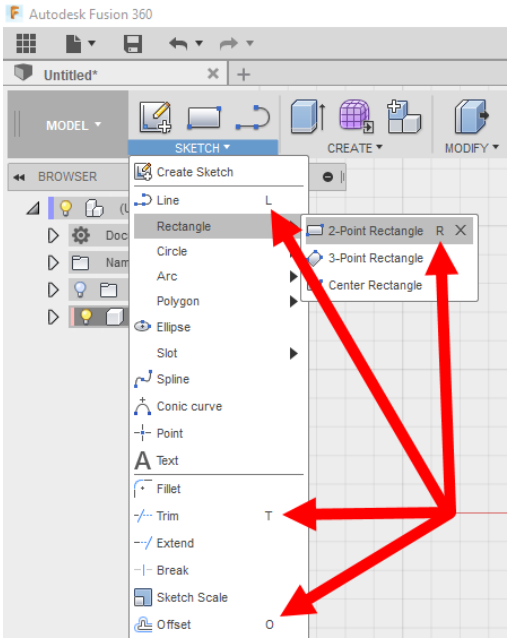


Just remember, if you want to create additional Components, think about your assembly structure. You may not always want the top level to be the parent, you can also right click on an existing component, and create a New Component inside that one instead. As soon as one Component contains another component, it becomes an assembly instead of a part.

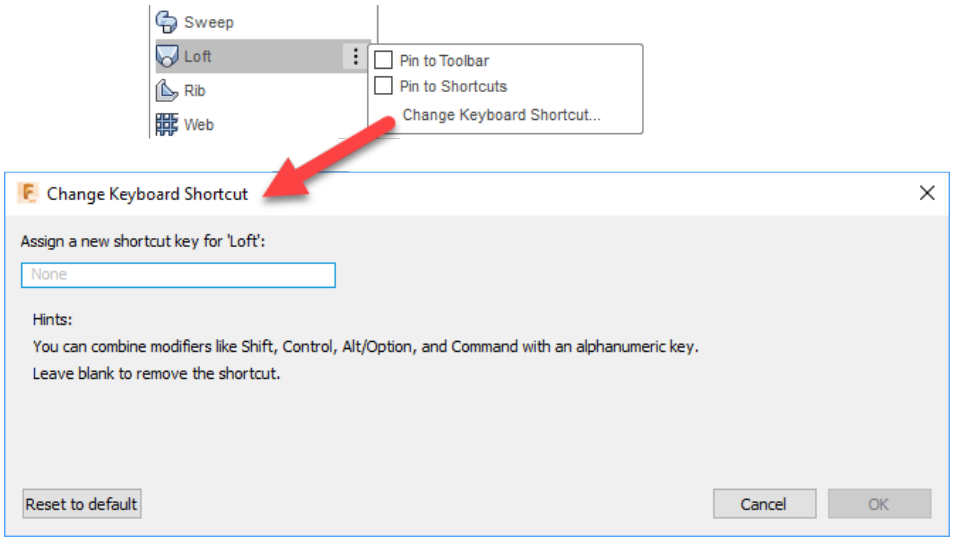
2 . Keyboard Shortcuts

Autodesk Fusion 360's keyboard shortcuts are now customisable, but the original default shortcuts also remain.





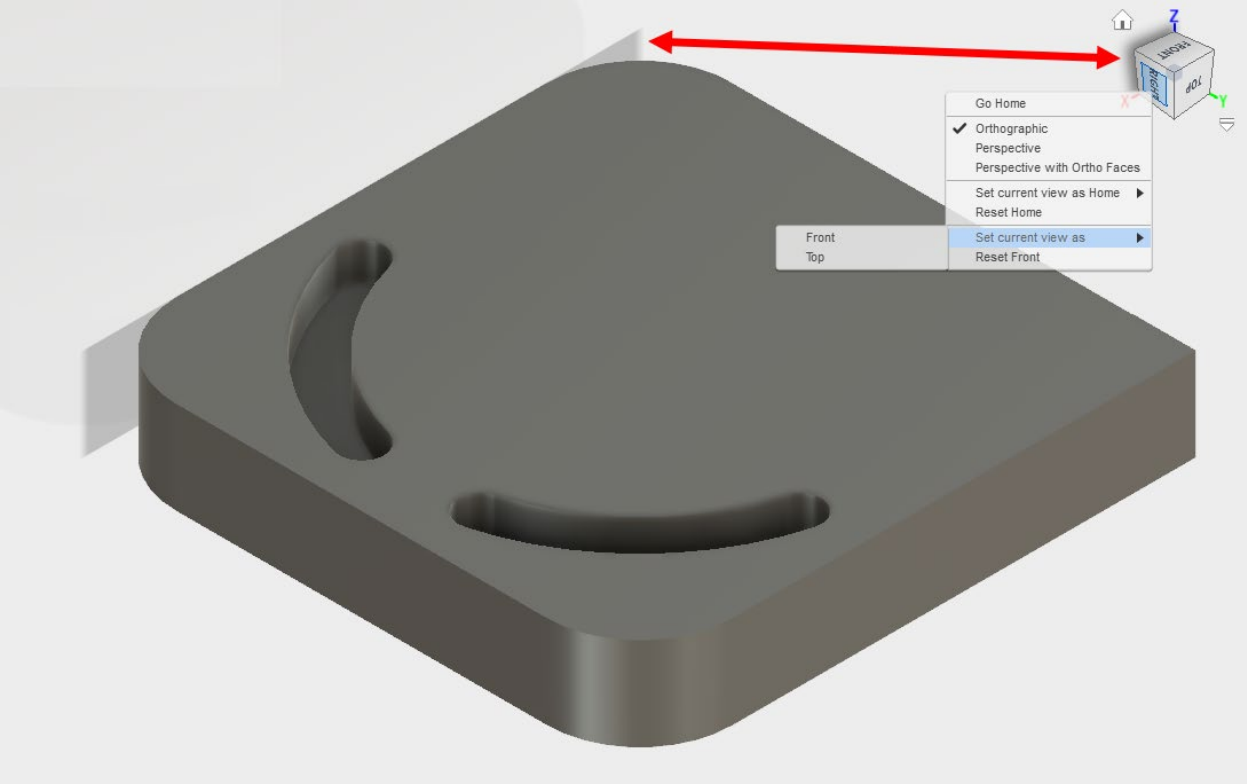
Earlier in 2018 the Fusion developers released support for customisable shortcuts, you can now setup a shortcut for any command by clicking on the ellipsis button next to each command in the menus.



3. Set Current View As

The view cube is an essential user interface element in Fusion 360. You must become familiar with it, and how it impacts your workflow to truly feel comfortable when using Fusion. Visually and functionally it's very similar to what you will find in many other Autodesk CAD products. The view cube influences the following things:

1. Your lighting orientation… take a look at the shadow on the view cube and compare it to the shadow and lighting on your model.
2. If you prefer to use Constrained Orbit (which is the default orbit type), then setting the 'Top' correctly will make Orbiting much more predictable.
3. View Cube settings are independent between the Model and CAM/Manufacture workspaces. Allowing you to configure your views differently to suit your needs.
4. Drawing views can also be chosen based on the view cube faces.

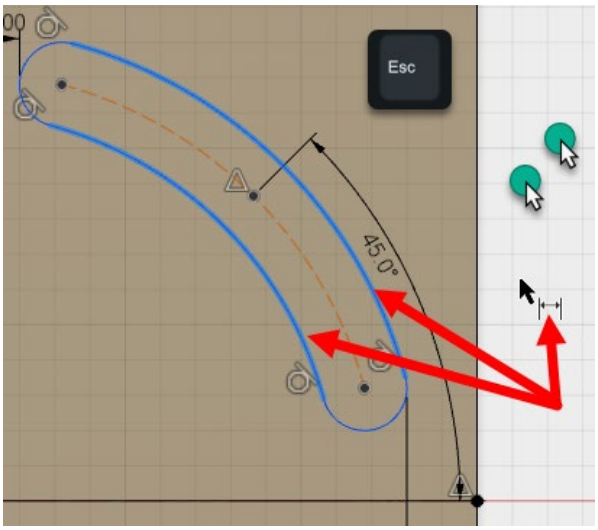


4. Zoom Fit / Extents

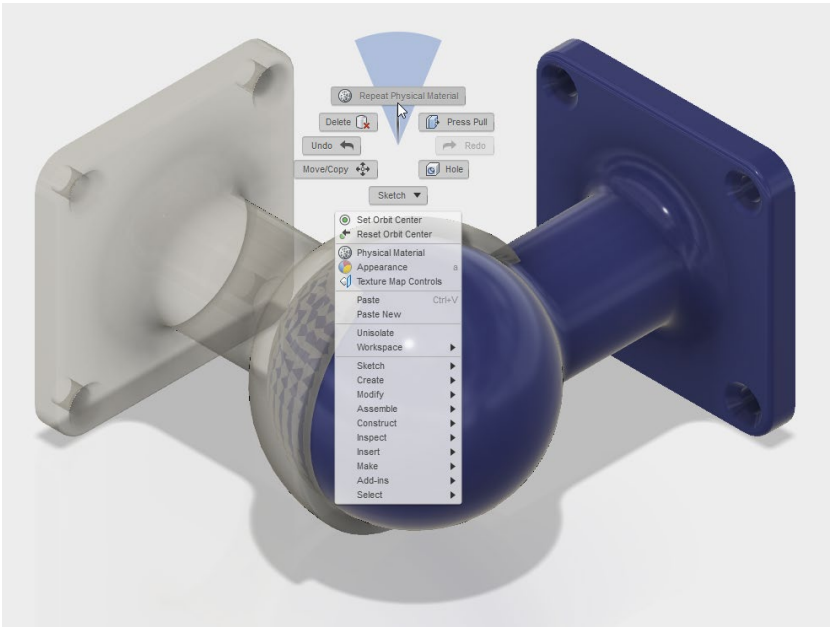
It's pretty easy to orbit your model right off your screen. Especially if you are in the early days of navigating around 3D CAD packages. It happens to even the most seasoned CADders. But there's an awesome trick which works in most CAD packages. Double click your middle mouse button, and your entire model will zoom to fit back onto your screen.

5. Escape & Left Click

Just like Ctrl + S for saving should be a nervous twitch for most CAD users. In Fusion 360 one of the things I find myself doing often; is pressing the Escape key on the keyboard and left clicking repeatedly on free space in the background. This ensures there aren't any commands active and no objects are selected which could be causing Fusion to behave oddly. A good example is having an arc or point selected before you start to dimension between two straight edges. In this scenario, you will be expecting to see a linear or angular dimension, but instead you get a radial one.



6. Repeat Last Command

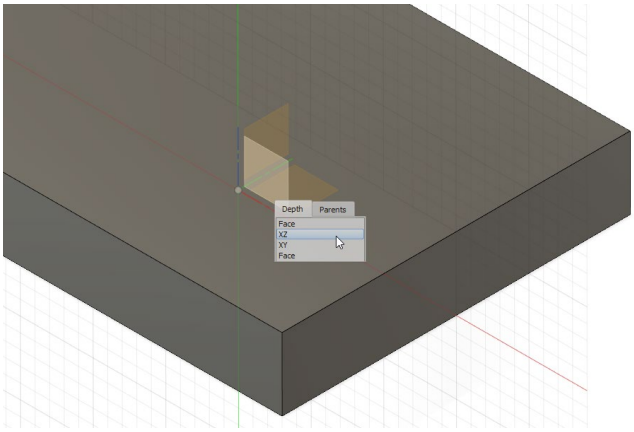


An essential part of any CAD modellers toolset is knowing how to repeat your last command. Fusion 360 relies on mouse gestures to achieve this. With no command active, right click, hold and move your mouse forward. This will sweep up through the 12 o'clock portion of the context menu.

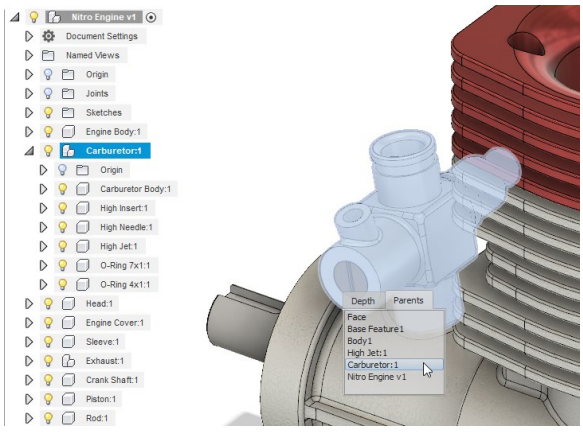
As an added bonus, save yourself some mouse movement, by using swipe right to select OK, instead of moving across to the command dialog to select the OK button.

7. Select Other / Parents

This tip can be useful right at the beginning of a design when you want to place a section sketch on a different WCS plane, after modelling the base feature. Or when you want to select the parent assembly of a component in the graphics window. Just left click and hold, without moving your cursor, and a two tab dialog will pop up at the tip of your cursor.



SELECT OCCLUDED OBJECT

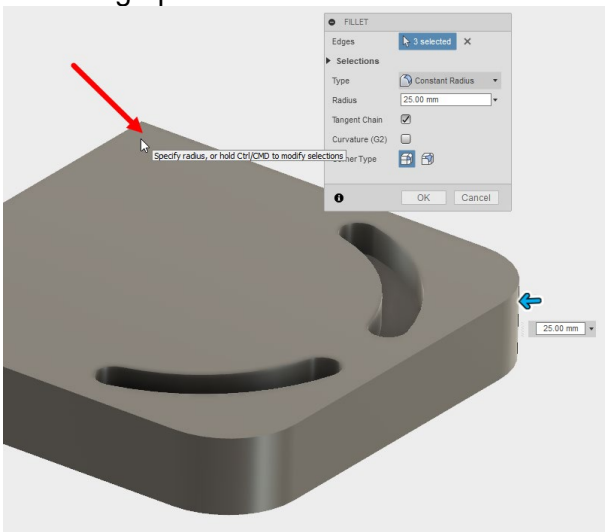


SELECT PARENT OBJECT

Select the item you want and continue with your work. This works with and without a command active.

8. Ctrl / ⌘ Selections

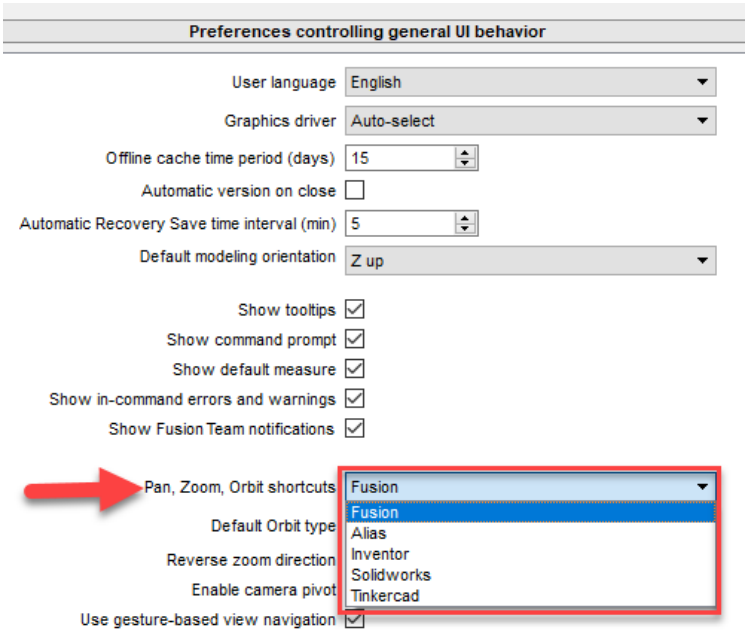
Using Ctrl on Windows and Command on a Mac is a very handy computing skill, but a lot of people either don't know about it, or don't realise they can use it in CAD as well. Their use in Fusion 360 is no exception. It can be used for selecting objects in the browser (Components or Toolpaths), the timeline,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

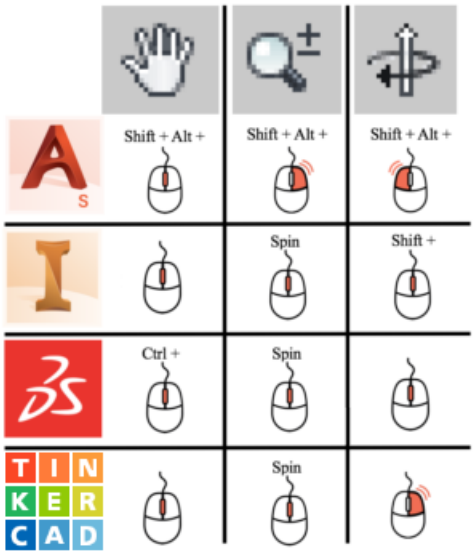


Fusion 360 however does have a unique application for it in the modelling workspace. When editing an existing feature like a Fillet or Chamfer, you need to press and hold Ctrl/ ⌘ to add or remove additional selections.

9. Mouse Pan, Zoom, Orbit Preferences

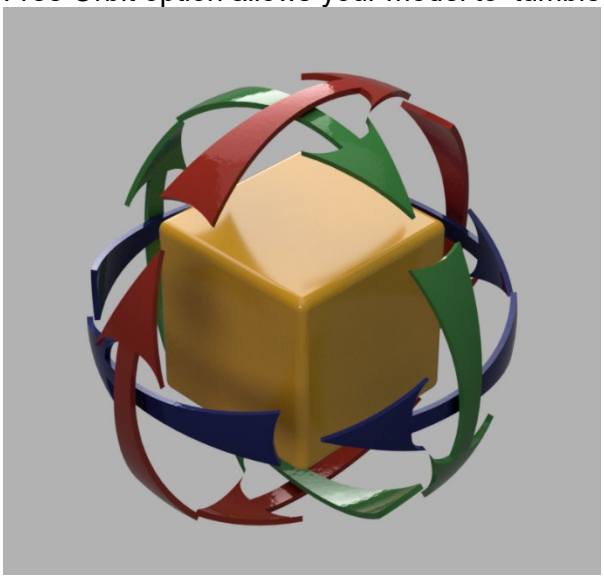
You can change the default button configuration for your mouse to suit CAD software you may have used in the past.



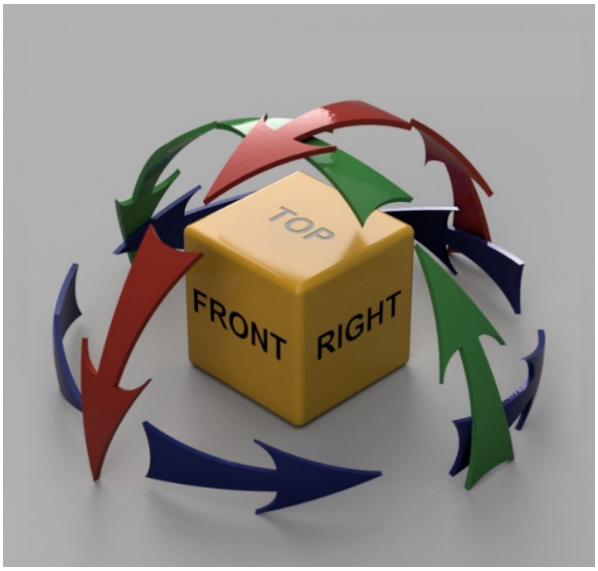


10. Free vs Constrained Orbit

The out-of-the-box setting for orbiting in Fusion 360 is Constrained Orbit. Which can make orbiting feel restrictive if you aren't on top of your View Cube settings. The constrained orbit is locked to orbiting around the axis between the Top & Bottom planes of the View Cube. So essentially you can imagine your model being on a turn table which can also orbit. Whereas the Free Orbit option allows your model to 'tumble' in any orientation.



FREE ORBIT

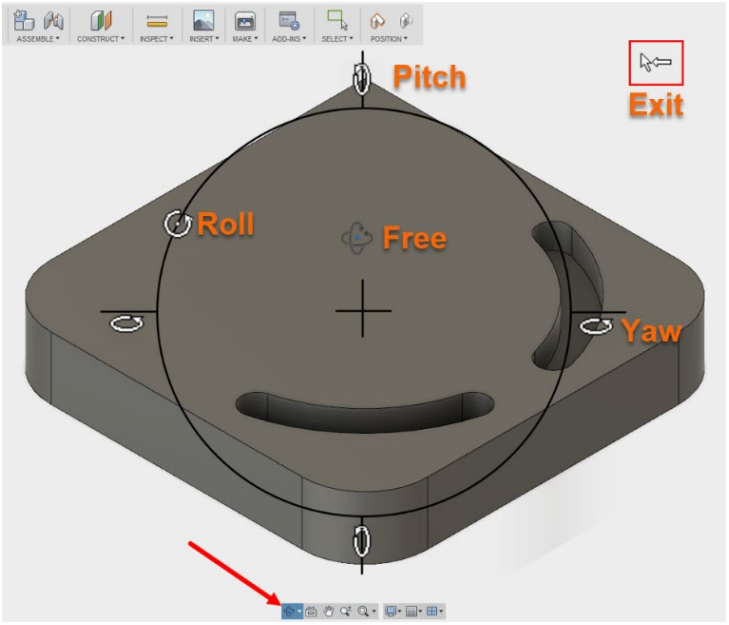


CONSTRAINED ORBIT

11. Orbit Wheel

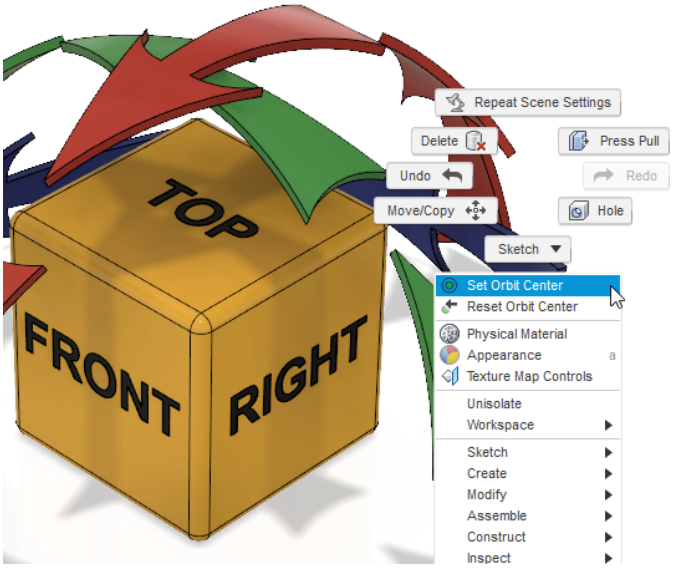
The Orbit Wheel is another common user interface element across a lot of Autodesk CAD products, and it has a few hidden functions. Each element on the wheel will display a different cursor icon giving you a clue as to its function.

You can also reset the orbit centre to the current position of the crosshairs by left clicking.



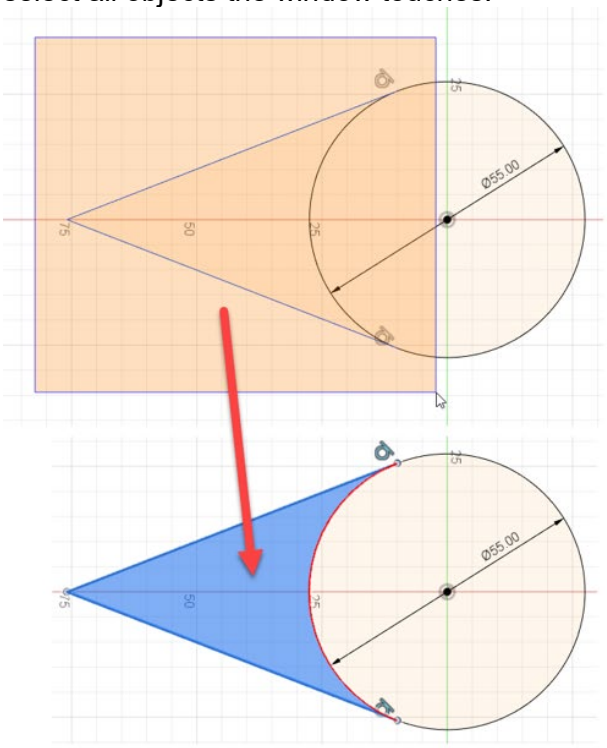
12. Set / Reset Orbit Center

When orbiting using the keyboard / mouse combo, as opposed to using the Orbit command. You can modify the orbit center using the context menu. Or you can do it on-the-fly by pointing to your desired orbit center, then while holding Shift, click with your middle mouse button. You can free orbit without using the command by holding down the Shift key on your keyboard and holding down your middle mouse button. Unless you have changed your navigation preferences. [This video] shows it in action.

