AUTODESK Fusion

Autodesk Fusion – 15 tips and tricks Volume 1
How to improve your workflow in the Design space.

AUTODESK

Welcome to the Autodesk Fusion tips and tricks guide for the Design workspace, Volume 1. This guide is designed to help you get the most out of your CAD modelling capabilities and is open to users of all levels. At the end of this, hopefully you would have picked up some extra ideas that will improve your overall productivity and allow you to design more and better. This guide is broken down into rough stages of experience with using Autodesk Fusion, however all users will find each of these tips useful if they're not already using them.

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General Housekeeping:

Let's start with some basic housekeeping tips that I'd encourage all users of Autodesk Fusion to employ.

A - Get a mouse with a scrolling wheel (and a spacemouse if budget allows).

First things first, make sure you've got the right tools. If you haven't already, get yourself a mouse with a scroll wheel. This is almost a necessity to modelling and will allow you to pan and rotate your 3D object with ease. If your budget allows, I'd recommend also getting a space mouse as this gives you full movement control in one hand.
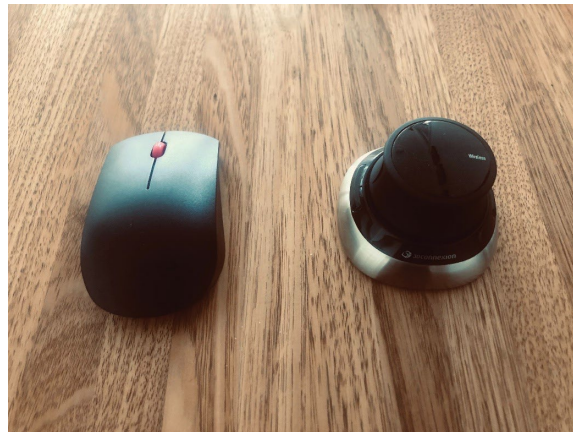
## B – Use the support services for when things are not working with the software.

Your Autodesk Fusion license comes with software support. Hit them up whenever you have an issue with installation, license access and any other glitches or issues with the software that might arise. Head to the Fusion webpage (https://knowledge.autodesk.com/contact-support) and choose from the many way to get in contact with them.

C - Use Fusion Teams to manage your files.

Although you can access your files inside Autodesk Fusion, Fusion Teams (the online file manager for Autodesk Fusion) gives you a much better level of control and management over your files. Not only is the layout better but you can archive, transfer and download files or folders en mass. You also have more options to manage your team members, assigning different levels of access, and you can easily pull up all the important information related to your files such as properties, components and lifecycle states (for those using the Manage extension).
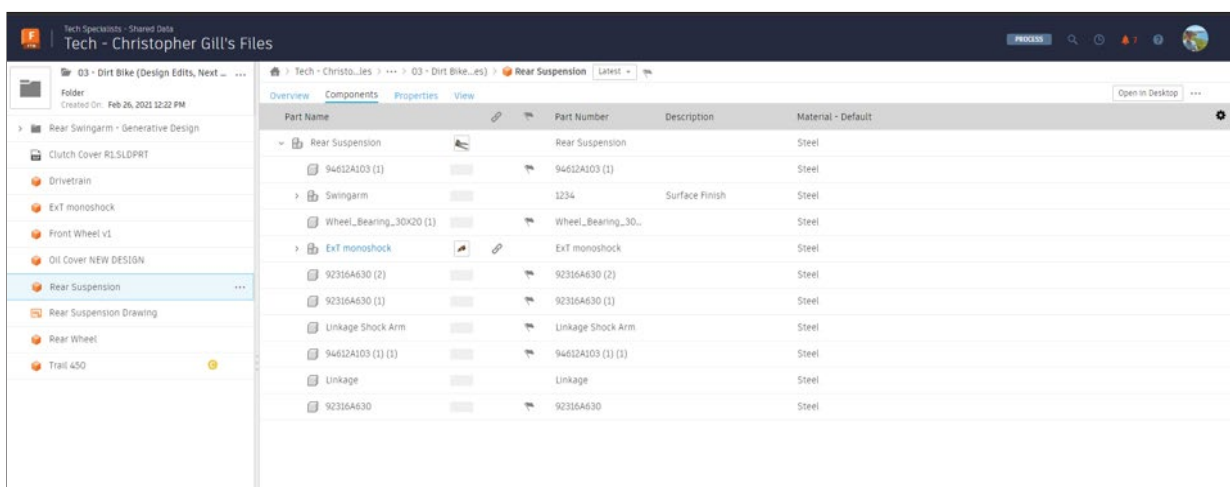
Beginner Level:

Now that we've got ourselves settled into Autodesk Fusion, let's start running through the main tips. These next few are aimed at the beginner level of Autodesk Fusion user.

## 1. Start with a save to invoke autosaving.

When you're in Autodesk Fusion and starting a new design, always begin with saving the file. This will invoke the autosaving function which periodically saves your file (~every 5 minutes) so that in the event of a crash, you won't have to revert back to the last version you created. Post a crash and upon loading up Autodesk Fusion again, you'll be presented with your autosaved files. This feature has personally saved me countless of hours of rework over the years.

2. The direction you drag you mouse changes what you select.

Clicking and holding your mouse will open up a selection box. However the direction you drag this box in changes what you select (and the colour of the box too).

Dragging this from left to right (→) will open an orange box with a purple outline, which will select anything that is fully inside this boundary of the box.

Dragging it from right to left (←) will open a yellow box with a dashed blue outline, which will select anything that is touching or fully inside the box. Use these box selection options to control what you highlight on screen.

Left to right selection

Right to left selection