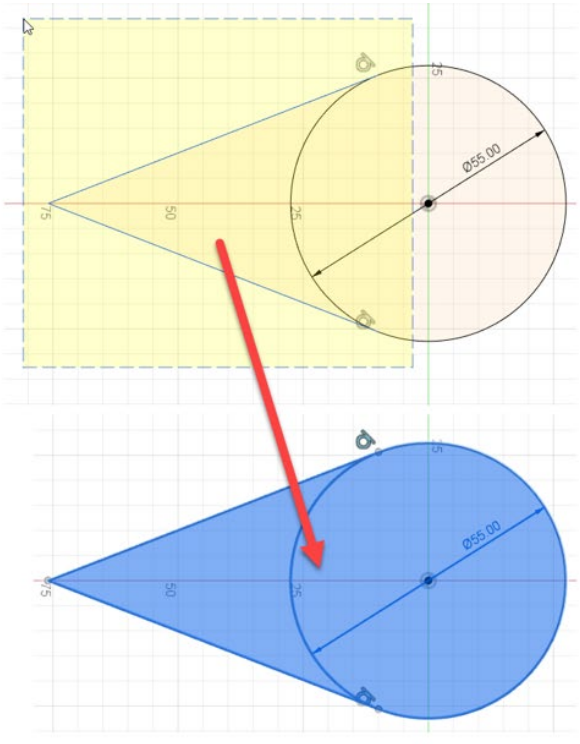


13. Window Selection

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.

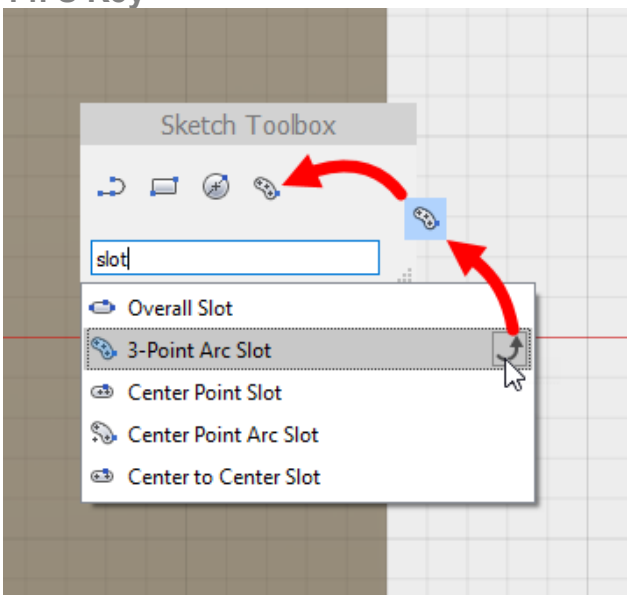


Top Left to Bottom Right / CW



Bottom Right to Top Left / CCW

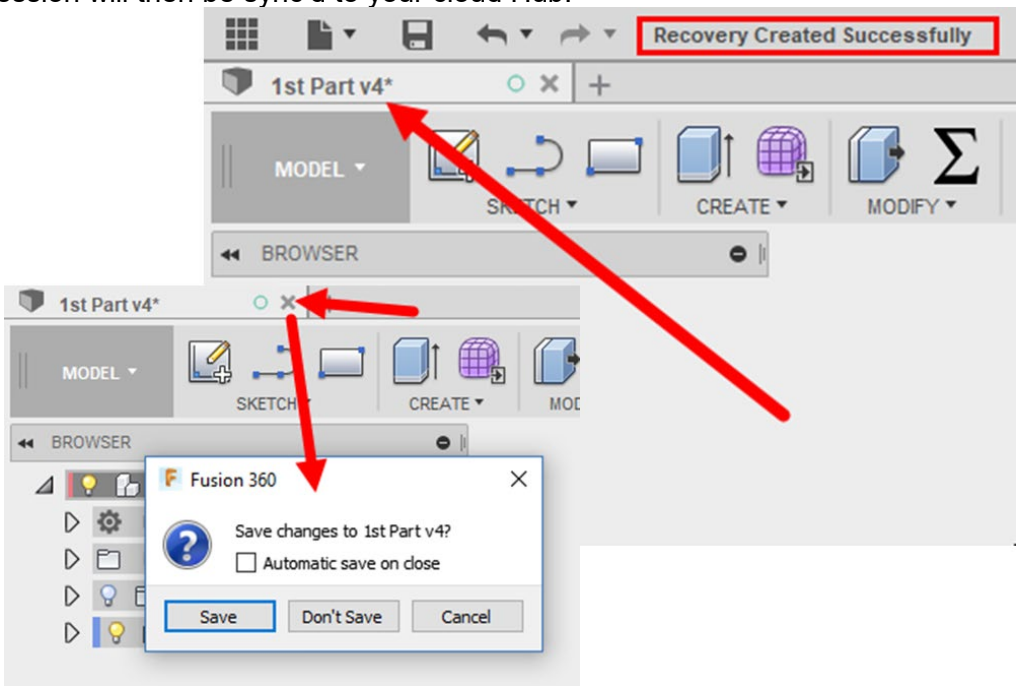
14. S Key



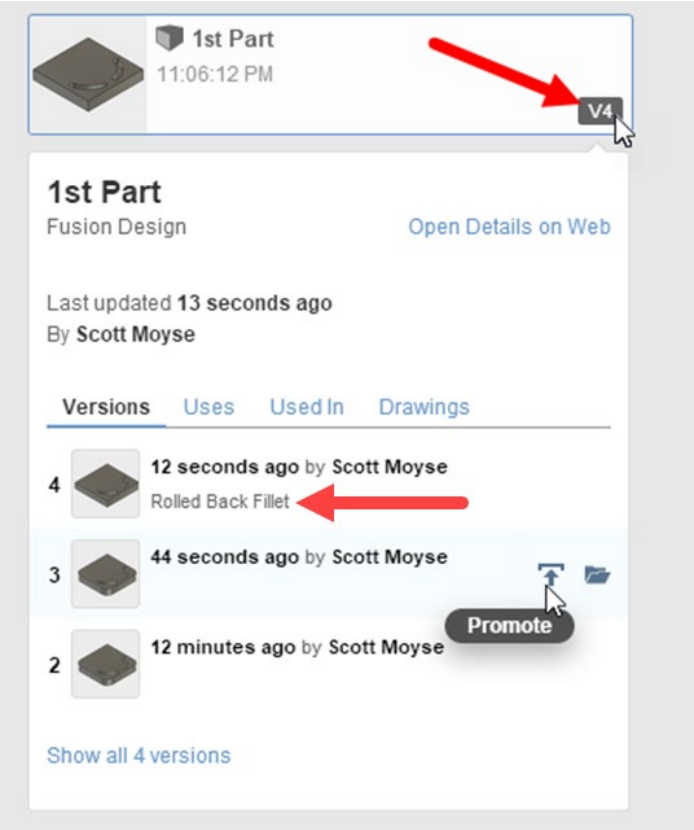
The S key is the single most useful command in Fusion 360. Press it, and all of its commands are at your fingertips. Not only is it a filtering search tool, but it's also a contextual customisable toolbar. Meaning you can pin the commands you use the most in the Sketch environment, and they only show 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.

15. Ctrl + Shift + S

Every time you save a file in Fusion 360, a new version is created and sync'd to your Hub on the Autodesk cloud servers. Not always ideal, and it quickly builds a long list of versions under your design in the Fusion Data Panel. However, if you use Ctrl + Shift + S instead, then it will just save a version locally as a recovery file. You can continue to save like this, until you close the Design, at which point you are prompted to save the Design. Your final version for that design session will then be sync'd to your cloud Hub.



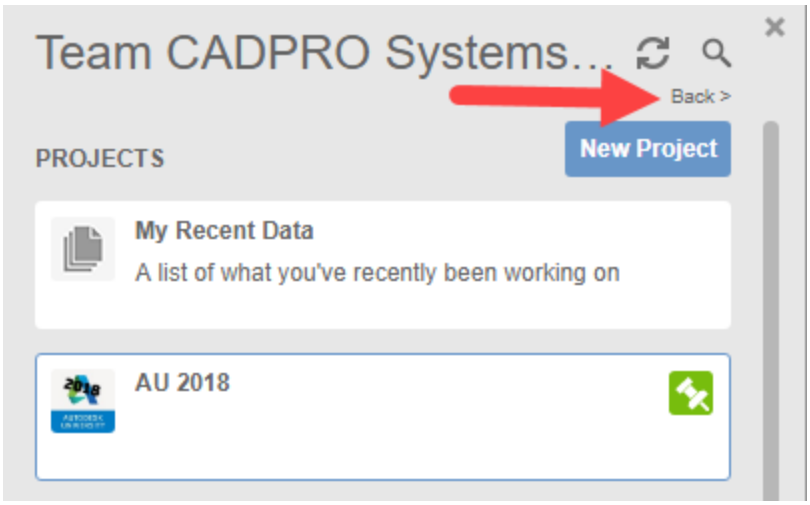
16. Versions



Each time you save your design, you have the opportunity to enter a comment about your recent modifications. If you enter a custom value, instead of leaving the default one, then it will show up next to the version in the Data Panel. To access all the versions of your design, you can click on the version number itself. In doing so you are presented with 4 tabs, Versions, Uses, Used In and Drawings. In the versions section you can see each version, the date it was created, who created it and a thumbnail. If needed you have the option to promote each version to the latest version or open the version.
The rest of the tabs give you the ability to assess impact of change on a design, as well as view a list of all the drawings which use this design.

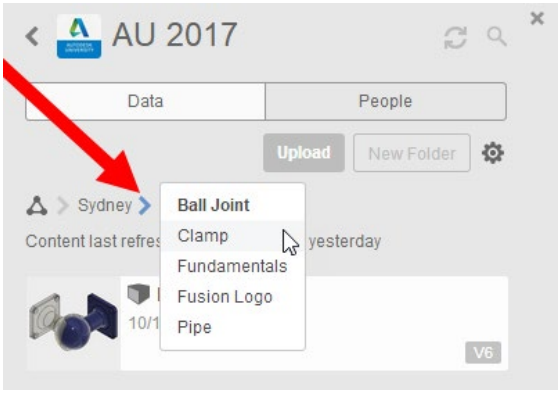
17. Go Back

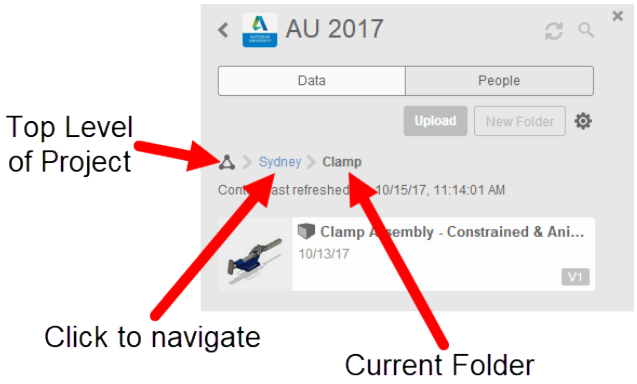
Everyone has done it… you choose the Data Panel home button, the chevron, instead of going back up a level in your project folder structure. So now you have to drill back down through the project and folder structure to get back to where you were. But there's no need… there is a stealthy little back button on the project home page you can use instead.



18. Data Panel Breadcrumb

Each element in the breadcrumb has a purpose and functions in a very similar way to the Windows Explorer breadcrumb in its address bar. Click the arrows to jump across folders via the resulting drop-down menu, and click the folder names directly to navigate straight to that location.





19. Offline Cache

If you travel a lot for work, have unreliable internet, or just want to make sure you have your core designs available at all times. You have a few options:

1. Automatic caching based on a duration setting in Preferences.
2. Project caching (right click on your project and choose Cache This Project)
3. Design caching (right click on your design and choose Add to Offline Cache)

The first option shouldn't be relied on, since it's susceptible to too many factors which may result in your not having a local cache of your data when you think you should. The latter two options are super helpful but are a one-time only gig. If a new version of the design is created and saved from another device, then your device won't automatically sync that change. You must manually cache again.

20. Use Fusion Team

When you use Fusion 360 for the first time, you are accessing it as a user, NOT a company. In the same way you access Dropbox as a private user. This is fine for single users, but if you are part of a company, or own a company with employees using Fusion 360, you need to get onto Fusion Team ASAP. Fusion Team is 'like' Dropbox for Business and allows companies to take complete ownership over the designs and projects their employees are creating inside Fusion 360. It's also a bit like using a network drive (Fusion Team), instead of everyone working on their own hard drives (Fusion 360 My Hub). On top of this massive benefit, you also gain extra features not available in the standard issue My Hub:

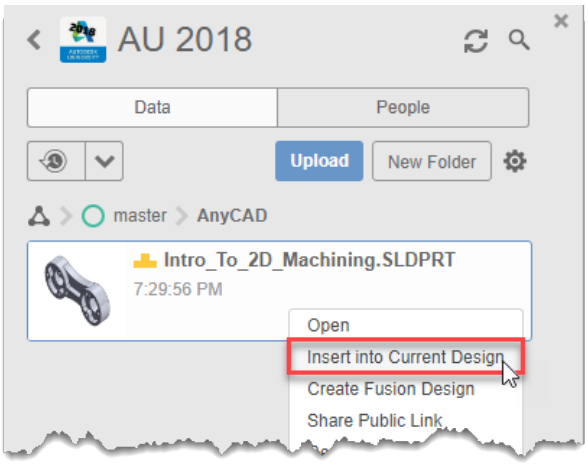
- Share designs across projects
- AnyCAD for Fusion 360 via the Desktop Connector
- Desktop Connector
- Common CAM Asset folders for posts, tool libraries, machine configurations and templates.

Contact your local reseller to find out how to gain access to Fusion Team for your company.

21. AnyCAD

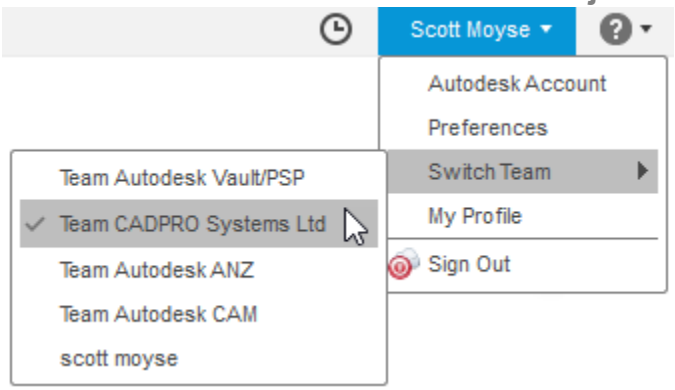
AnyCAD technology allows you to reference 3rd party CAD data as if it was native Fusion 360 data. So get this setup you need to:

1. You have to be signed up for Autodesk® Fusion Team. If you are not signed up yet, click here to learn how to do so.
2. You have to be a part of a Team Hub.
3. Desktop Connector has been properly installed. Click here to learn more.
4. Make sure that the AnyCAD preview checkbox is enabled in the Preferences dialog.



From there you can browse through your Team Hub projects and folder structures, saving 3rd party CAD data where you need to. Once uploaded, from the Fusion 360 Data Panel, you can right click on any of those files, and insert them into a saved Design file in the same way you can reference in other Fusion 360 designs. From there, you can work with the data as normal, and everything updates to match any future changes in the geometry of that file. Check out this example with soft jaws & CAM.

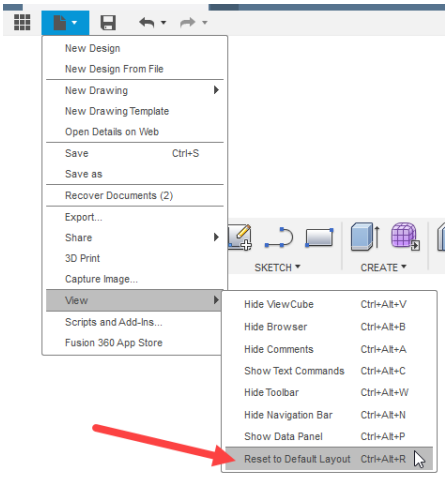
22. Reference files from a different Project



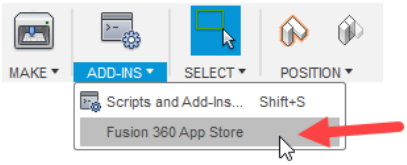
From the very beginning of Fusion 360 users have wanted to reference designs from one project into another. This isn't possible with the standard Hub, but it is when you are logged into a Fusion Team Hub. Just drag-and-drop or right click and select Insert into Current Design as you would normally, it's as simple as that. As a bonus Derive also works across Projects in a Fusion Team Hub as well.

23. Reset to Default Layout

So… you've lost all your toolbars, and the browser? The toolbar customisation reset tools aren't doing the job… where's the next port of call? The file menu; under the View submenu select Reset to Default Layout, and all will be right in the world again.

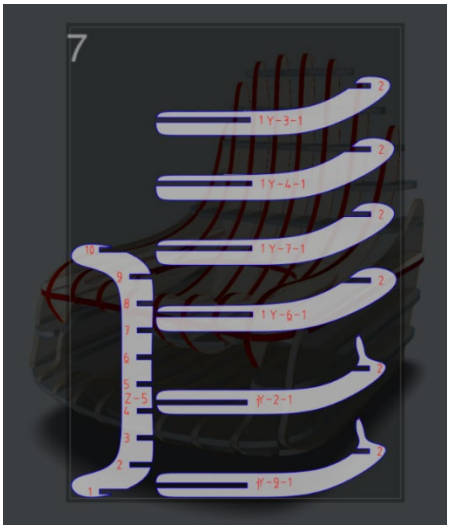


24. Fusion 360 App Store



There are loads of awesome apps to download and install on the Autodesk Fusion 360 App Store. Go check it out.

25. Slicer for Fusion 360



Slicer for Autodesk® Fusion 360™ slices and converts 3D models into 2D patterns that you can cut out of any flat material. It also creates 3D instructions you can interact with, to help assemble your model.

Create a model in Fusion 360 and with a few clicks you can send your model to Slicer for Fusion 360. Apply various slicing techniques to your model and create 2d plans in EPS, DXF or PDF formats that you can cut using Laser cutter or CNC machine.

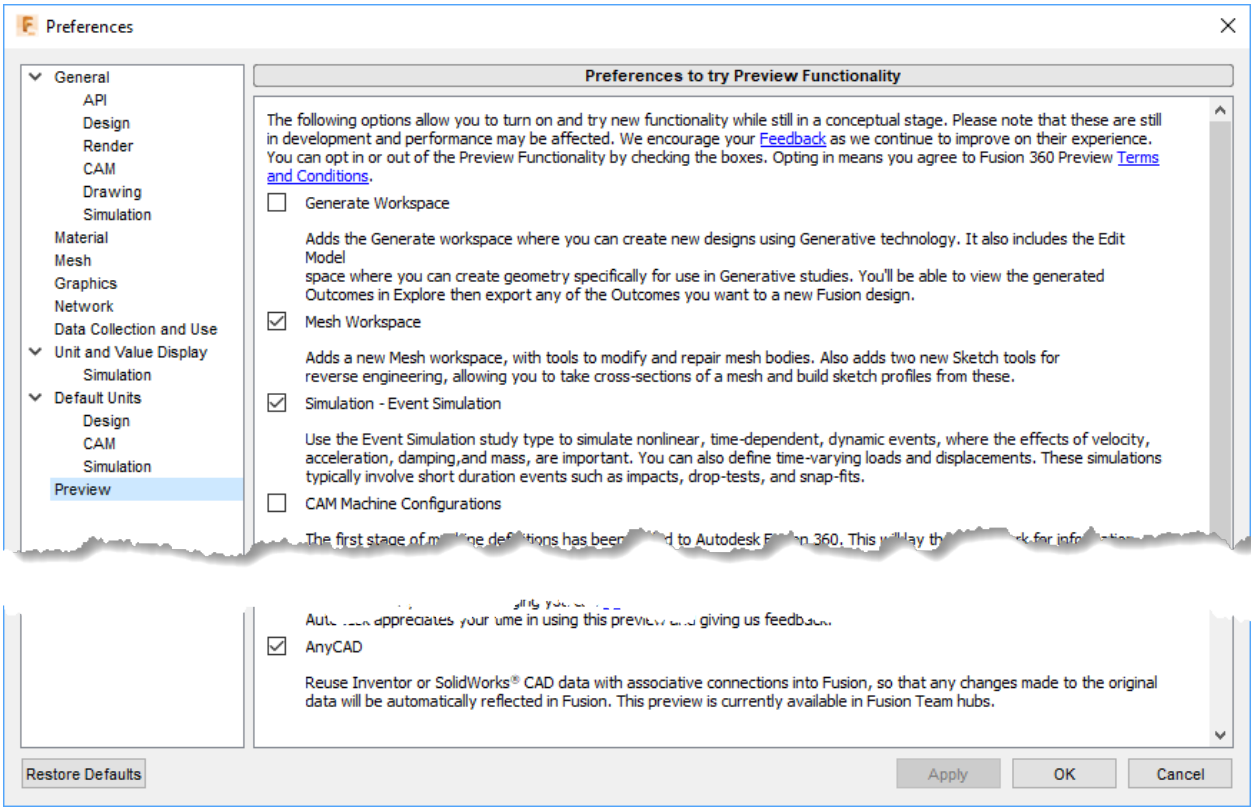
Slicer for Fusion 360 can be used standalone or as an add-in for Fusion, and lets you use different construction techniques to build your model based on 2D slices and animated assembly instructions.
Download it from here.

Fusion 360 Preferences

Access your preferences by clicking the drop-down next to your username in the top right corner of the Fusion 360 application window.

26. Preview Mode

This is where Autodesk put their shiny new goodies for you to try out and provide feedback on. Some of them are well worth using in your everyday workflows. Others are quite early in their development cycle for 'production' use, so they should be experimented with that in mind. Once you are in the Preferences dialog, select the Preview option from the browser on the left-hand side.



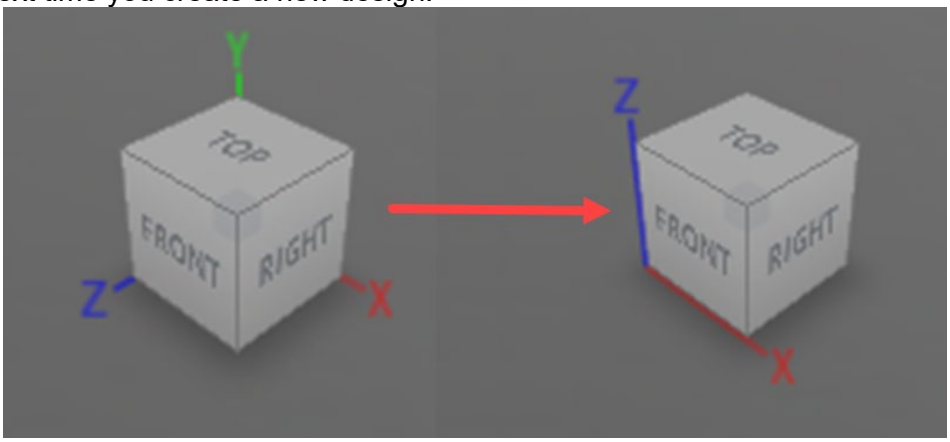
All of these except the Branch & Merge preview are worth having a look at. So go ahead, turn them on and accept the T's & C's. The Mesh & AnyCAD previews are on permanently in my configuration.

27. Set default material

Are you fed up with constantly changing the material of your components over to your most common material? Don't be, jump into your Preferences and on the Material tab you can configure your default material and appearance.

28. Z Up

By default Fusion 360, like most mechanical CAD modellers, has the WCS set as Y Up. For some users this can be counterintuitive. So Fusion allows users to change this in their preferences. Go to Preferences > General > Default modelling orientation. This setting will take effect the next time you create a new design.

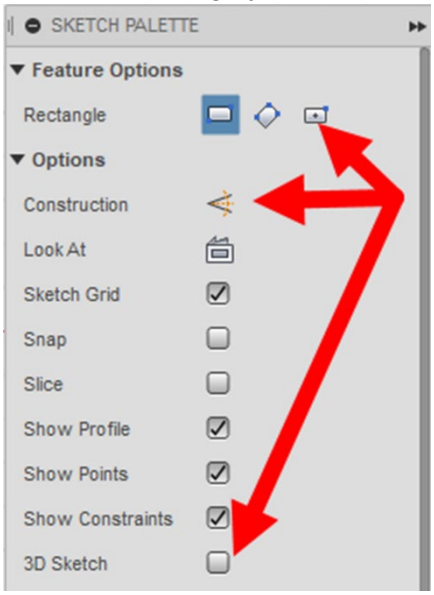


Sketching

Sketching is the foundation of any 3D model. So it's important to create stable and reliable sketches, especially when modelling with History enabled in Fusion 360. When using the history free, direct edit, mode in Fusion it's not quite as important. Nevertheless being able to sketching quickly and efficiently is a fundamental skill with 3D modelling. This section will help you in that endeavour.

29. Sketch Pallet Modifiers

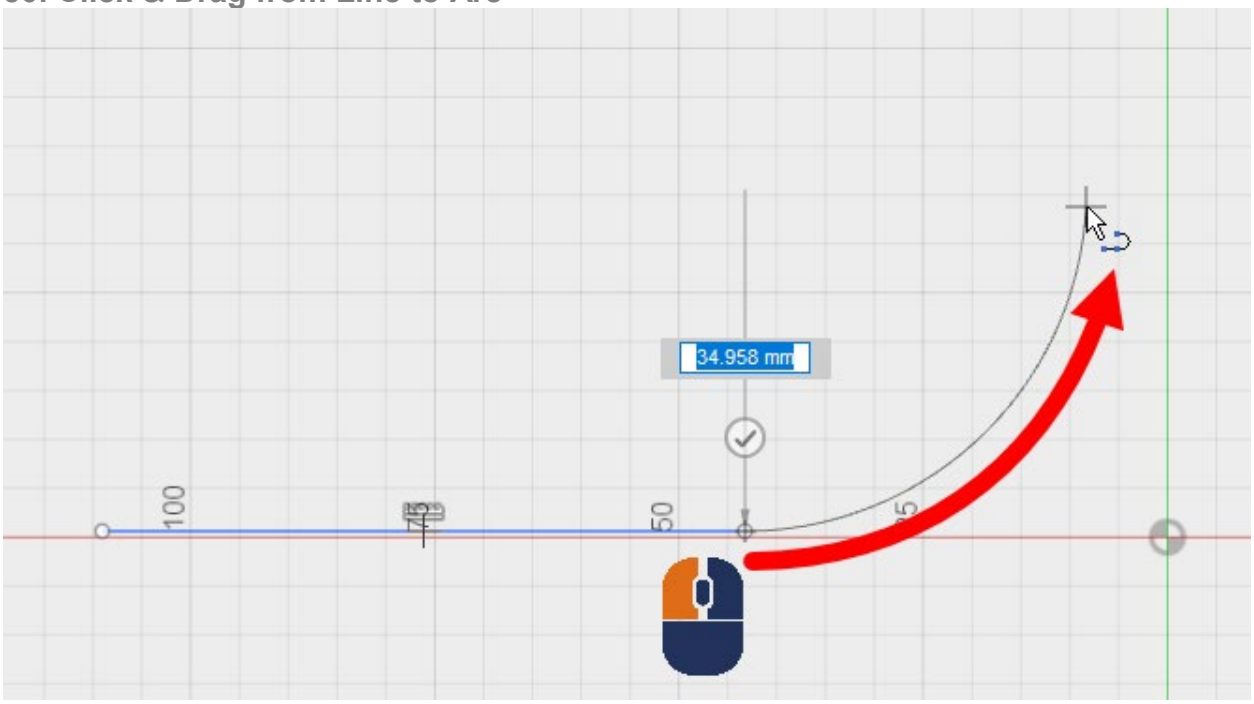
The sketch pallet in Fusion 360 is a hybrid of the old school pre-ribbon & new school context sensitive ribbon user interfaces. AutoCAD also uses pallet extensively. But this sketch pallet in Fusion 360 is highly context aware, and constantly adjusts the options available based on the active command or objects selected in the graphics area. The thing is, most of these things are easy to miss unless you catch it in the act of morphing to your needs, or you get told about it like you are now.



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.

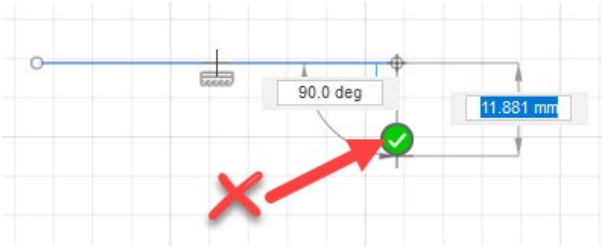
The Options section allows you to turn some very useful things on & off…. Is your sketch getting too busy? Then turn off all the Constraints. Setting sketch geometry as Construction is also something you should be making a habit of doing if you aren't already, since it makes your sketches more lightweight. 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.

30. Click & Drag from Line to Arc

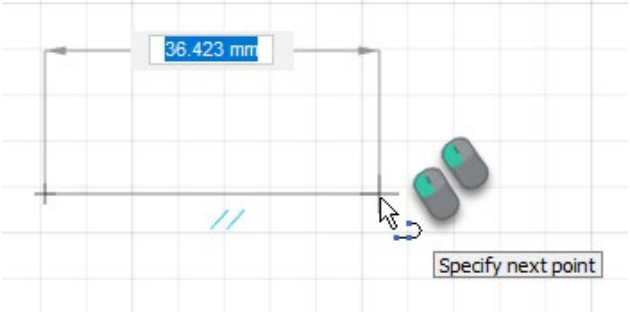


Fusion's Line command is actually a Polyline command, hence the icon. But to sketch out a polyline, you need to click twice to place the initial line, then click on the 2nd point of the line and drag away from it. In doing so you will see a tangent arc forming off the end of the line. Keep the left mouse 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.

31. 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.



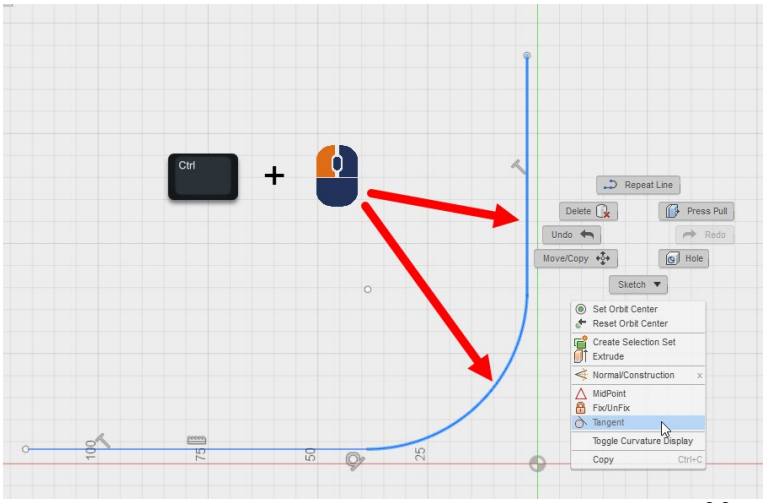
32. Infer Sketch Constraints

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, but 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.

33. 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.

34. 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.

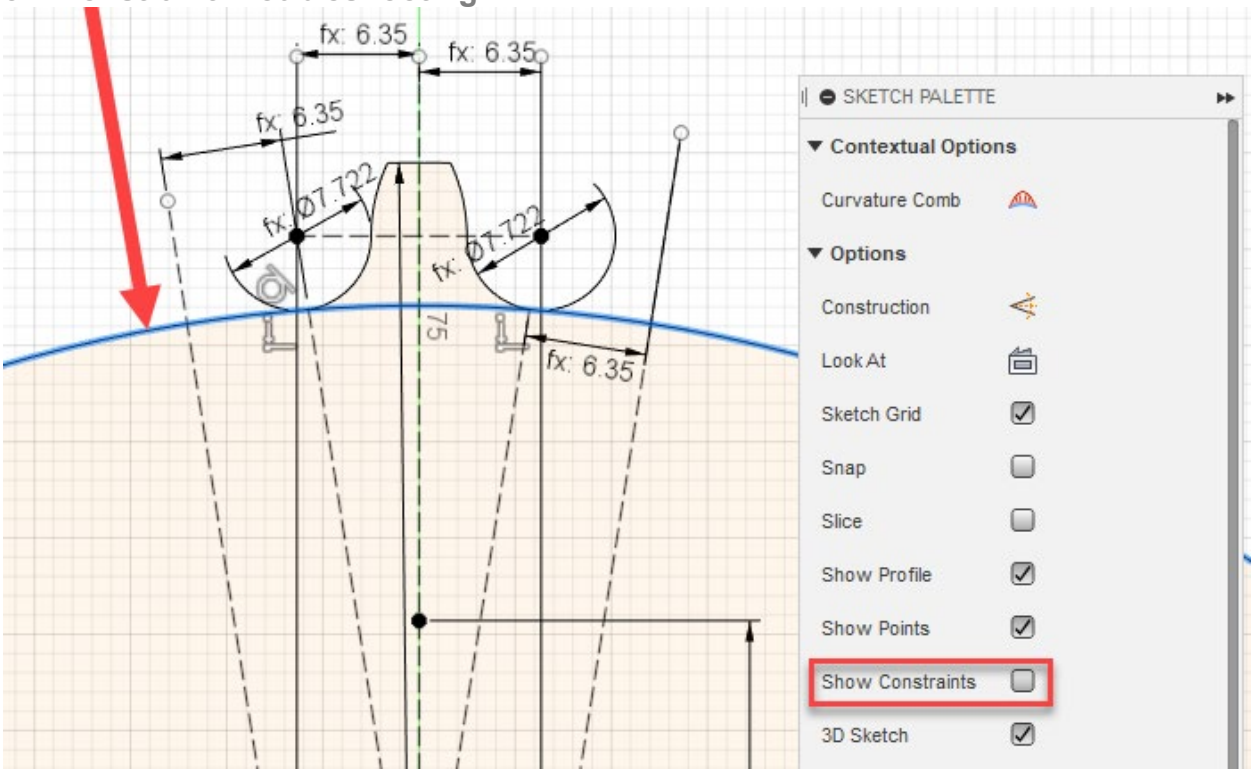
35. Click & Drag – Auto Perpendicular & Tangent

This one is hard to describe without just seeing it in action. But if you click and drag the first point of a line from a circle or an arc, it will lock the line tangentially. If you do the same thing but move away from the circle/arc it will create a radial condition. The same technique can be used to 'pull' perpendicular lines from other lines. Check out the video.

36. Midpoint Selection

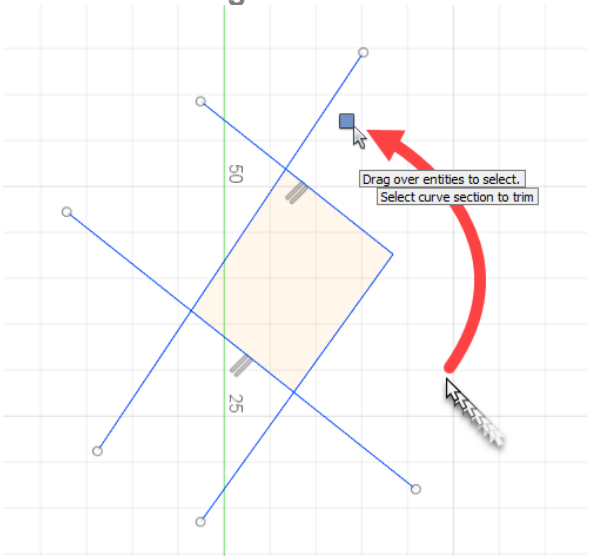
If you want to dimension to the midpoint of a line/arc or create a constraint to the midpoint of a line/arc. Then you just need to hold down the SHIFT key to activate the midpoint snap. Once your cursor reaches the midpoint of the line or arc, a triangle icon with a coincident constraint X will appear.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

37. Constraint Troubleshooting



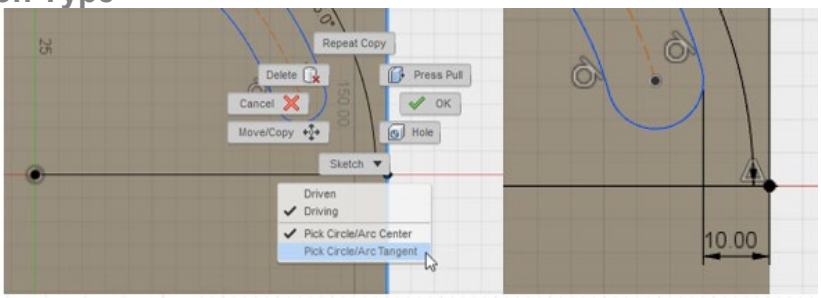
With complex sketches, the number of constraints showing on screen can be immensely distracting when trying to troubleshoot the parametric nature of a sketch. To overcome that, just turn off the Show Constraints checkbox on the Sketch Pallet. Then when you select any sketch objects in the graphics window, it will display only the constraints relevant for that object.

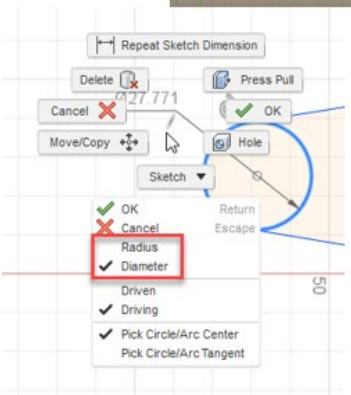
38. Click & Drag to Trim

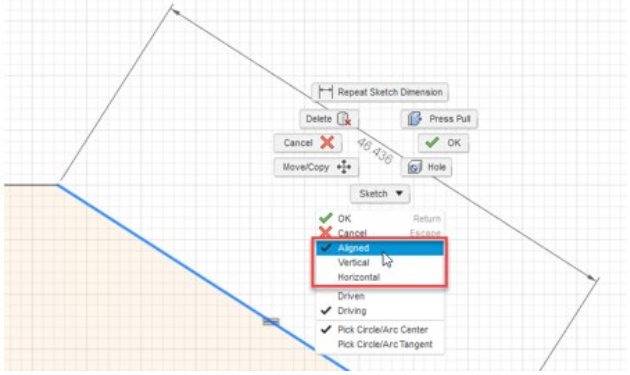


Once you activate the Trim command, you don't have to click each individual element. If there is a path to move your mouse through which will 'touch' all the elements you need to trim. Then just left click and drag your cursor along that path to remove all those elements.

39. Modify Dimension Type

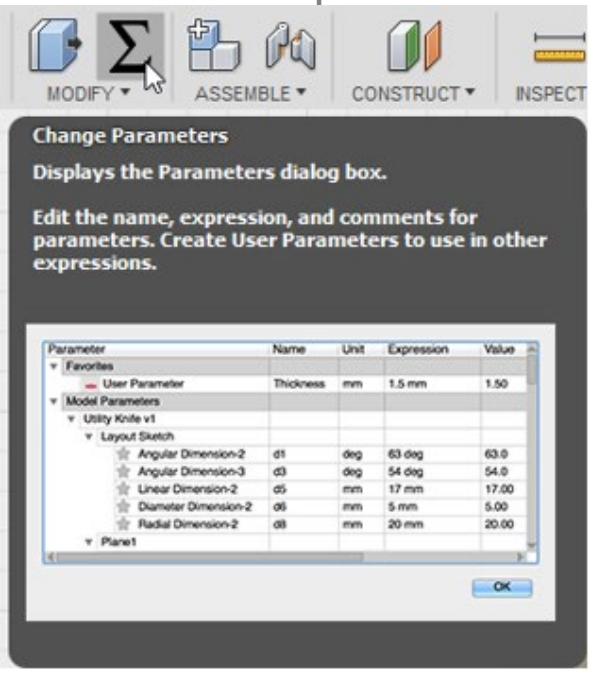


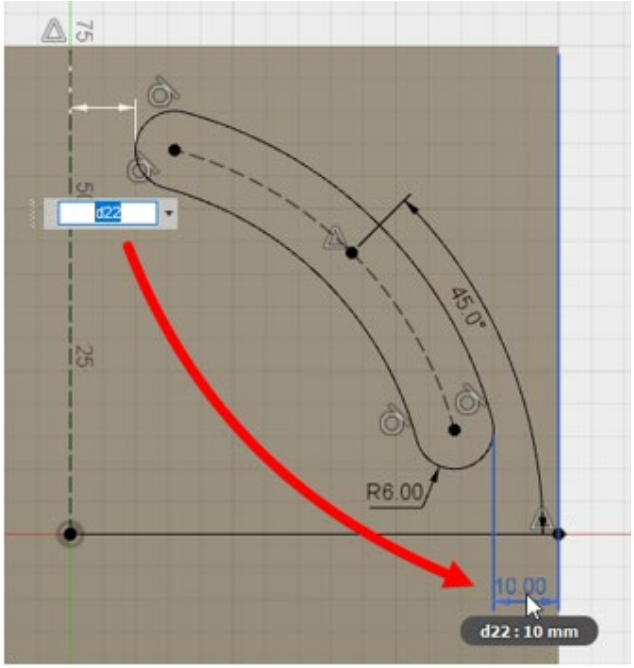




When dimensioning in a sketch, it's possible to change the format of your dimension via the context menu. Make sure the dimension command is active, then right click. With the radius/diameter toggle, you need to have selected the arc/circle already to see the options. And the same is true when dimensioning a line at an angle. When it comes to choosing between the center or tangent dimension, you can set it prior to the first selection or prior to the second one.

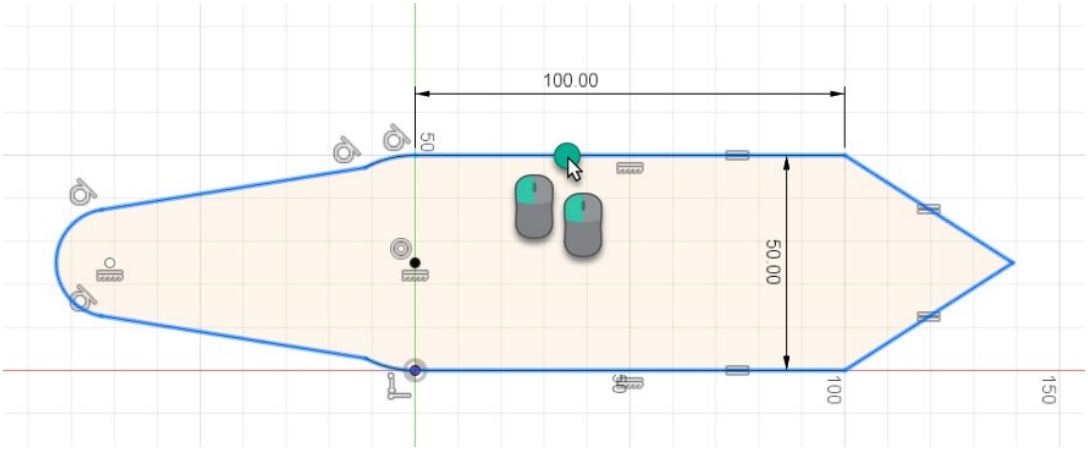
40. Build Relationships between Dimensions





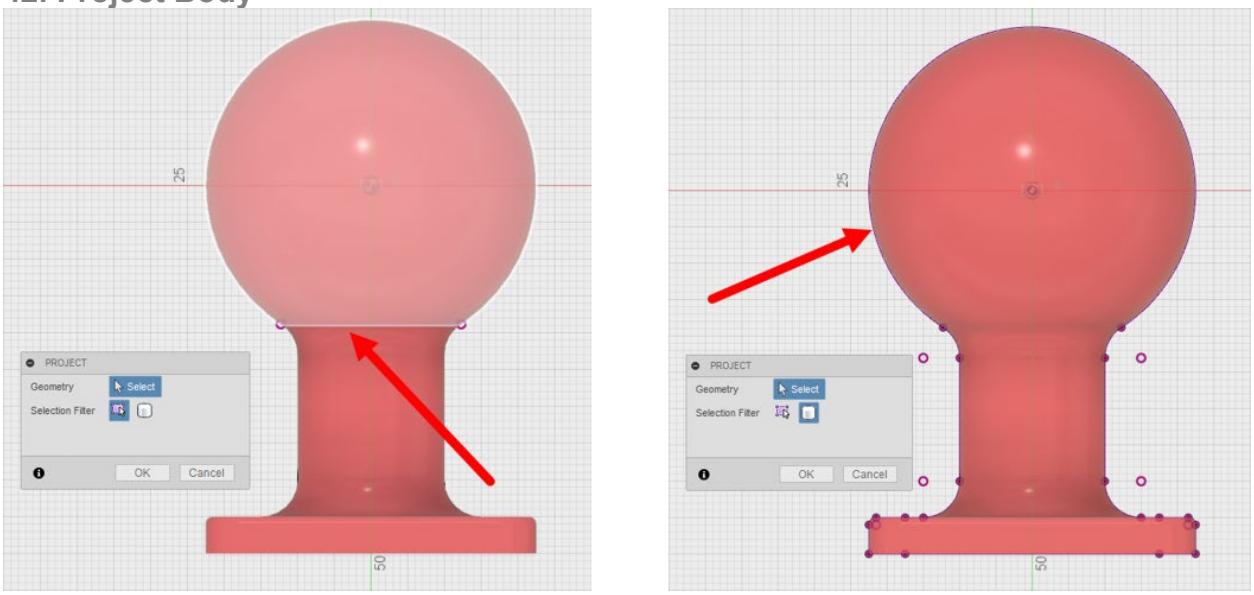
To build intelligent models in Fusion 360, you have to learn how to leverage parameters to their fullest extent. You can easily make one dimension/parameter equal to another, by selecting the dimension on screen that you want it to equal. This will instead the name of the parameter driving that dimension, 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.

41. Double Click to select Chain



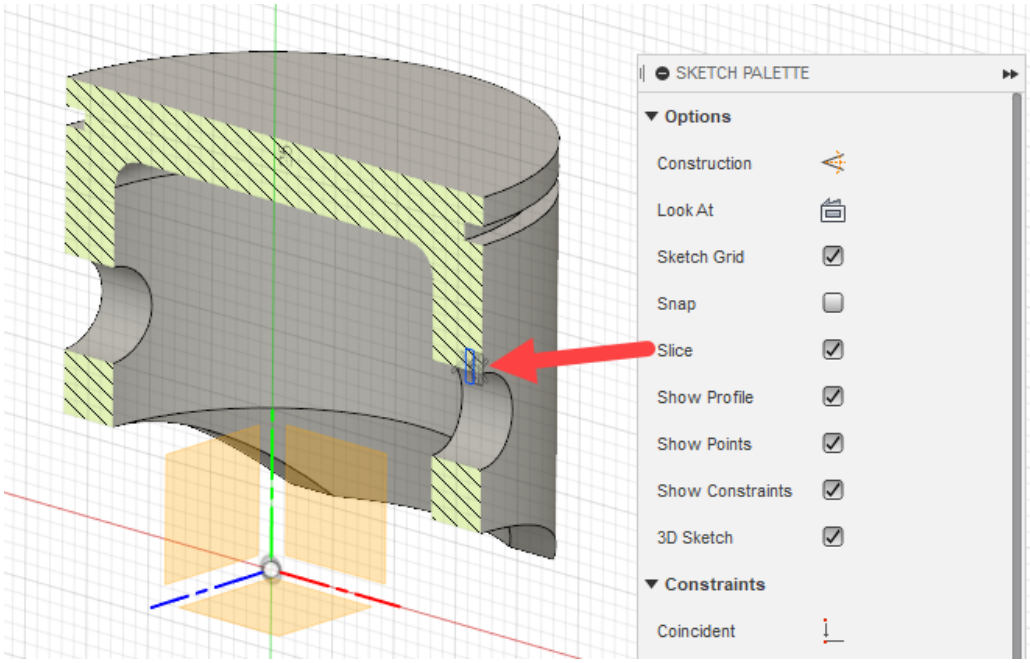
With no command active, double click a sketch element, and Fusion will select the entire edge chain around the profile.

42. Project Body



Press P for project, and you will notice on the Project dialog box that there are two filters. One for edges/faces and another for bodies. Selecting bodies will allow you to project the silhouette outline, instead of just edges. In the case of a sphere there are no edges to select which represent the overall size of the body. Project body takes care of this. It's a super useful command in many different scenarios.

43. Slice



Turning on Slice in the Sketch Pallet is really handy for when you need to create a sketch inside a component.

44. 3D Sketch

The **Allow 3d sketching of lines and splines** option must be activated under the Design tab in preferences for you to be able to create 3D sketches. From there you create the sketches a few different ways.

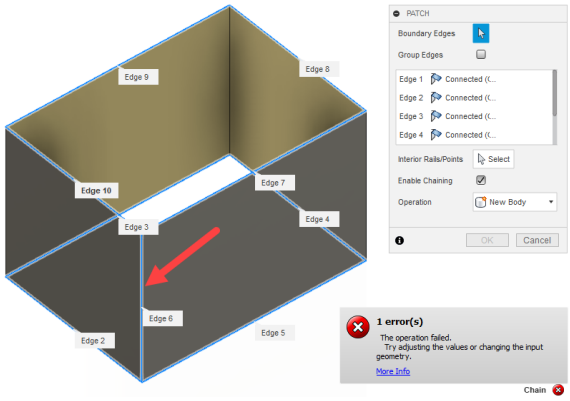
1. Use the Include 3D Geometry command
2. Use the Move command in a 2D Sketch to move sketch entities or point off the plane
3. Snap to objects in other sketches or model entities while creating lines & splines.

The last point is only possible if you enable the 3D Sketch checkbox on the Sketch Pallet.

Sketch Troubleshooting

You shouldn't have any issues with sketches you have created yourself. It's most likely imported geometry you will have to troubleshoot. Nevertheless, these techniques aren't fussy about the source of the sketch data, so give them all a try in your quest to clean up those sketches!

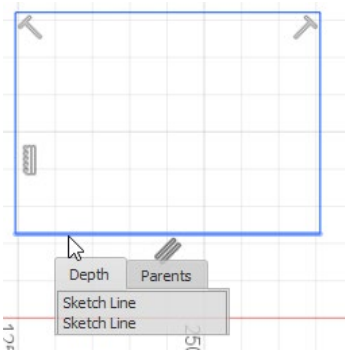
45. Surface diagnostics



If you can't extrude a solid in the Model Workspace because there's no profile to select, then the odds are you will be able to extrude a surface in the Patch Workspace. Once you have done this, you can use either the Stitch or the Patch tool to expose the point at which there is an issue in your sketch loop. In the case on the left you can see there is an additional edge which shouldn't exist for the Patch tool to try and fill.

At this point you can delete the extruded surface and edit the base sketch to take a closer look at that coincident relationship between those two objects.

46. White Points & Select Other

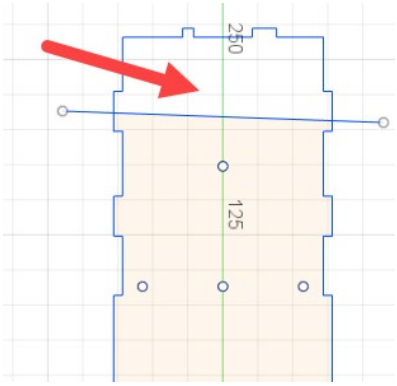


With the base sketch active, if you don't see any white points, then make sure the Show Points checkbox is active on the Sketch Pallet. This setting is sticky within each sketch, so it's possible for it to be ON in one sketch but OFF in another sketch. If you still don't see any white points, then it's possibly 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.

47. 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

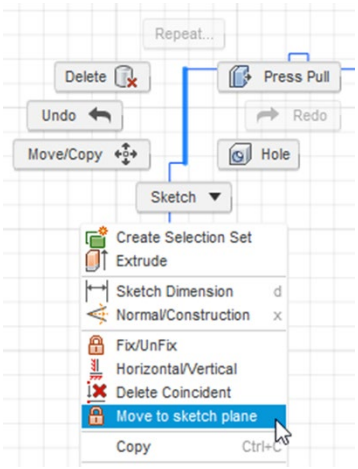
see black points is at the center of a fully constrained circle, or the control points on a fit point spline.



48. Divide & Conquer

A super useful technique, and my go to when troubleshooting sketches is the divide and conquer method. Just draw a line across where you should be seeing a profile and move it around until you see a profile display. This allows you to quickly zero in on the area(s) of the sketch loop which are problematic.

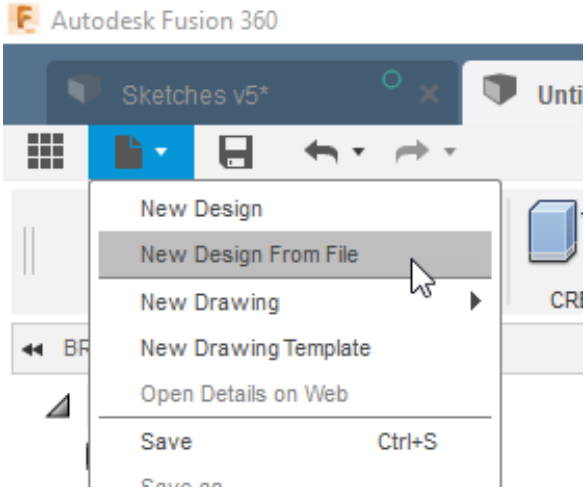
49. Move to Sketch Plane



Once you have narrowed down the search area, its highly likely that the techniques outlined in the previous tips will allow you to fix the issue… but another handy trick in your arsenal, is to right click on some of the problematic geometry and see if the Move to Sketch Plane option exists. If it does, this means that some of the geometry isn't on the same plane as the sketch. Essentially, it's a fragmented 3D sketch. Window select all the geometry in the area which won't generate a profile, right click and select Move to Sketch Plane to fix it all in one go.

Modelling

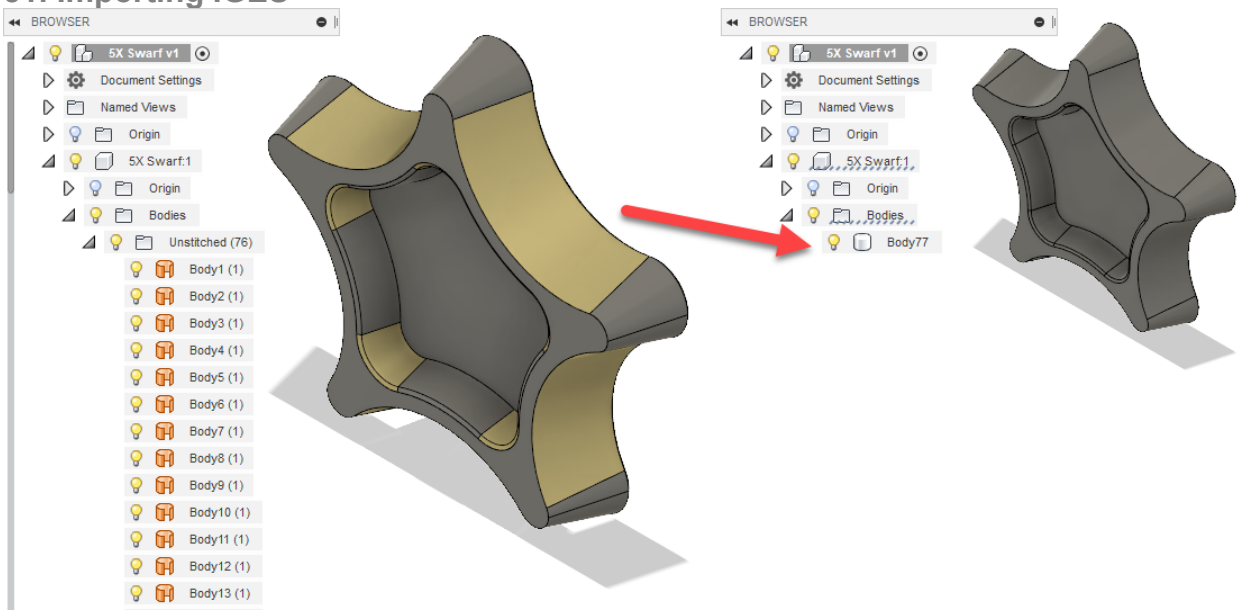
50. New Design from File



With some file types you don't have to upload the files via the data panel or Autodesk 360 website. From the File drop-down menu you can select New Design From File. In there you are able to import 4 common 3rd party file formats; IGES, SAT, SMT & STEP, as well as the native Fusion 360 design F3D file.

This import method can be completed without an internet connection and so uses local translators to get the job done. Once you save the file, Fusion will sync the new design file to your Autodesk Cloud account. Either MyHub or Fusion Team depending on your configuration.

51. Importing IGES



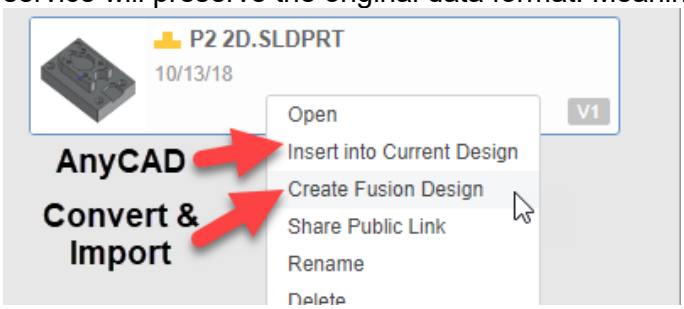
It's not obvious when you import an IGES file into Fusion 360, but by default it imports each face as an individual surface body. You will notice something isn't right if you start trying to perform extrude cuts or start programming toolpaths in the CAM workspace. If you expand out the Bodies folder (for the relevant component) you will see a full list of all the surface bodies. To fix this up, you need to switch to the Patch workspace, and from the Stitch command window select all of the surface bodies in the graphics area and stitch them all together. At this point the bodies folder should only show a single solid body.

52. Uploaded via Autodesk 360 / Fusion Team?

There are 3 ways to add 3rd party date to Fusion 360:

1. New Design from File
2. Upload via the Data Panel
3. Upload via Autodesk 360 (your Hub or Fusion Team Hub).

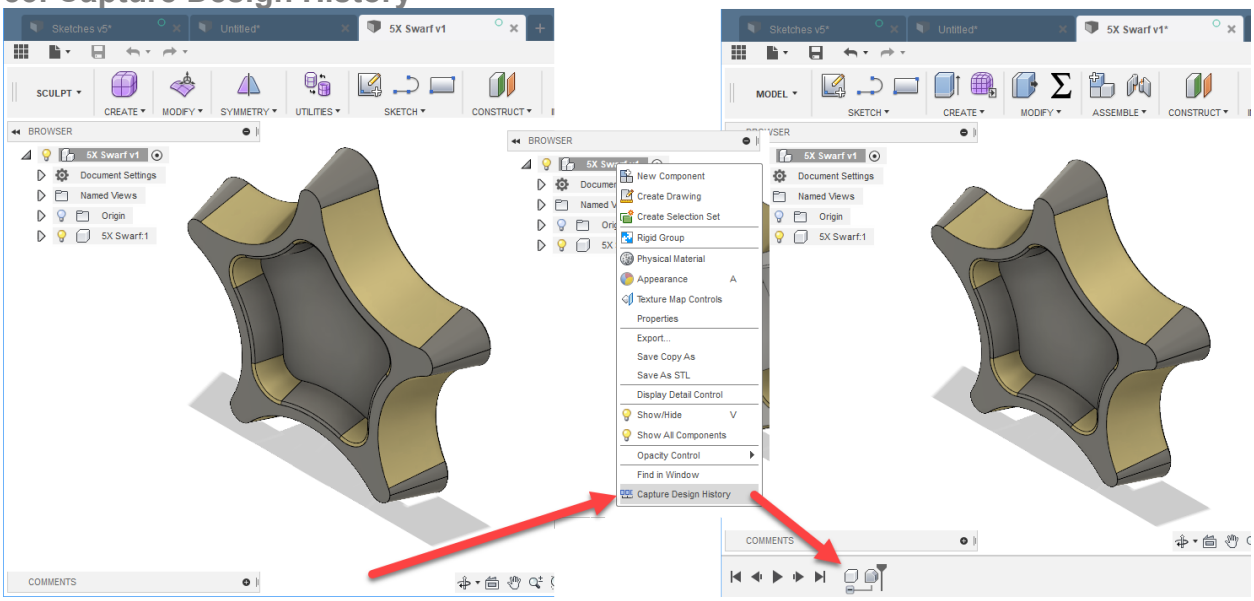
If you upload via the first 2 options, the 3rd party data is immediately translated into a Fusion 360 design. However, if you upload via the website or your Desktop Connector tool. Then the A360 service will preserve the original data format. Meaning you can use the service instead of Dropbox (A360 understands CAD file relationships), or take advantage of the AnyCAD technology I covered in Tip 21. If it translated it immediately this wouldn't be possible.



Consequently, in the Fusion data panel, if you right click on the file you have two options to open the file (assuming you have the AnyCAD preview enabled and you're in a Fusion Team Hub).

1. Insert into Current Design (AnyCAD)
2. Create Fusion Design (converts the data & saves a copy of the file)

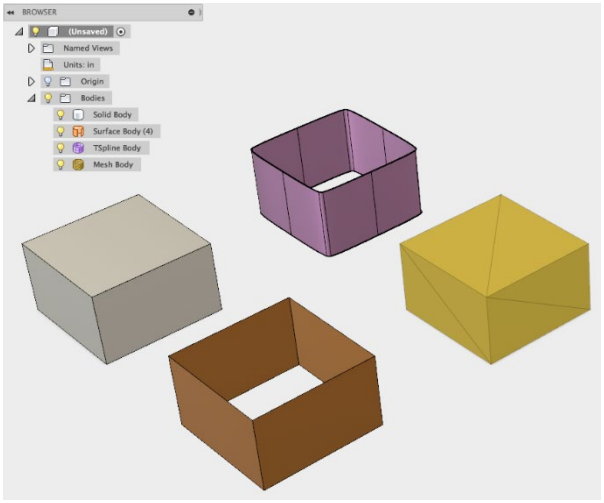
53. Capture Design History



Imported models don't capture design history by default. As such they are in Direct Edit / History free mode, so there is no Timeline shown at the bottom of the Fusion window. To enable history, right click on the top level node in the browser (normally the same name as your design), then select Capture Design History. You will now see a Timeline at the bottom of the Fusion window. Bear in mind, you can turn off history from the same context menu, however, it will permanently delete any build up history or features on your timeline.

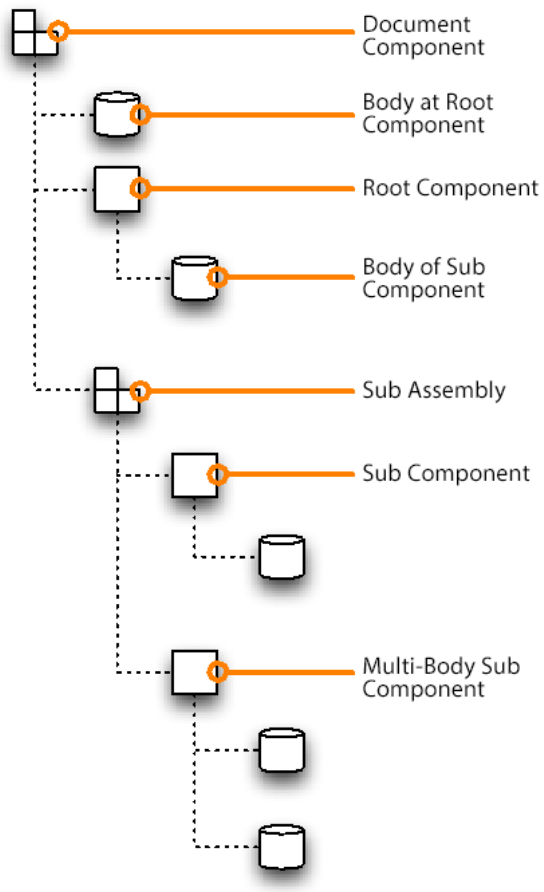
54. Bodies

- Have a type
  - Solid
  - Surface
  - TSpline
  - Mesh
- Made from features
- Can be organized in folders
- Do not move with respect to their parent origin
- Can have materials (but probably shouldn't)



55. Components

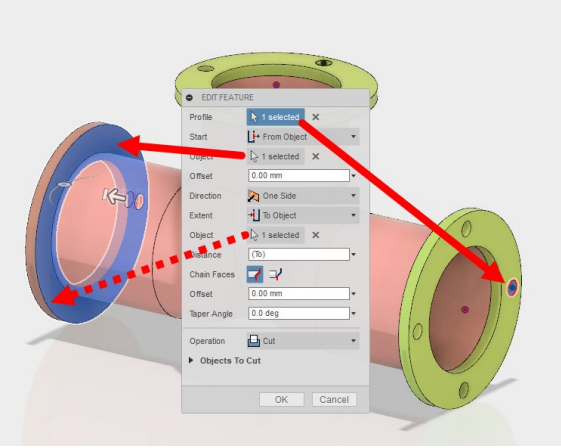
- Any Design is a Component and can be a part, or an assembly
- Fusion 360 calls both parts and assemblies “Components”
    - Any component can have child components
    - Any component can have 0, 1 or more bodies
    - Components can have both bodies and child components
- General rule: always create a component as soon as you know you need one.
- Components can move relative to their parent origin
- Activating components helps to manage your timeline
    - Components ‘own’ features & resulting bodies
- Joints only work with components
- Drawings can be (easily)made of individual components
- Have part numbers and other properties
- A BOM is based off components and their properties
- Can have multiple occurrences
- Can be saved out to a new design
- Can be inserted
- Each component has its own origin



56. Extrude From

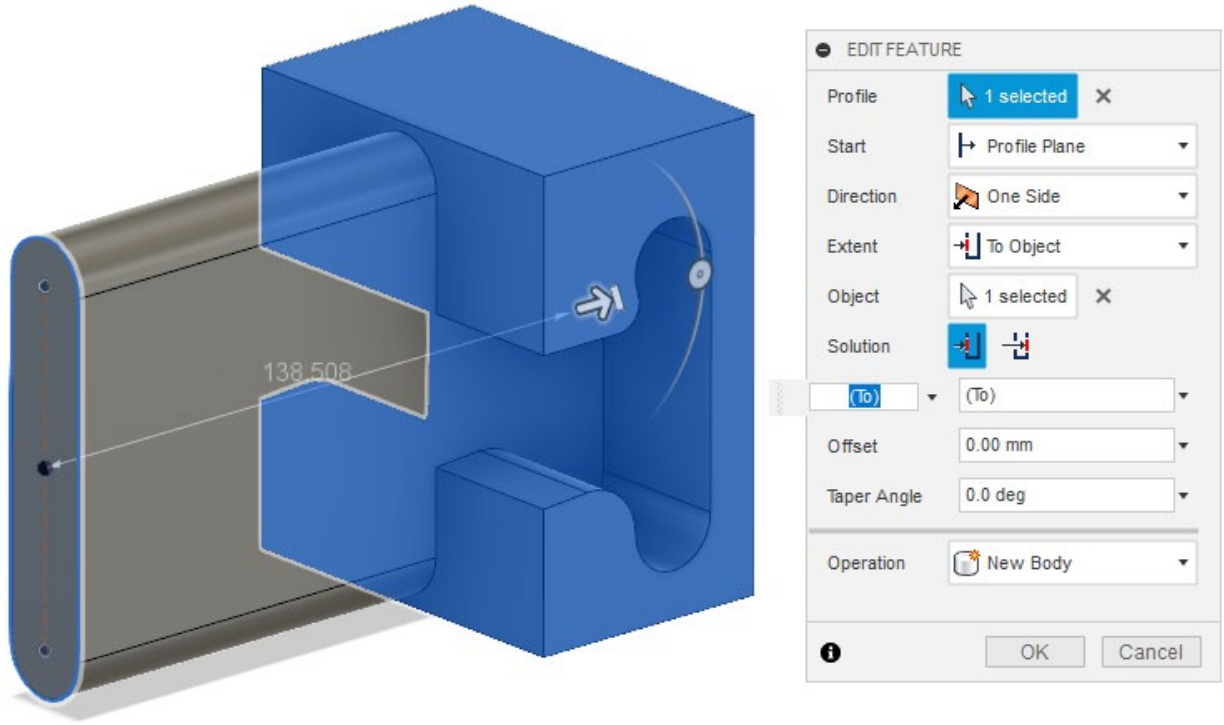
This pretty handy feature allows you to start an extrusion from a different plane to where the sketch is. In a similar vein you can select an object to extrude to, allowing you to ‘project’ a profile to create an extrusion between two difference references nowhere near the sketch profile itself.

- Select Profile on another plane
- Start - From Object
    - Grab arrow glyph and drag to start
- Extent – To Object



57. Extrude to Body

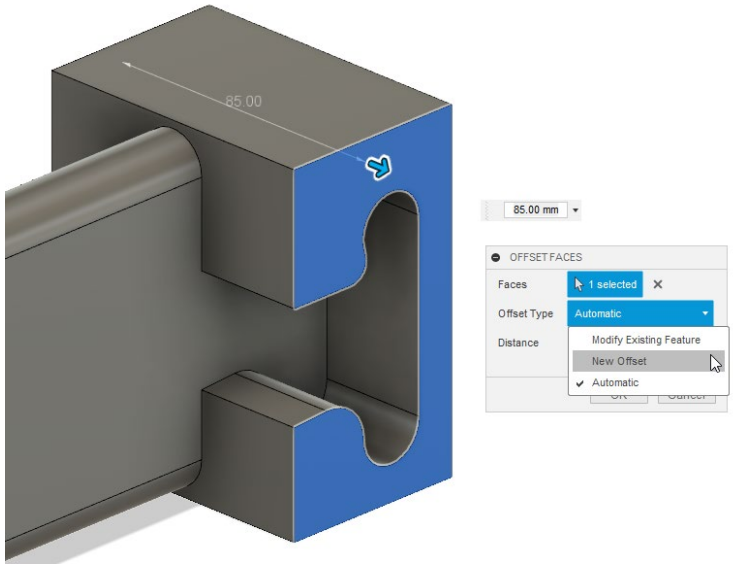
You can choose a body as the extent of your extrusion. The profile will terminate following the topology of all intersecting faces on the target body. Even undercuts are included.



When select the target body as the object, make sure you select the edge of the body to successfully select it.

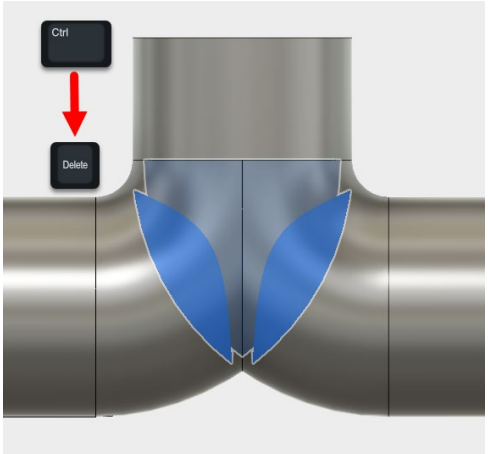
58. Offset Face

To execute the Offset Face command, you need to press Q for the Press Pull command. It will then adapt to the selection you make. So, if you select a face, it will morph into the Offset Face command. Once active, there are essentially two modes, Modify Existing Feature & New Offset.



However, the default is Automatic. If you select a face which is part of an existing feature, then the command will adjust the base dimension driving that feature, otherwise it will apply a new offset. Naturally, there are times when you just want to add say 2mm to an existing extrusion, this is when you would override the command to use New Offset instead.

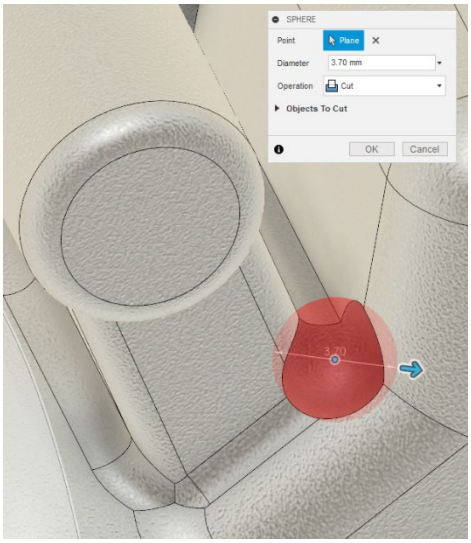
59. Delete Faces



If you select a face in Fusion 360 and press the Delete key on your keyboard, it will delete the face and heal it again thereby maintaining a solid. It's an incredibly powerful feature in a lot of situations, which can save a lot of additional modelling. Or in the case of imported models, it can help greatly with troubleshooting poor geometry or defeaturing for the sake of representation only. Sometimes for it to work effectively you have to select a group of logical faces, so make sure you use the Ctrl key to select all that you need for the operation to complete successfully.

60. Destructive Repair

Sometimes you just have to burn things with fire to start again. In CAD terms, a big fat boolean subtract is just the ticket. Dodgy fillets on imported models are often troublesome when trying to rework them. A really neat workflow is to place a workplane on one of the edges, then use a Sphere Primitive from the Create drop down on the Ribbon. Set it to cut away the model and expand its size just enough to remove all the troublesome faces. Then you can use the previous tip, Delete Faces, to either delete adjacent faces you couldn't before, or delete the resulting concave spherical face itself. Fusion will then heal the model if it can.



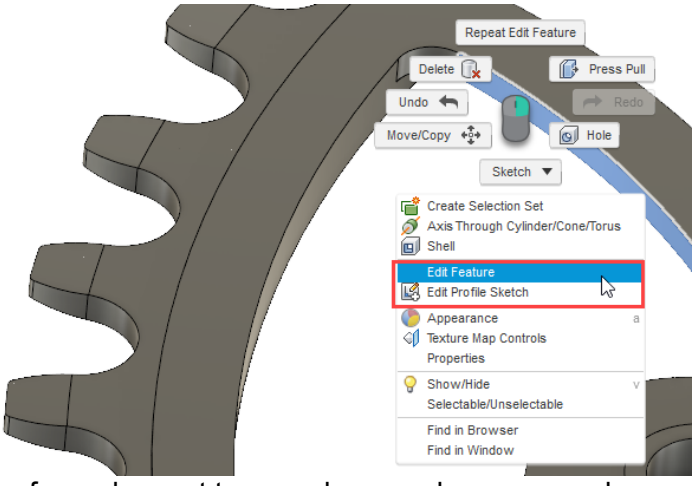
61. Name Objects

Parameters

Parameter	Name	Unit	Expression	Value	Comments
Favorites					
User Parameters					
User Parameter	RollerDia	in	0.304 in	0.304	
User Parameter	Pitch	in	(1 / 2) * 1 in	0.50	
User Parameter	RollerWidth	in	0.090 in	0.09	
User Parameter	MountHoleDia	mm	6 mm	6.00	
User Parameter	Teeth		46	46	
User Parameter	PitchDia	mm	Pitch / sin((180 / Teeth) * 1 deg)	186.101	
User Parameter	OutsideDia	mm	Pitch * (0.6 + (1 / tan((180 / Teeth) * 1 deg)))	193.287	
User Parameter	SprocketThick	mm	0.93 * RollerWidth - 0.006 in	1.974	
User Parameter	MountHoleQty		5	5	
User Parameter	MountPCD	mm	PitchDia * 0.75	139.576	
User Parameter	BrandTabFillet	mm	3 mm	3.00	
Model Parameters					
Linear Dimension-2	d111	mm	2.00 mm	2.00	
Chain Ring Parameters exam...					
Tooth Profile					
Base					
Base Tooth					
Tooth Fillet					
Tooth Pattern					
countU	d25		Teeth	46	
TotalAngle	d26	deg	360.0 deg	360.0	
Hole Boss Sketch					

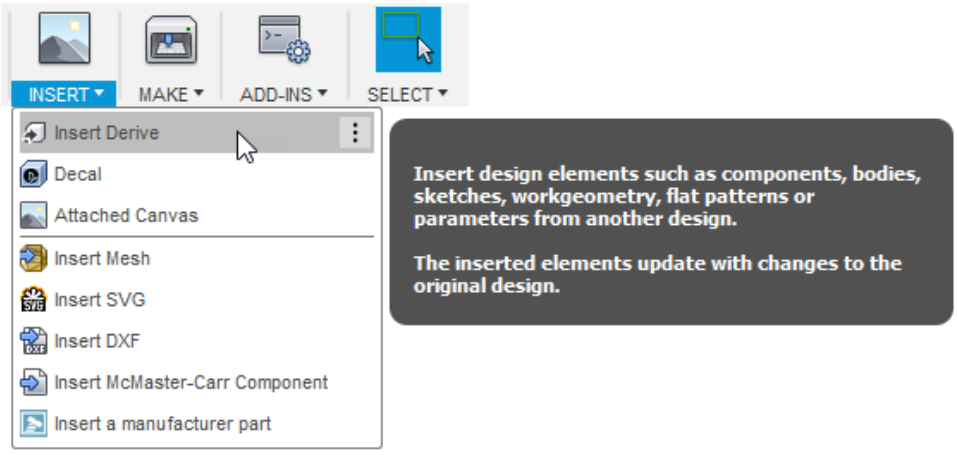
If you rename your sketches, construction features and modelled features working with your design in the future becomes a lot easier. Expanding out the sketch folder reveals a series of logically named nodes, allowing you to immediately edit the sketch you need, instead of bouncing in and out of hidden sketches to find what you need. However, the main point of this tip is to show you that by naming all these things, navigating the Parameters dialog becomes an absolute breeze.

62. Edit from canvas



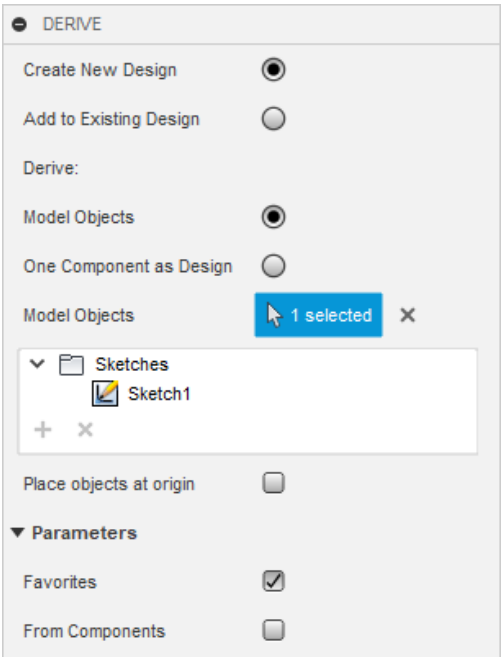
Fusion's Timeline is far from pleasant to use when you have a complex model with a lot of features. So even if you are a total legend when it comes to naming your objects, you're going to need some extra help to find what you need. If you right click on a face which belongs to a feature, you will find an Edit Feature option in the context menu, if that feature is a child of a sketch, then Edit Profile Sketch will also be present. Selecting either of these will get you to where you need to be without hunting around on the Timeline. The Select Other popup appears if there is more than one possible Feature to Edit, just select the one you want to edit.

63. Derive all the Things

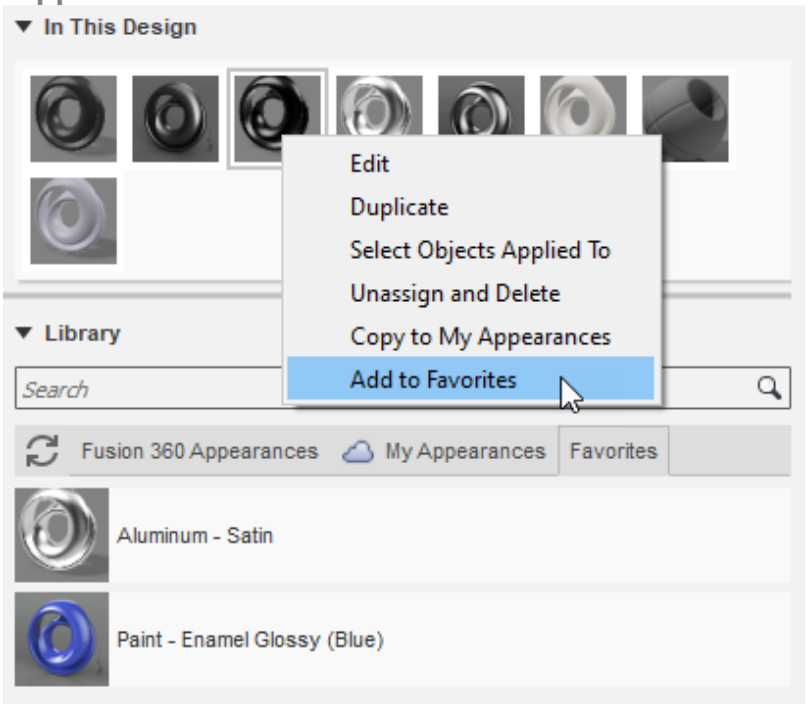


There are two Derive commands in Fusion 360. Create Derive and Insert Derive. This tool unlocks a massive amount of power for Fusion which previously wasn't possible. Two great examples are:

1. Skeletal modelling – you can create a base Design with layout sketches and common parameters. Then you can derive that into a bunch of other designs, either individual parts or subassemblies. After that you can build the top-level assembly by inserting all the derived Designs into it. Gone are the days of humungous Timelines in Fusion.
2. For CAM users, previously you had to include all construction geometry, fixture models, and toolpaths for each component inside one Design. Chaos quickly ensues. And if you work in a company where you have Engineers designing and Machinists programming, then Engineers tend to be protective over their work and don't like Machinists messing with it. Plus, it completely stuffs up your BOM & potentially also the drawings. Now the CAM programmers can Derive in just the component they need to 'CAM up' and build everything they need to achieve the result.

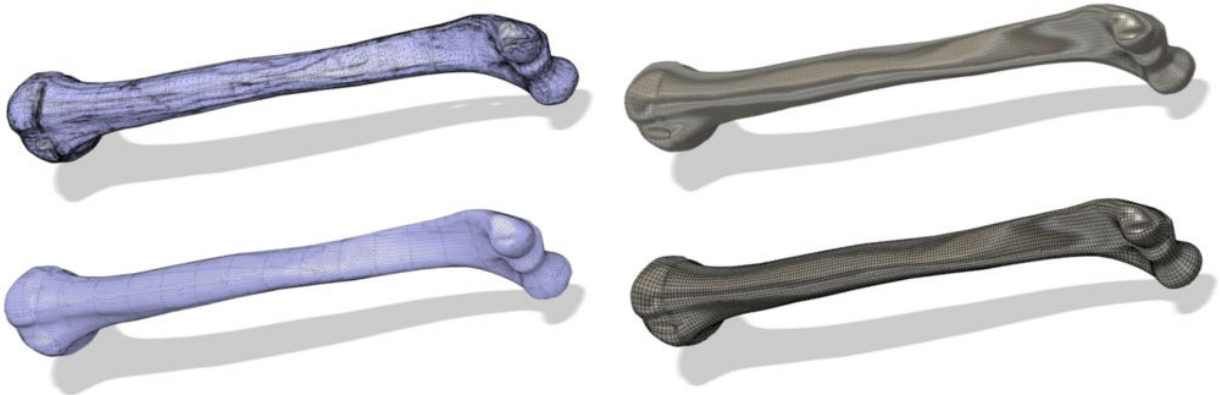


64. Materials & Appearances Favourites



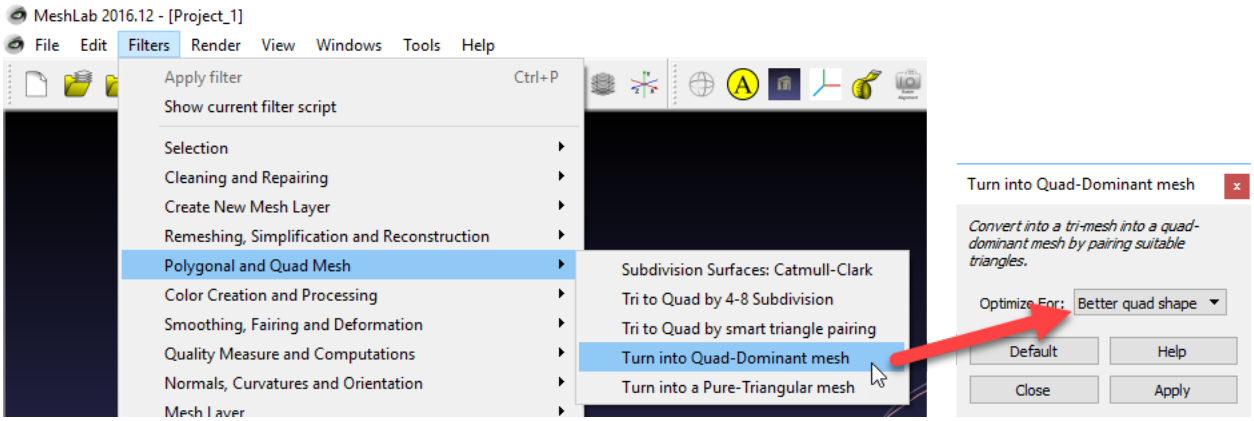
Instead of rummaging through the extensive list of materials and appearances all the time, you can create a list of favourites which you can access from any Design document. Just right click on the appearance and choose Add to Favourites.

65. STL to Quad to Sculpt



There isn't a native way of achieving this solely in Fusion 360 at this stage. The piece of the puzzle that's missing is a Tris to Quads conversion tool. A quads model is what's needed for a successful conversion to a T-Spline body. So, the current workflow is to either import the source STL into Meshmixer or Fusion 360 (with the Mesh workspace preview turned on), then perform any repairs needed and remesh to a maximum of 10,000 faces. At this stage you need to use a tool to convert the triangles into quads. I know of 3 options:

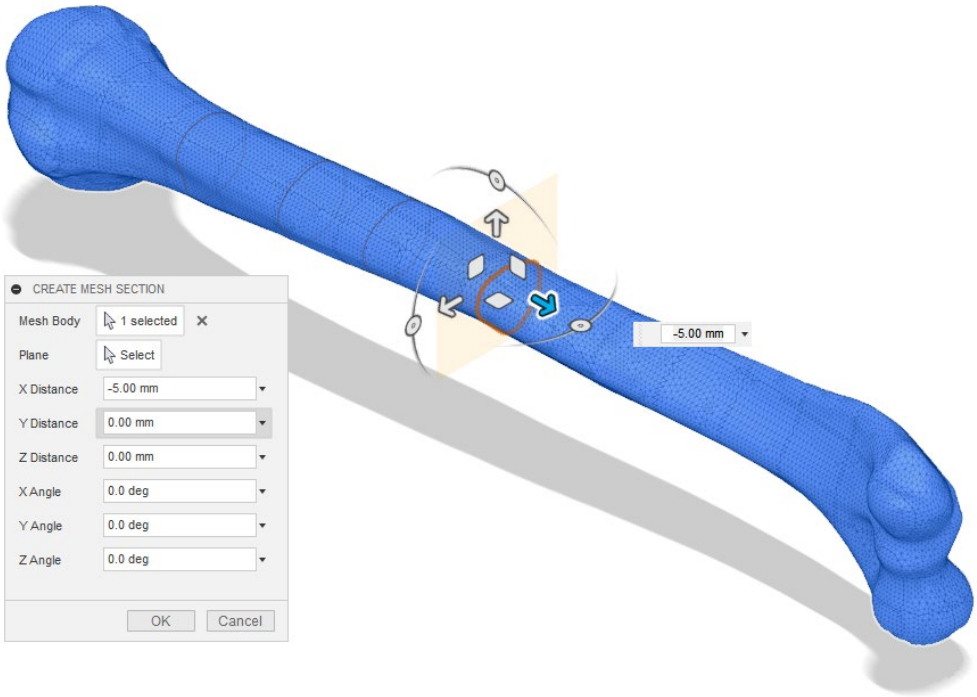
1. Meshlab FREE (low quality)
2. ReCap Photo (Subscription, high quality)
3. Netfabb Standard (Subscription)



You may be able to get better quality quad output from Meshlab than I did, but ReCap Photo did a perfect job. This video does a great job of explaining the process.

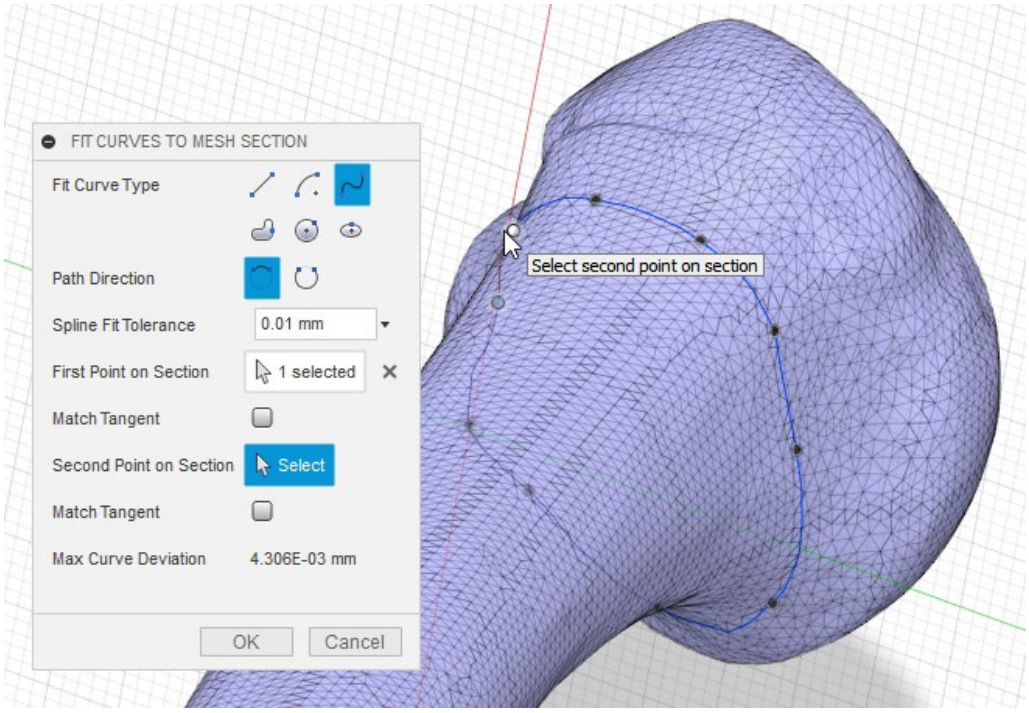
The other option, and sometimes it's the only viable one, is to import the stl into Fusion, then reconstruct t-spline faces over the top using the Face & Pull commands.

66. Create Mesh Section



YES! You can create sections through meshes. You have to be in the Model Workspace though, not the Mesh Workspace. From the Sketch ribbon drop down, under the Mesh menu towards the bottom, select Create Mesh Section. Then you just select the mesh body you want to section, and either precisely or visually position the plane through which you want to take the section. Done.

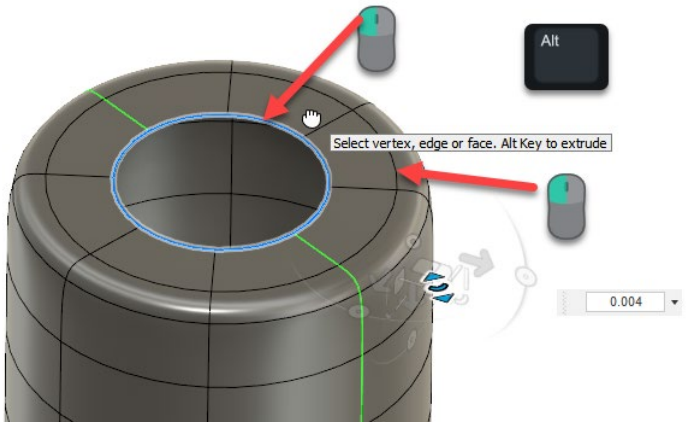
67. Fit Curves to Mesh Section



IN THIS CASE, USING THE CLOSED SPLINE TOOL IS MORE APPROPRIATE

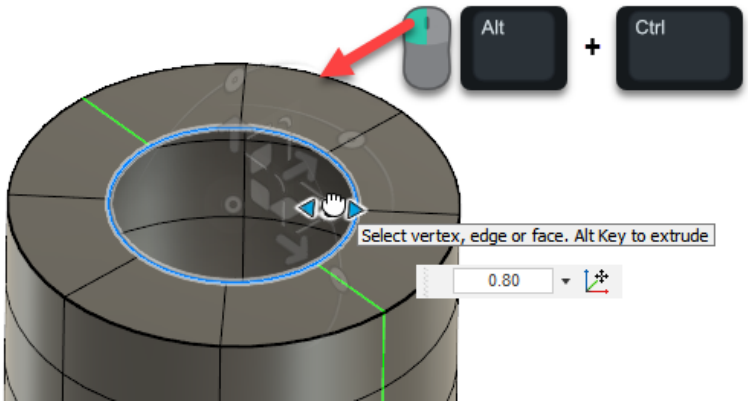
Once you have created a Mesh Section, you can use the Fit Curves to Mesh Section tool to build an open or closed profile to use for modelling. This dialog has an extensive set of options allowing you to effectively reverse engineer your mesh into a full blown NURBS surface or solid. If you know that sections of the mesh should be flat or have fillets, then you can use lines & arcs etc. Then in more organic areas of the mesh you can use splines or closed splines to get the geometry you need. Top tip… if you are going to use splines, try to use the Closed Spline tool where ever possible, it does a fantastic job of creating great geometry. Once you have a series of sketches with valid profiles, you can use them to 3D model your component.

68. Alt to add edges in Sculpt



With the Edit Form command active, select an edge, edge chain, face or face chain. But before you start tugging on one of those handles on the widget, mash down the Alt key and hold on. With each successive tug on your widget handle of choice, Fusion will gift you a new edge to play with. Pretty handy for gaining more control over the geometry of your freeform Sculpt body. So, hold down Alt, click & drag, once your release you get an edge, click & drag and you get another edge.

69. Alt + Ctrl to add creased edges

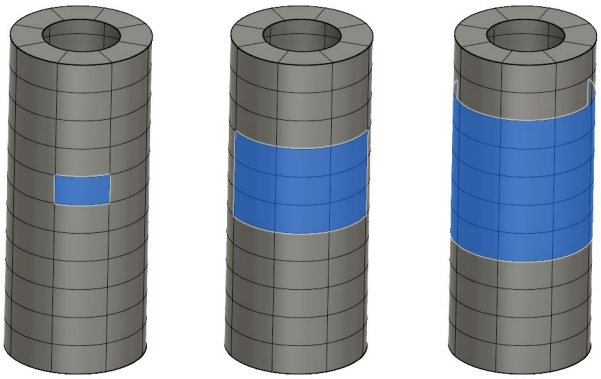


This one is similar to the previous tip, but instead of adding edges and maintaining G2 curvature on the adjacent faces, holding Ctrl as well will crease the edge instead. So Alt + Ctrl then drag to ‘extrude’ a face from a creased edge.

70. Make Uniform

In the image on the left, you see the model looks a bit stressed. Extruding out that extra face didn’t go down too well. We could leave it, but it doesn’t look good, and we will likely regret it later. The star points formed by extruding the face with the Alt key, are pulling on the rest of the model too much. Using the Make Uniform command, will redistribute those points relaxing the model, making it a much more reliable model to use moving forward.

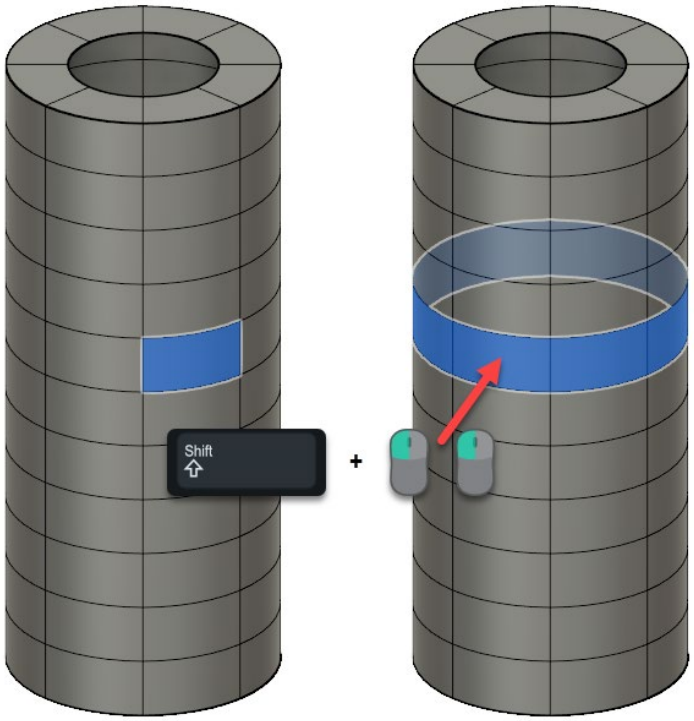
71. Shift + Up to grow Sculpt selection



Once you have your initial face(s) selected you can grow the selection out ‘radially’ by one face at a time by pressing the Shift & Up keys at the same time. Conversely Shift & Down will shrink the selection.

For more Sculpt shortcut keys, you can refer to this Fusion 360 shortcuts help page.

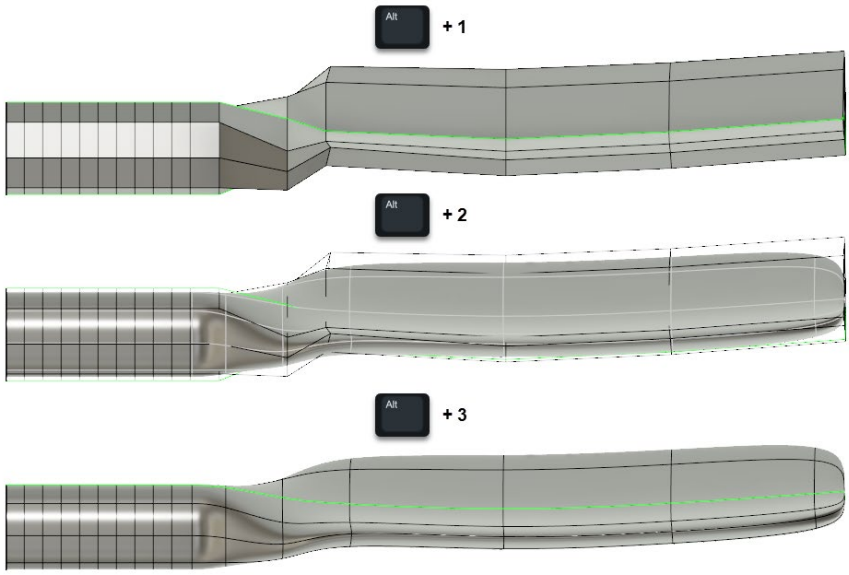
72. Shift Double Click



SELECT FIRST FACE(S), HOLDING SHIFT DOUBLE CLICK ADJACENT FACE

73. Alter Sculpt display mode

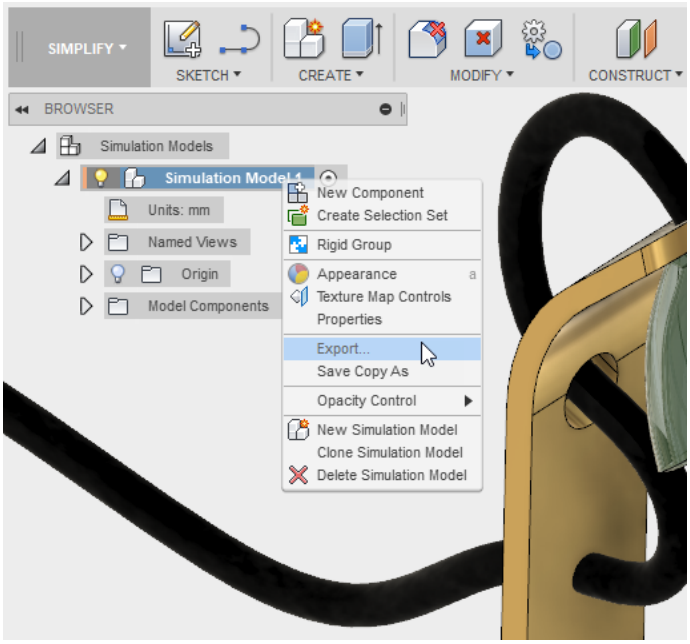
Alt + 1, Alt + 2 and Alt + 3 will toggle the Sculpt display mode between Box, Control Frame & Smooth respectively. Sometimes it's easy to get lost editing in smooth mode, so switching to Box mode allows you to see the frame controlling the surface in a simplified representation. You still use all the Sculpt tools as per normal.



74. Simplify

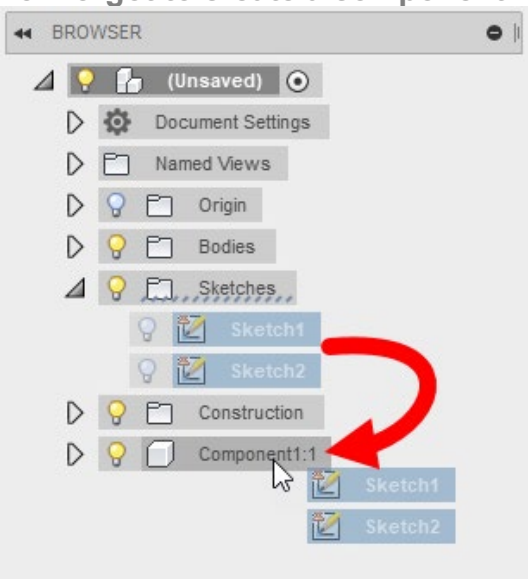
Hidden away in the simulation environment are a series of simplification tools. They are naturally aimed at simplifying the model for Finite Element Analysis (FEA), but thanks to a sneaky Export option in the Browser context menu, you can save out a simplified version of your design. This can be super useful if you must provide a collaboration model, but don't want to give away IP or overcomplicate things. BIM is a great example of needing to keep your models simple… so this may be useful for those of you creating products for buildings.

To access the Simplify workspace you must create a fake Simulation Study.



Assemblies

75. Forgot to create a component?



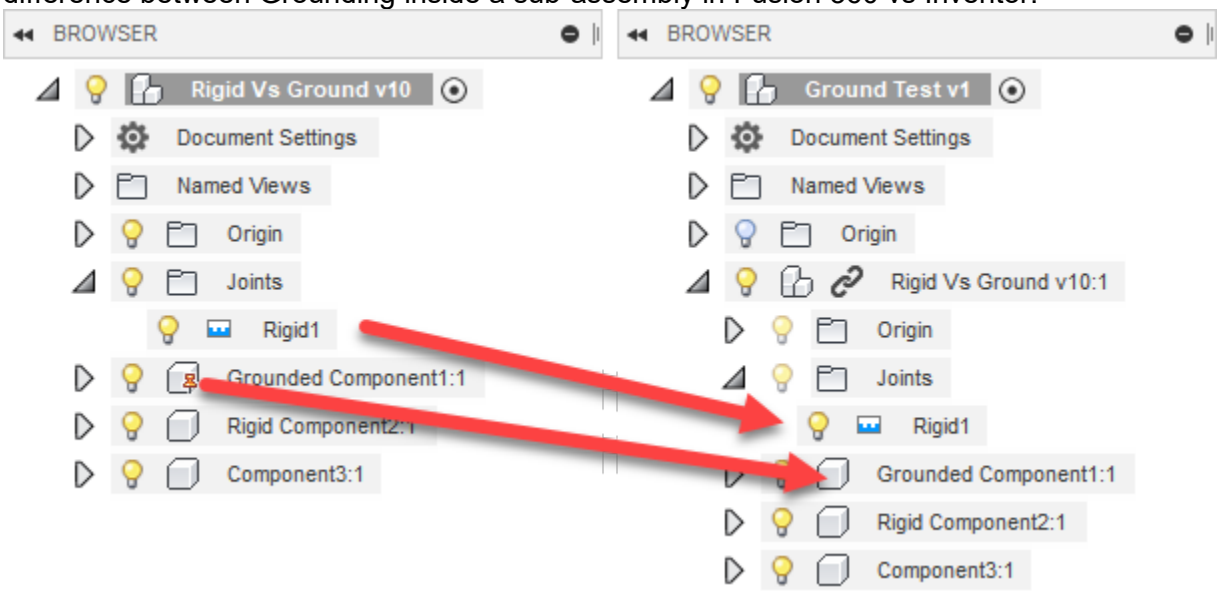
Tip number 1 was Rule number 1… but don't worry if you've forgotten to do it, but need to, there is a way to fix it up in a lot of cases.

1. Create a new component
2. Drag it to the beginning of the Timeline (or to where you want the component to be created)
3. Then drag the sketches from the top-level design document Sketches folder onto the newly created component in the browser.

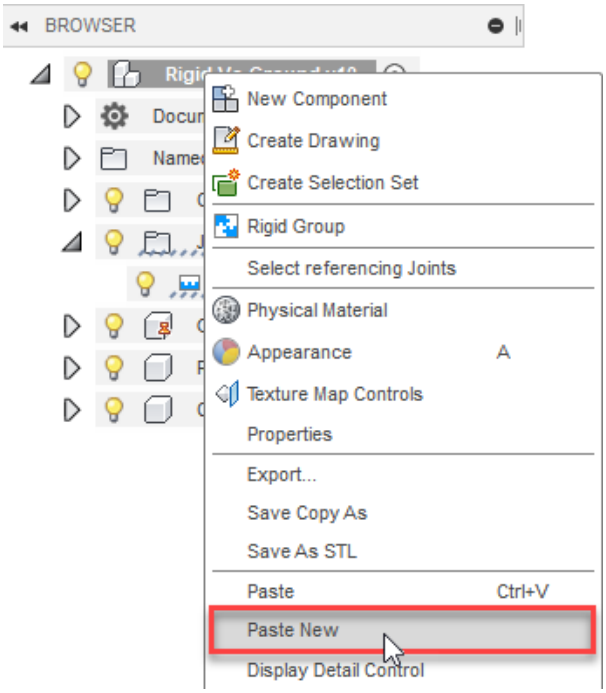
Once you have done this, the new component owns those sketches, and any features created using them.

76. Rigid Joint to Origin on 1st Component

In Inventor and Solidworks, you have the ability to ground your component. Generally this is best practice for the first component in your assembly, so you have a 'fixed' reference to work from for the rest of the assembly. However, in Fusion 360 the Ground option only applies to the component instance in its current assembly structure position. If you move it into a different level of the assembly hierarchy, then the Ground status gets removed. Effectively, Joints are carried with a sub-assembly anywhere it is used, but the Ground flag doesn't. This is the fundamental difference between Grounding inside a sub-assembly in Fusion 360 vs Inventor.



GROUND STATUS IS REMOVED. RIGID JOINT IS RETAINED.

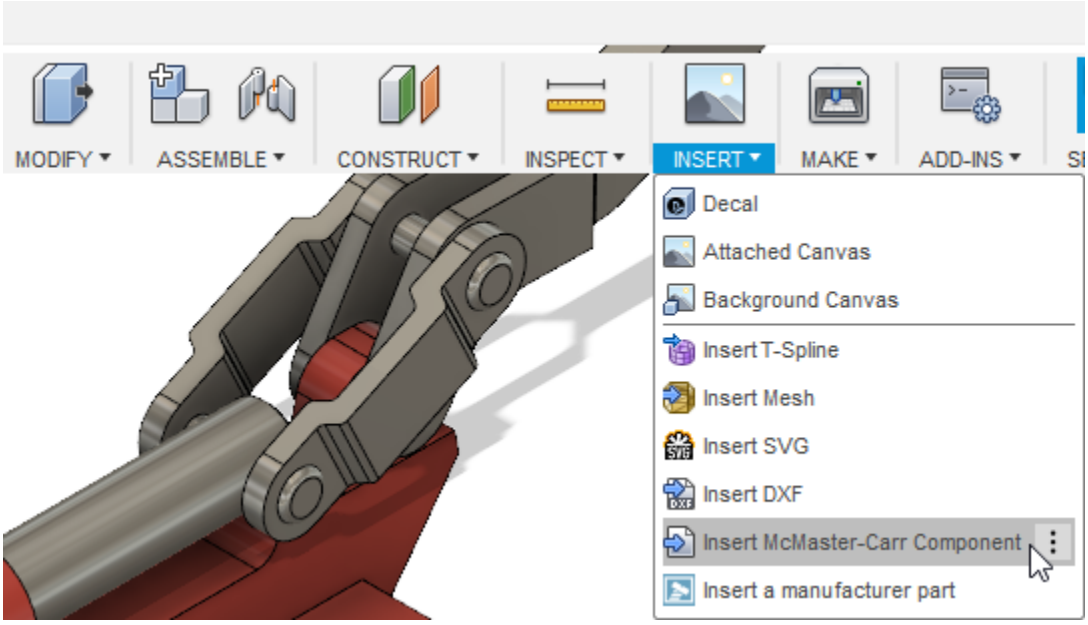


77. Paste New

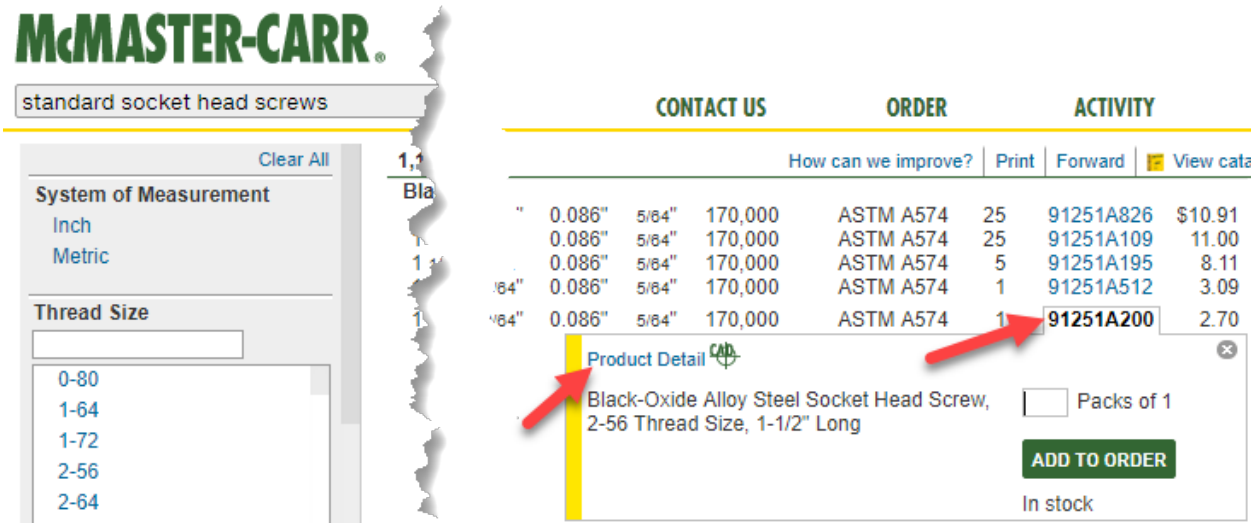
If you want to have 2 or more components of the same kind, then you can simply copy and paste the components. But what if you want the same basic component, with just a few tweaks and a new part number? That's where Paste As New comes into play. Copy the Component as normal, but this time, select Paste New

Doing so will result in a brand-new component of quantity 1, with a fully copied timeline of features, which can be edited independently of the original component. Changes you make to this one won't affect the other.

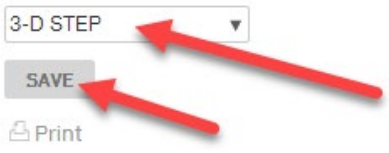
78. McMaster-Carr



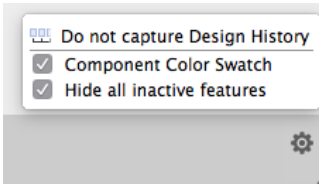
McMaster-Carr are an online engineering components eStore and they allow you to download 3D models for nearly all their catalogue. It's seriously impressive. Fusion 360 has integrated with their website, so from the Insert drop-down menu you can choose to insert one of their components.



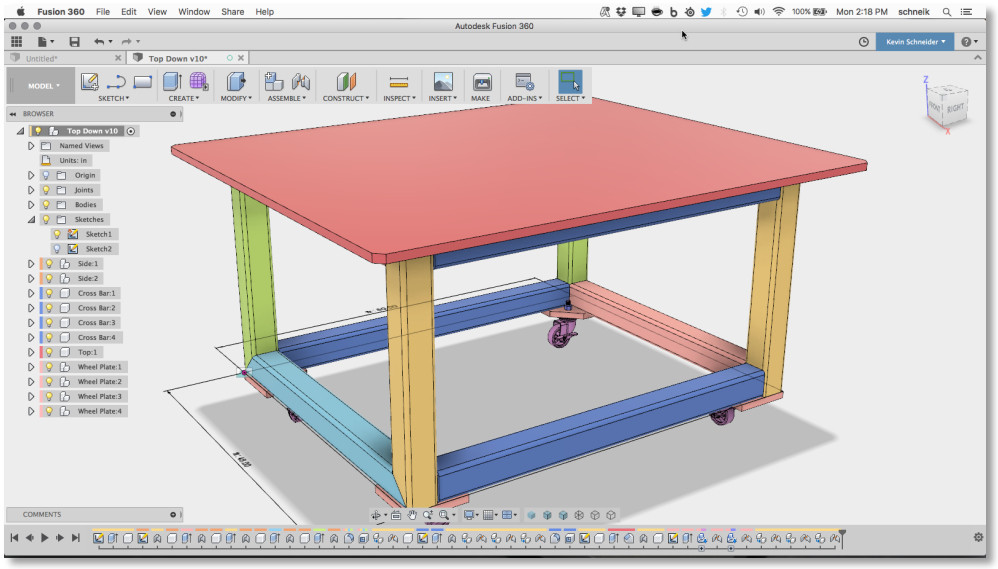
A new window appears allowing you to search and browse through their website. You can either search for a SKU directly, navigate through their menu system or use product filters to find what you need. To access the 3D models, you must select the part number hyperlink, then click the Product Detail link to get through to that product's page, scroll until you see a drop-down for selecting the file format you would like. I recommend you use the 3-D STEP file type, then click Save to have it inserted into your Fusion Design.



79. Component Colouring

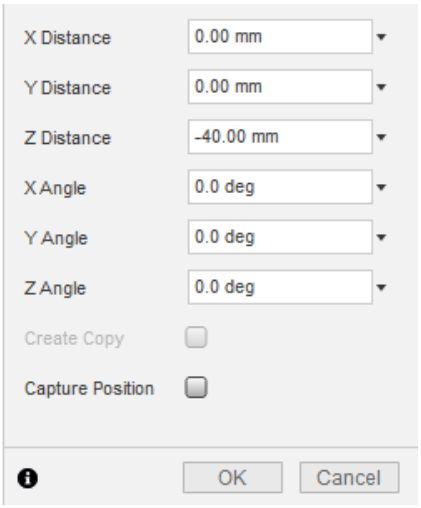


To enable this goodness just press Shift + N on your keyboard and Fusion's graphics window morphs into something from Willy Wonka's chocolate factory. Each component in the browser gets colour bar, which corresponds with all the features each component owns in the Timeline and the components themselves in the graphics window also take on the same colour.
If you click the gear icon on the Timeline, you can enable the colour bars in the Browser & Timeline without the components being coloured in the graphics window.

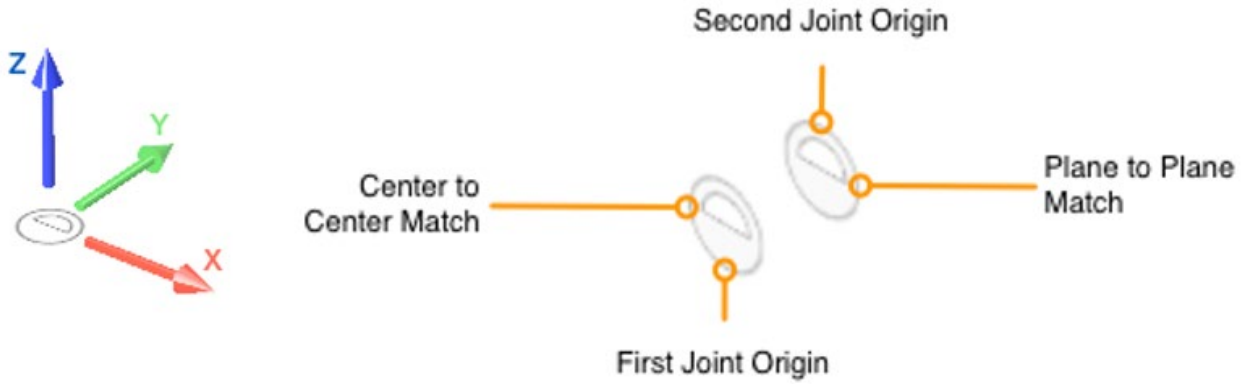


80. Free move on placement

When you insert a component from the Data Panel, or copy and paste to create another component instance, the Move/Copy manipulator appears on screen. If you are capturing history, then this initial placement of the component is a 'Free' move. Fusion doesn't try to track the position of the component until after it has been placed. So it's worth making sure you get it in a good spot for whatever you are going to be doing with it next.



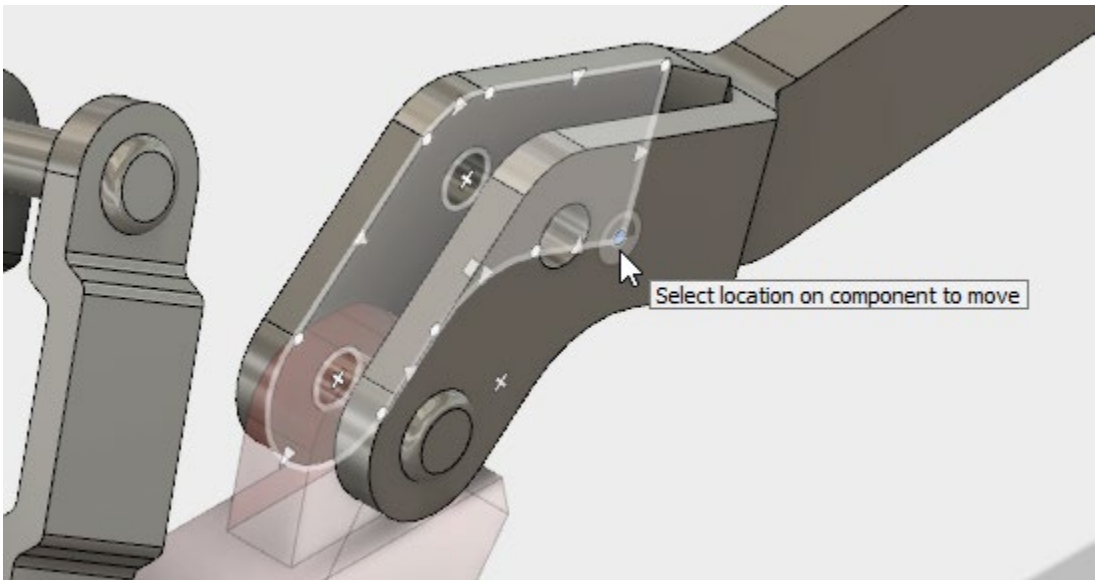
81. The Joint 'Coin'



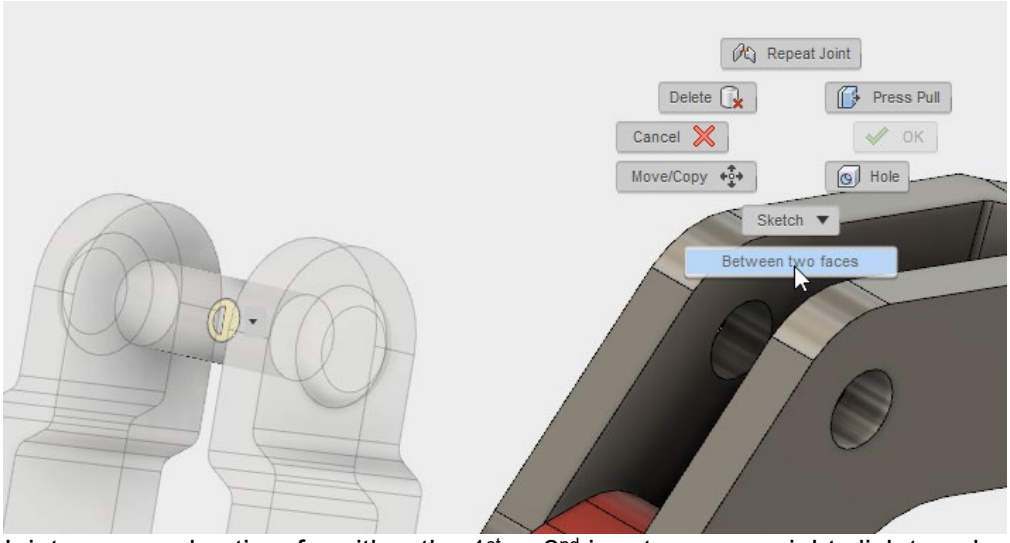
- Not used by 'As-Built' Joints
- The semi-circle line aligns with X axis
- The semi-circle arc points towards Y axis
- The Z positive direction is driven by the XY direction and is normal to the coin
- The Coins align by default, but can be adjusted using the Angle & linear offset input boxes in the Joint dialog

82. Lock Joint selection

Sometimes the Joint origin you need to select is in thin air or occluded by another face. As a result, it can be tricky to lock onto and select. Holding down Ctrl on your keyboard will lock the face or edge selection, so you can mouse over to the origin you need to select to get the job done.



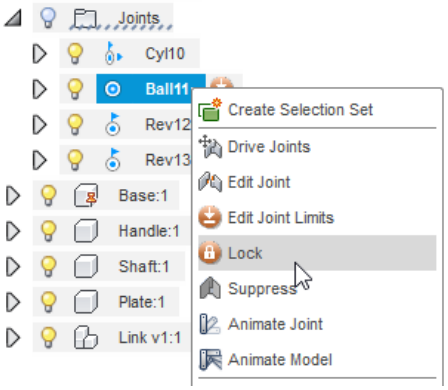
83. Midplane Joint



With the Joint command active, for either the 1st or 2nd input, you can right click to select the Between two faces option from the context menu. Once you have selected the two faces, you only need to select a reference for the Joint Origin from any of the valid snap points on that Component. The snap point doesn't have to be on one of the seed faces.

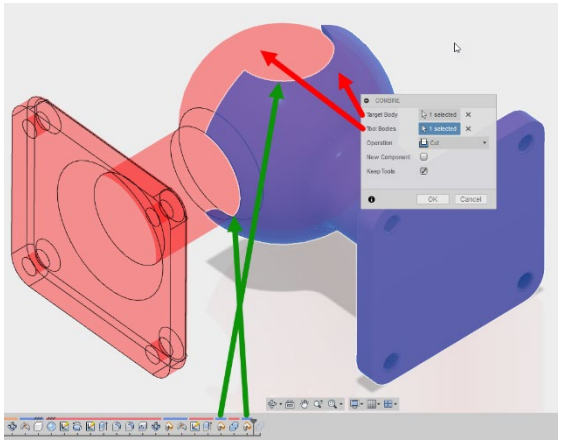
84. Lock Joint

This is handy if you are trying to troubleshoot your joints, or temporarily restrict motion without completely suppressing the Joint. Suppressing removes the Joint from motion calculations, but locking it just stops it from moving, similar to Rigid, but without changing the Joint type.



85. Combine Cut using Assembly position

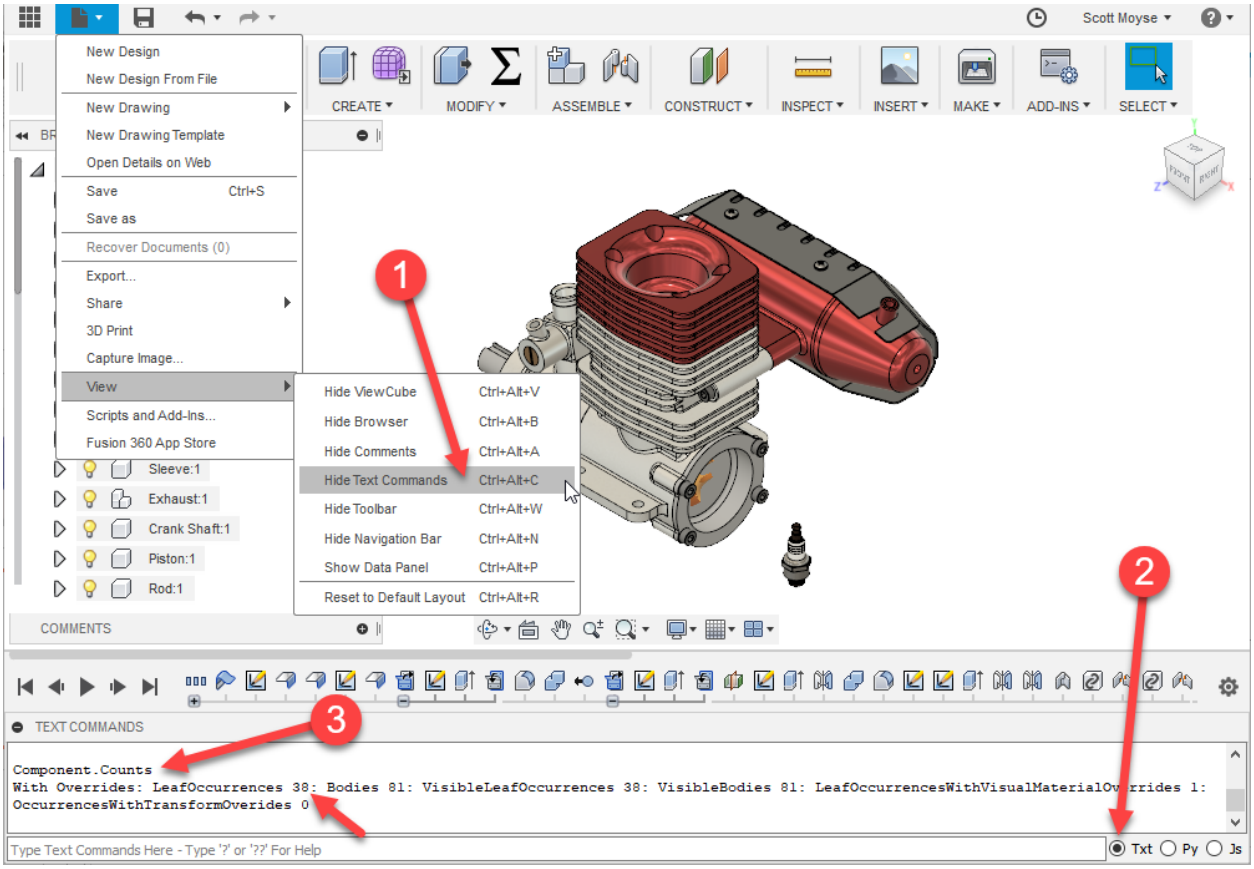
Because Fusion 360 tracks the history of its components positions, you have the ability to use the components themselves to drive the geometry of adjacent or intersecting parts at various positions during their motion. Just position your components, capture their position on the timeline, then use your modelling operation of choice. This ball joint is a great example.



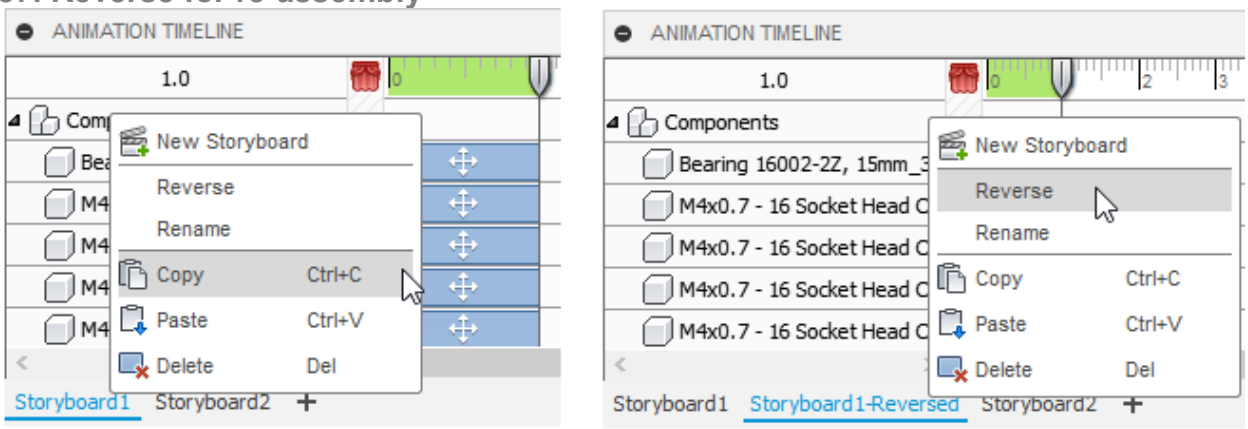
86. How many components are there?

There are a number of addins you can install to get this information and there are some scripts on the forums you can use as well. But there isn't anything in the Fusion user interface which easily gives you this information. However, you can access this information from the Text Commands pane:

- Turn on the Text Commands pane (File > View > Show Text Commands)
- Switch to the Txt radio button
- Type Component.Counts & hit Enter
- Look for LeafOccurrences value



87. Reverse for re-assembly



The animation workspace is where you go to create your exploded views for your drawings. But you can also create cool assembly animations and save them out as videos. Generally, the way things go together is the reverse of the way they come apart. So, after you have finished creating your exploded view animation, you can copy the Storyboard, then paste it as a new one. Maybe be a good CAD person and rename it to *exploded-reverse* or something equally as meaningful. After that you can right click on that storyboard and select Reverse from the context menu, then hit play to feel a sweet sense of satisfaction having done a good day's work in just a few seconds.

Drawings

89. Special Characters

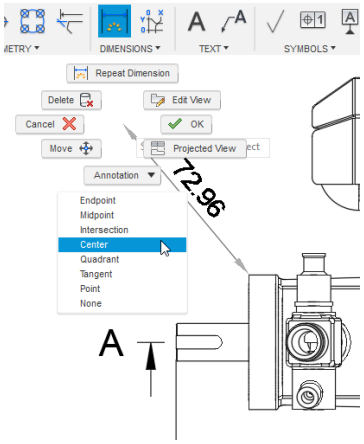
Fusion doesn't support special characters, and several forms of engineering notation just yet. So if you have to get this information onto your drawings, then it can be achieved via the use of Alt codes or Unicode. Just type the codes into a text box on your drawing. Here are a few references you can use to find the codes you need:
https://knowledge.autodesk.com/support/autocad/learn-explore/caas/CloudHelp/cloudhelp/2019/ENU/AutoCAD-Core/files/GUID-518E1A9D-398C-4A8A-AC32-2D85590CDBE1-htm.html
http://www.thecontrolsfreak.com/2011/tech/alt-codes-you-look-smarter-simple-key-strokes/
http://www.engineeredbydesign.co.uk/Pages/AltCodes.php

90. Shift to suppress OSnaps

When placing dimensions, you can toggle between a 'grid' like snap for the position or completely free movement. Most of the time using the grid snap helps you to quickly align multiple dimensions, whereas having fully free movement will allow you to precisely squeeze in additional dimensions. This behaviour is changed by using the Shift key to toggle it on & off.

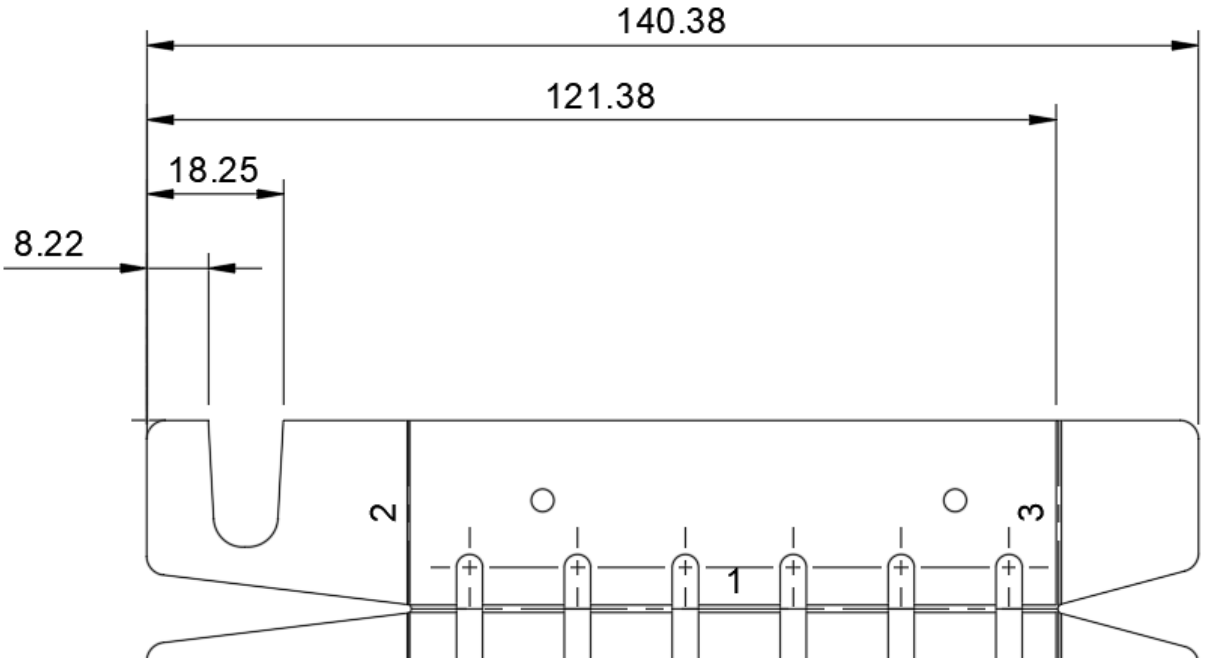
91. Smart Dimension OSnap

When placing a Smart Dimension, you can choose to limit the objects your dimension points will snap to. Prior to selecting the point, you want for dimensioning, right-click and select an override from the context menu. You will need to do this for each dimension point you pick, since the selection resets back to auto.

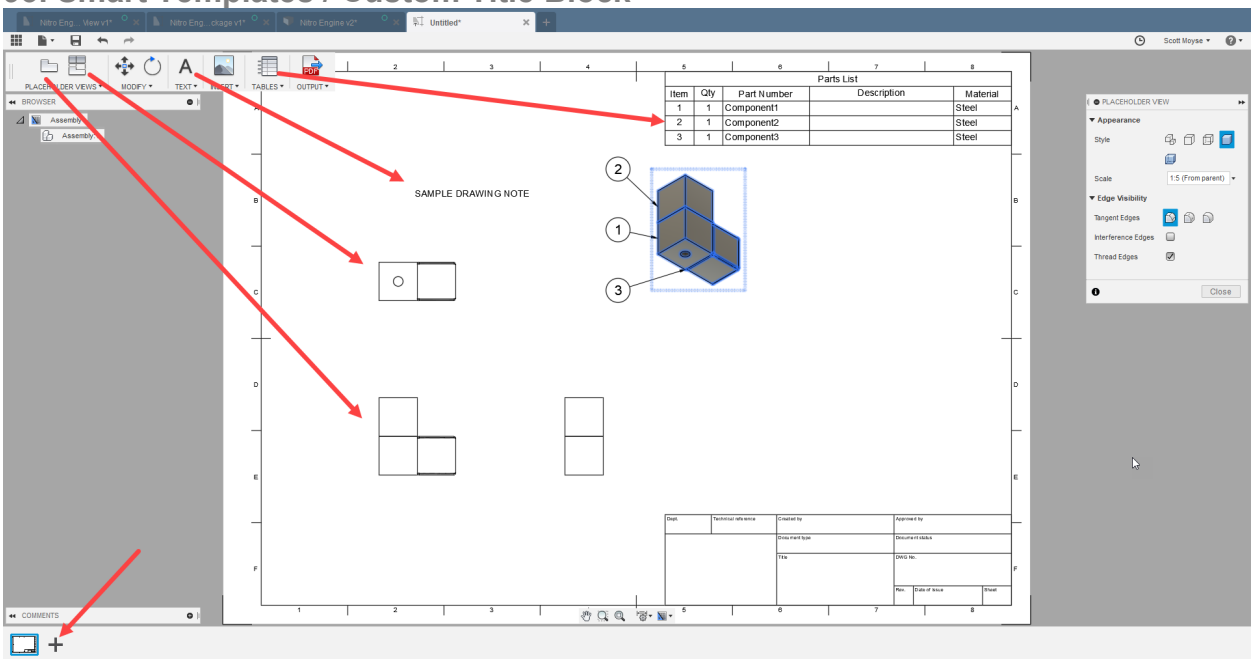


92. Stack Dimensions

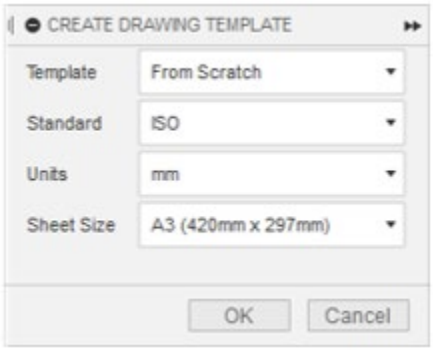
When creating a new dimension, while placing it, click on a dimension that it should stack with. They will stack appropriately, with the larger overall dimensions to the outside, and smaller dimensions on the inside.



93. Smart Templates / Custom Title Block

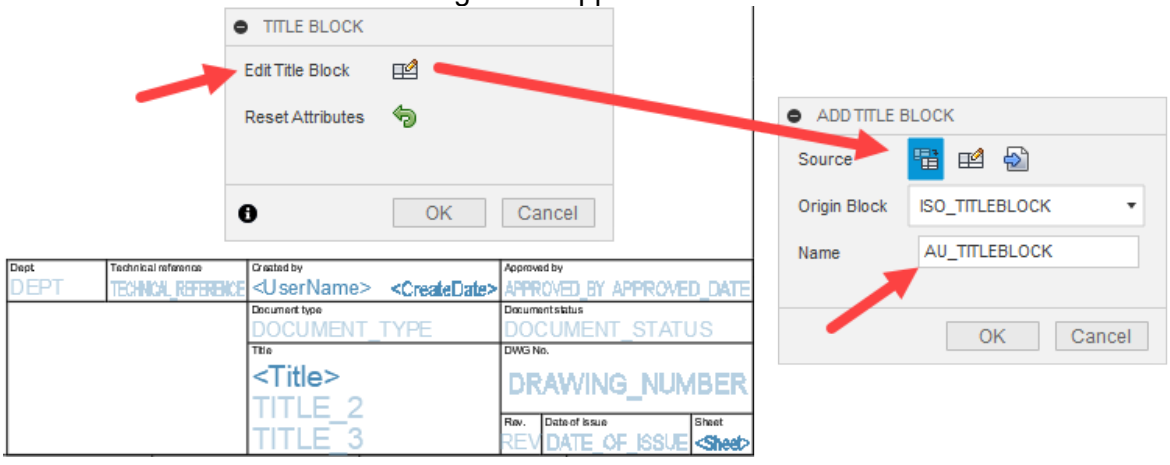


Using the normal Create Template command in Fusion 360 also allows you to create Smart Templates. The difference between the two is a standard template will generally only have a single sheet with zero drawing views. Whereas a Smart Template just adds placeholder views and tables and can contain more than one sheet of varying types.

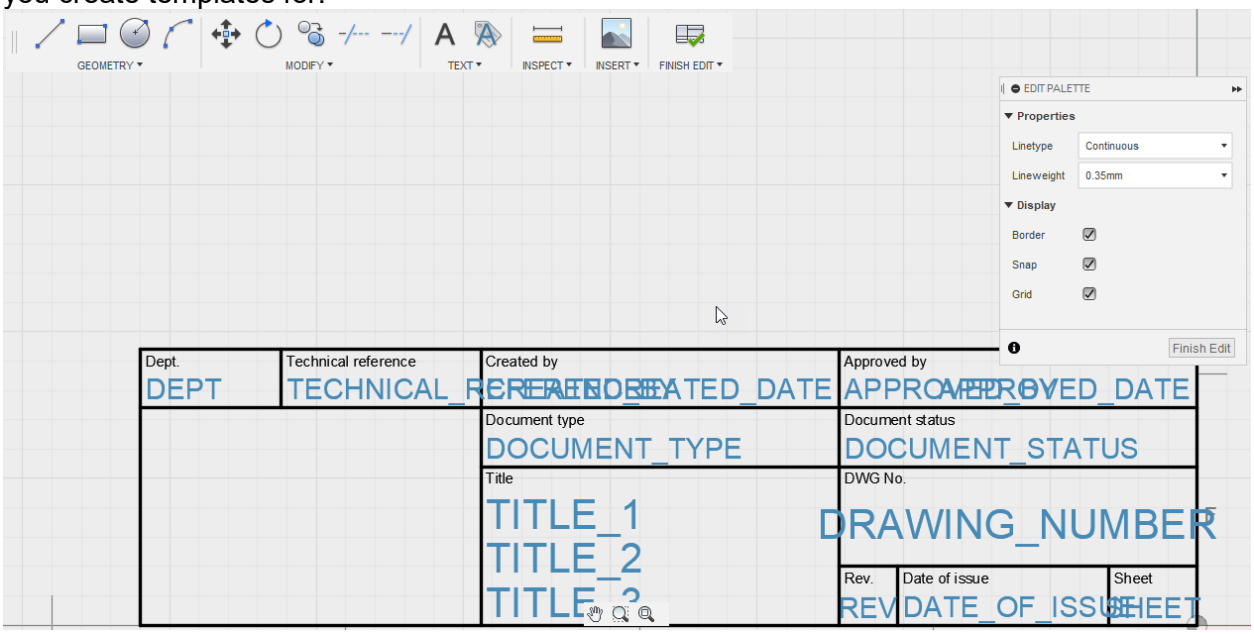


To get started, from the File drop-down menu select New Drawing Template. In the resulting dialog, choose the standard, units and primary sheet size. You can add different sheet sizes to the Smart Template later.

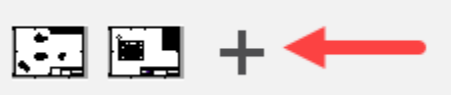
If you want to modify the default Title Block, then double-click on the one in the sheet which gets loaded. Then click the Edit Title Block button on the dialog which appears.



Next you must choose if you want to use an existing Title Block as a starting point, start from scratch or import one from AutoCAD. You're mostly likely going to want to select the default Title Block as a starting point. Give your new custom Title Block a name at this point before clicking OK. Use the sketch, text & attribute tools included on the toolbar in this environment to build up your custom Title Block. You may want to create more than one to suit each sheet size you create templates for.



Once saved, you return to the main template workspace, where you can insert placeholder views and tables, as well as any drawing notes you may want to include. Create a new sheet by clicking on the + sign next to the last sheet created at the bottom of the screen.

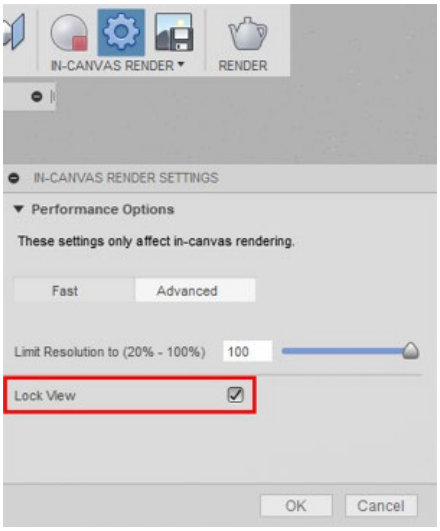


You can create sheets for:

- Assemblies
- Storyboards (created in the Animation Workspace) which can either be exploded views or configurations of your design assembly.
- Component
- Folded Components
- Flat Patterns
- Linked full sheet parts lists Tables

On each sheet you can pre-stage elevations, isometric & section views. For each view you can define view styles as you would when placing a normal view in a normal drawing. Once you are happy save the template and choose a meaningful name, since it will appear as a selectable option in the Create Drawing dialog in the future. Note: if you are using a Fusion Team Hub, instead of the standard personal Hub, then you can create a templates Project and save your templates into that. But still refer to them from other projects.

To create a drawing package using your Smart Template, from your Design you want to document, select New Drawing From Design, and choose your newly created Smart Template from the Template drop-down. You may have to browse to find it the first time. Enjoy!!

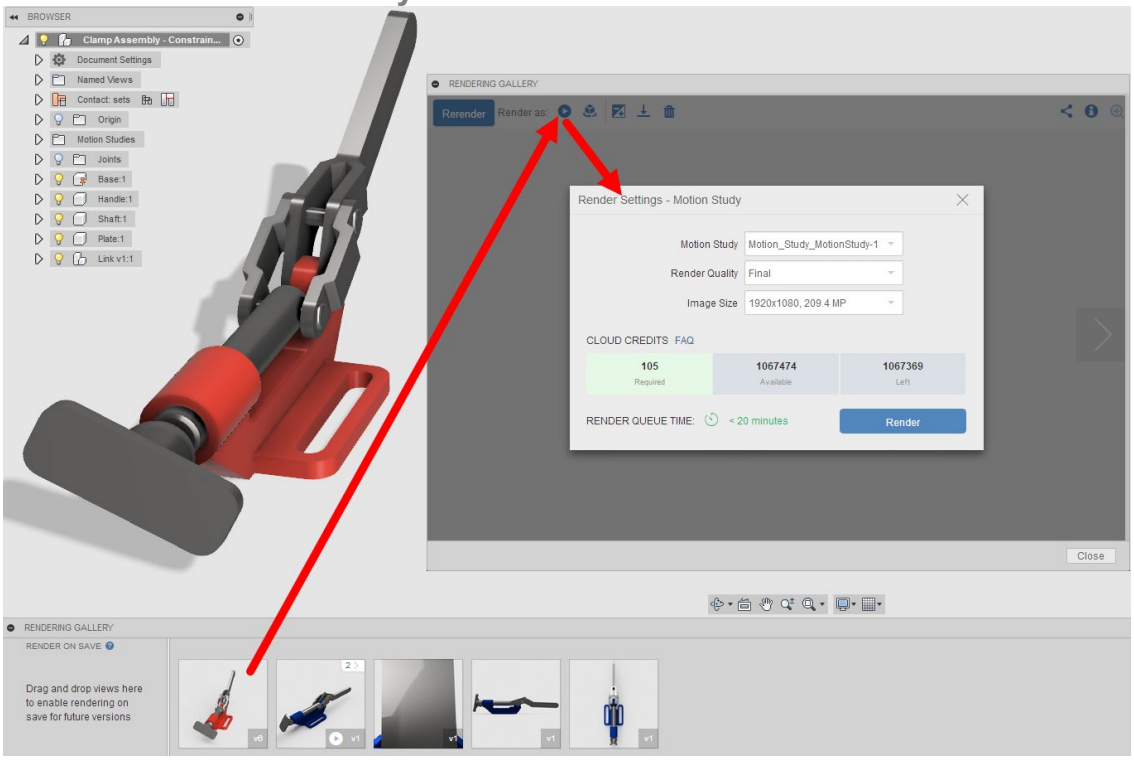


Rendering

94. Lock View

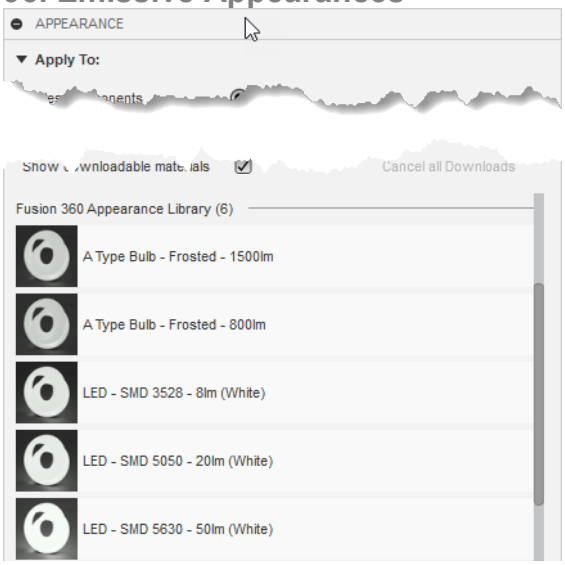
When rendering using Ray Tracing in canvas, the slightest knock of your mouse or keyboard will reset the render. NOT what you need when you're angling for that final result. Fear not, all you have to do once you have the view setup, is click the Gear Icon on the rendering Ribbon and activation the Lock View checkbox.

95. Re-Render as Motion Study

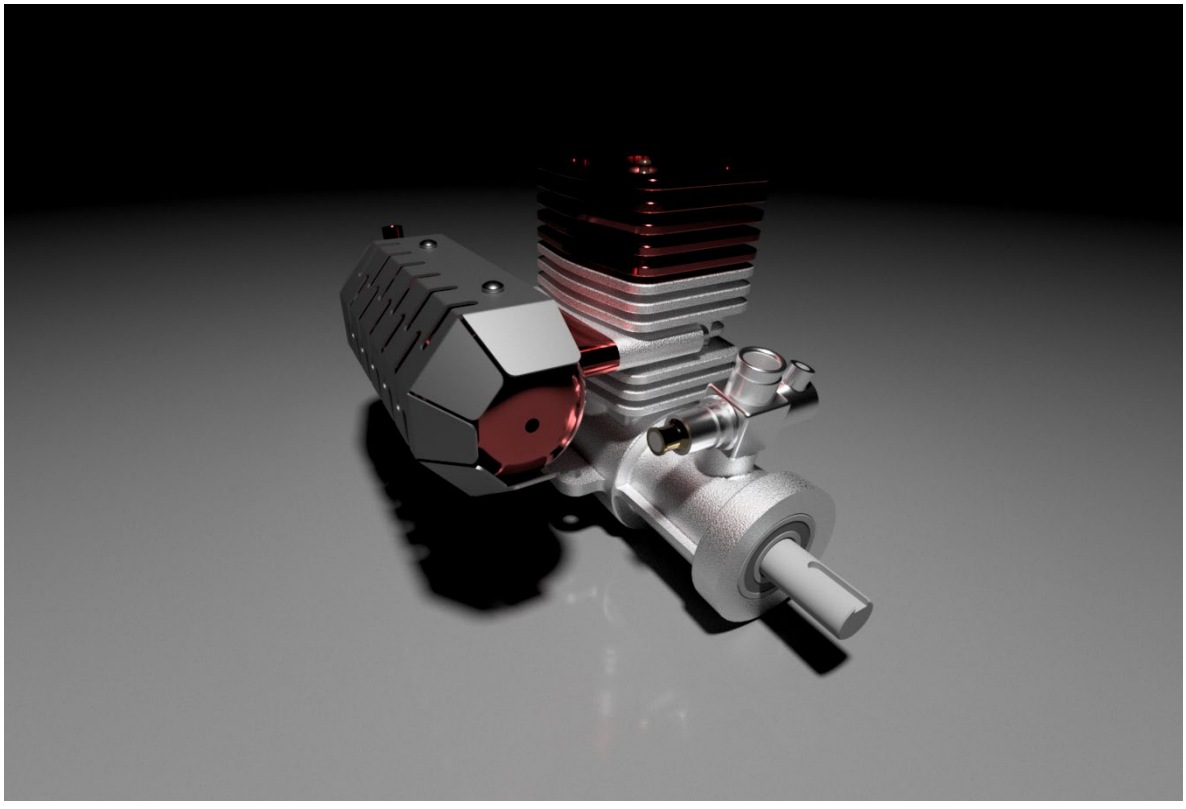


Get the motion you would like to render out as an animation sorted by defining all your Joints and define a motion study. If you create a Named View for the camera angle suitable for the motion study, then a preview rendering will be created by Fusion automatically on save. Switching to the Render Workspace, have a look at the preview thumbnails in the Rendering Gallery at the bottom of the screen. If you click on the one for the Named View you created, in the resulting window you will have the option to Render a Motion Study. Click on that bad boy, select your motion study and quality settings and click Render. You will need some Cloud Credits to make this happen. Then crack on with some other work while the Cloud does its thing in the background.

96. Emissive Appearances

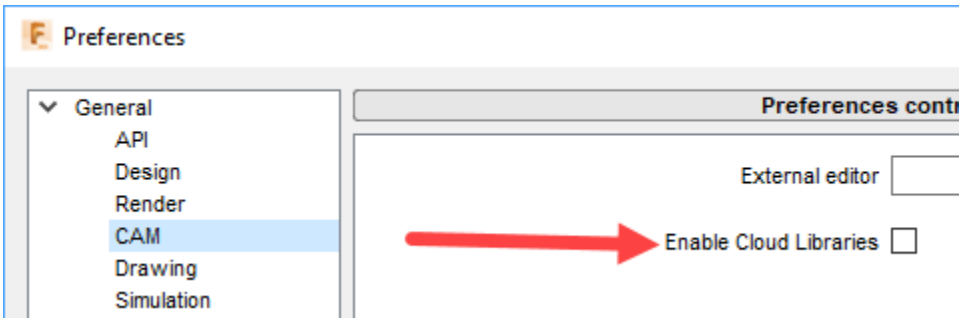


There are a number of emissive appearances available in Fusion 360. These can be used to model up custom lighting rigs for rendering. Out of the box Fusion 360 get the light it needs from the environment in the Rendering Workspace. By creating 3D models with emissive appearances, you can introduce additional light sources. A really simple trick is to add two additional components to your design, the first being a Sphere, and the second a cylinder or a cone. Set the material of the Sphere to Air, so its becomes completely invisible, and assign an LED or Bulb appearance to the other component. Both of those appearances will emit additional light into the scene and can be directed onto certain objects easily. Position the Sphere so its center is at the center of subject you want to render, then use a Rigid As-Built Joint to lock it in place. Then use a Ball Joint to lock the axis of the conical/cylindrical component to the center of the sphere. At this point you can just drag the light emitting object around the sphere into the position you want. Just into the rendering environment and take a look at the impact it has on your rendering. Think about modelling in other objects to give the light more to bounce off, maybe walls & a floor would be a great start?

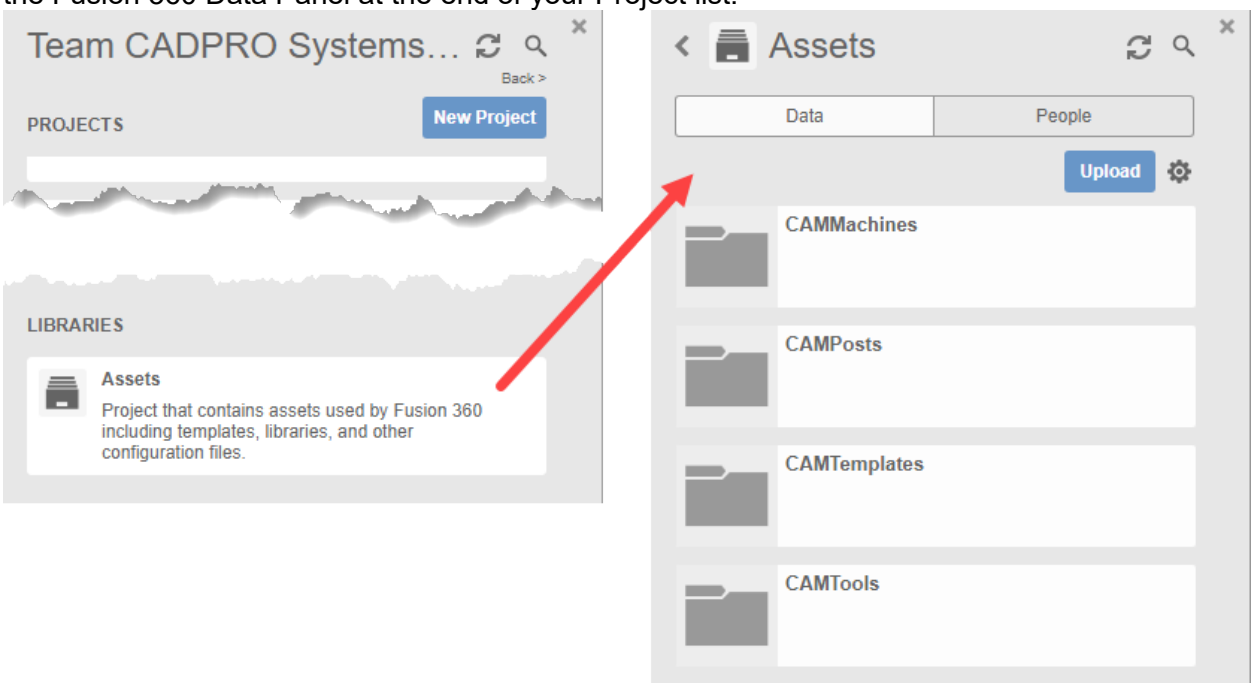


Manufacturing

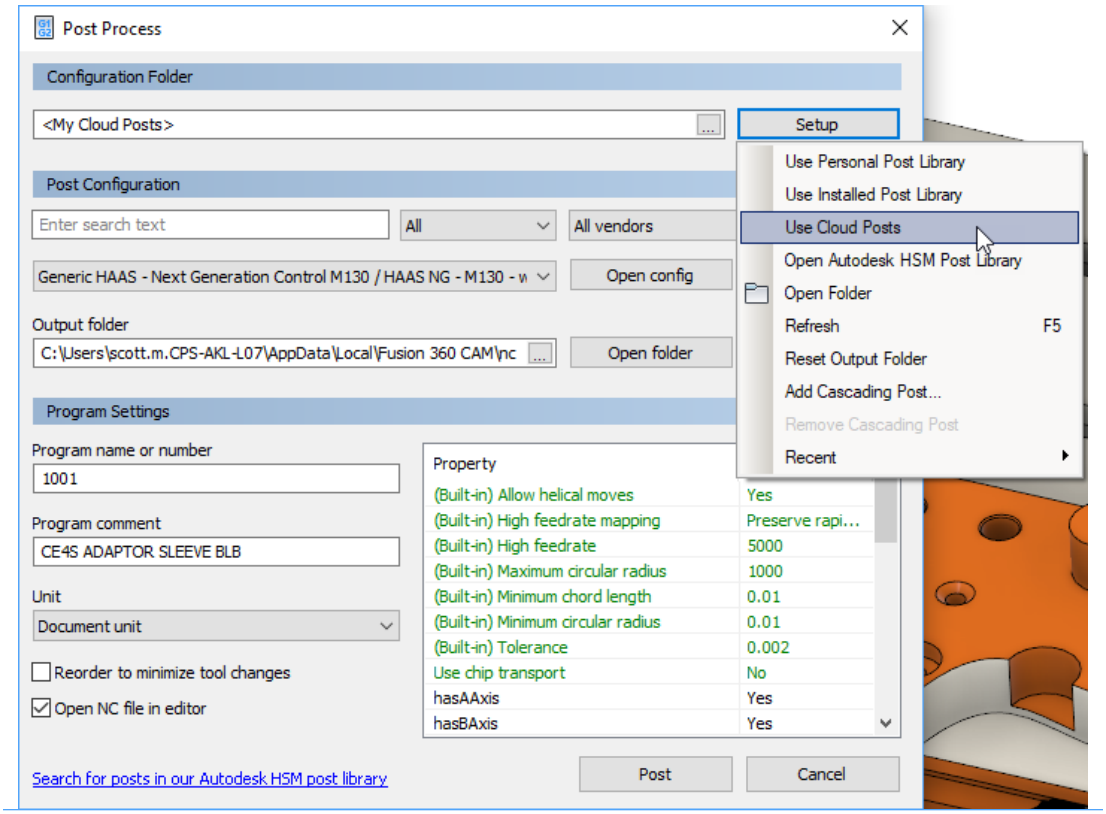
97. Cloud Libraries / Fusion Team

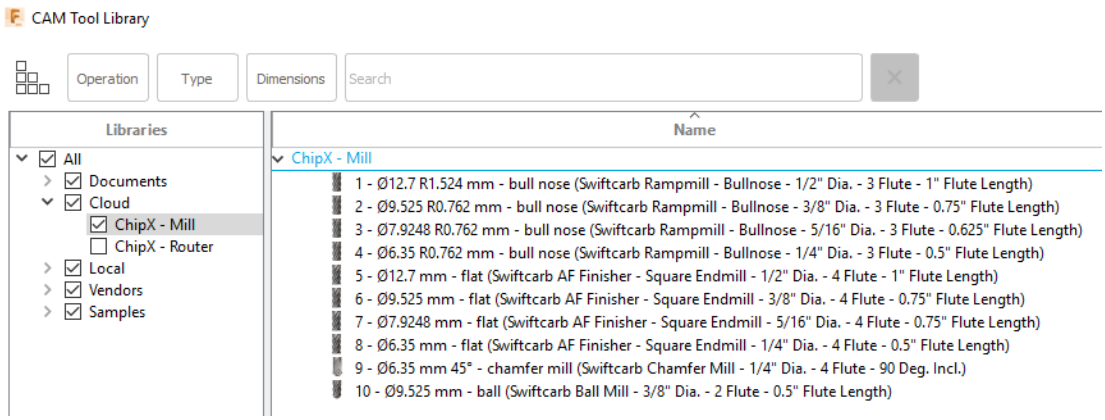


If you are a one-man band, then you should still consider activating CAM Cloud Libraries in your Fusion 360 Preferences. Doing so will create an Assets Project in your Hub which will appear in the Fusion 360 Data Panel at the end of your Project list.

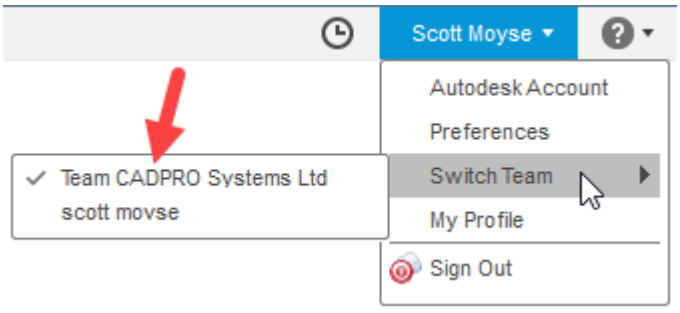


Within that Project, 4 folders are created from Machine Configurations, Post Processors, Toolpath Templates and Tool Libraries. Upload your proven-out Post Processors into the CAMPosts folder. Setting this up means that you can log into any device and access all your own custom posts, tool libraries and templates if you have created any. And because those files are being looked after by Fusion you can version them all just like you can with your Designs if you so choose.



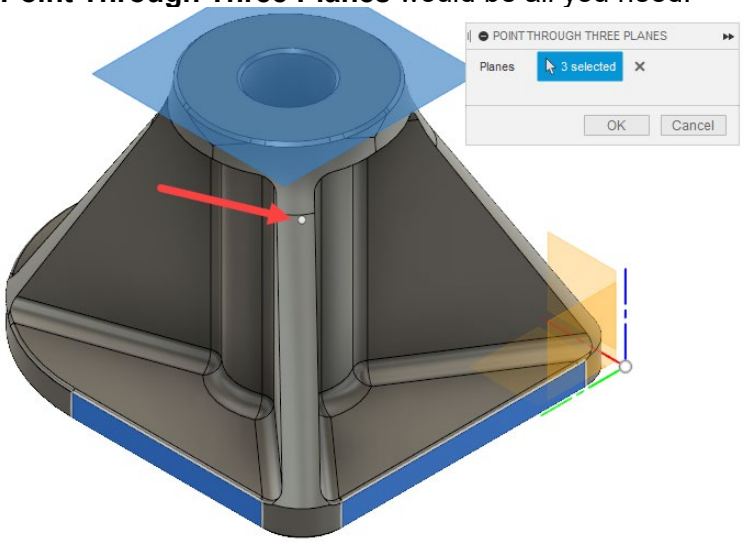


If you are using Fusion Team, then setting this up is a must, because it tightens up your programming users' collaboration. If you enable Cloud Libraries in each users' account, then they will all be using the same posts, templates and tools as each other, all seamlessly integrated and managed natively by Fusion 360. Just make sure each user switches over to your companies' Fusion Team Hub.

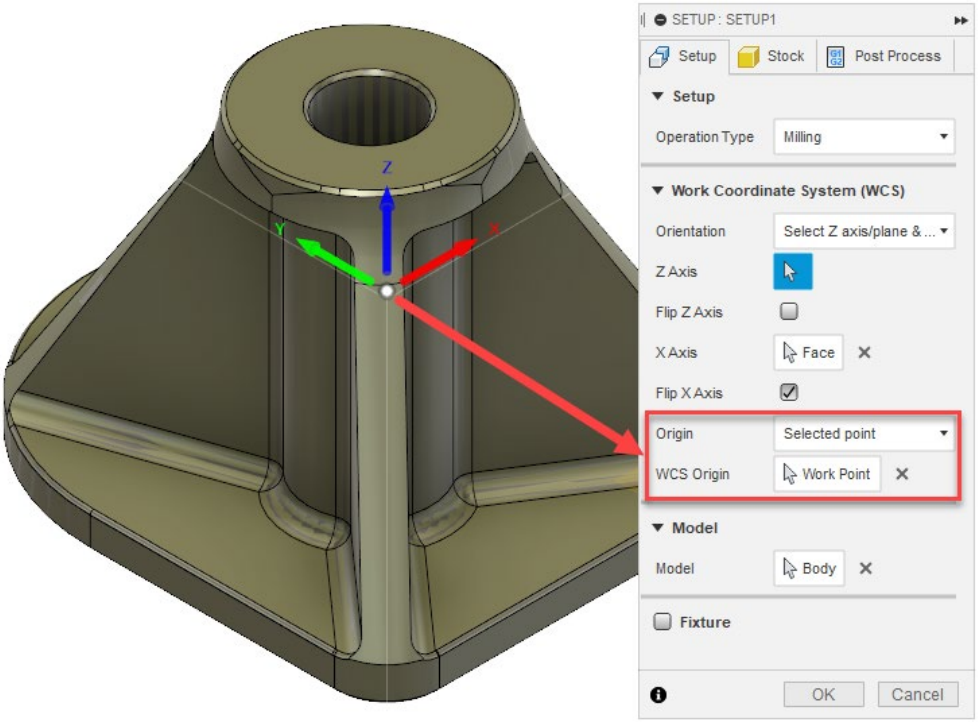


98. Selected Point WCS Origin

The geometry of some parts and the details you are required to machine, sometimes means that finding a zero-reference point on the part can be tricky. You may have to 'touch-off' on 3 different faces on the part to find it. This results in a 'virtual' reference point which you can't define with the options in the CAM Setup alone… you need some 'help' from the Modelling Workspace. You may need to create some workplanes and/or sketches to find the point you need, but often the **Point Through Three Planes** would be all you need:

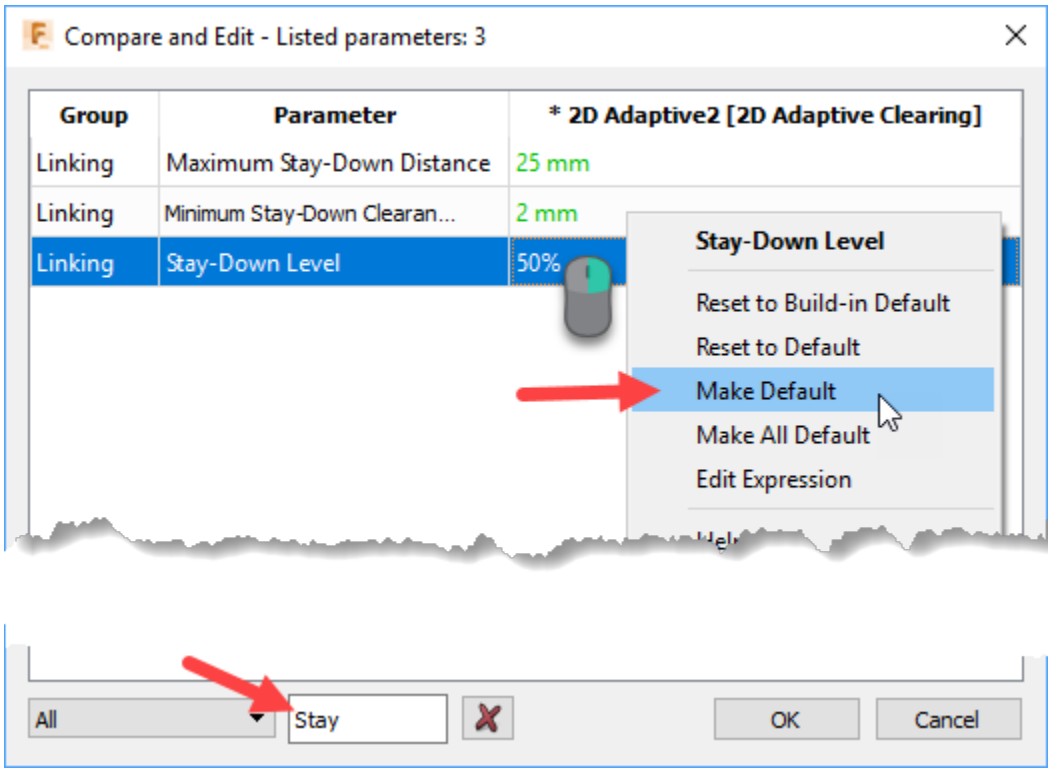


From there you can select that Work Point as your reference when creating your CAM Setup. Just choose Selected Point from the Origin drop-down, then choose the Work Point you have created for the resulting input requirement.



99. Compare & Edit Defaults

You may already know that you can right-click on any input box while creating or editing a toolpath and set the value as the default. But you can't do that with drop-down values and checkbox states. To do this you must right-click on the toolpath and select Compare and Edit. Then you can search for the setting name, set the value you want (if you haven't set it yet), then right-click on that input box and you will see the options you need to set the default. Be sure to select Make Default and not Make All Default. Selecting the latter will result in all values set for that toolpath becoming the default moving forwards, not just the value you want to set as a default.



100. G-Code Syntax Highlighting for Brackets

When you post process toolpaths for the first time in Fusion 360, Brackets gets installed. The trouble is, Brackets is an open source simple text editor used for software development. It's not designed for reading the G & M codes output for CNC machines. However, someone has created an extension for it which improves things somewhat. Check out this video to learn how to implement it and in the description there is a download link for the Brackets Extension.

101. Fusion 360 Adoption Portal

Fusion 360 is a rather large product, with tonnes of different skill sets required to master it. If you want to learn some new skills and excellent place to start is the Fusion 360 Adoption Portal, be sure to check it out.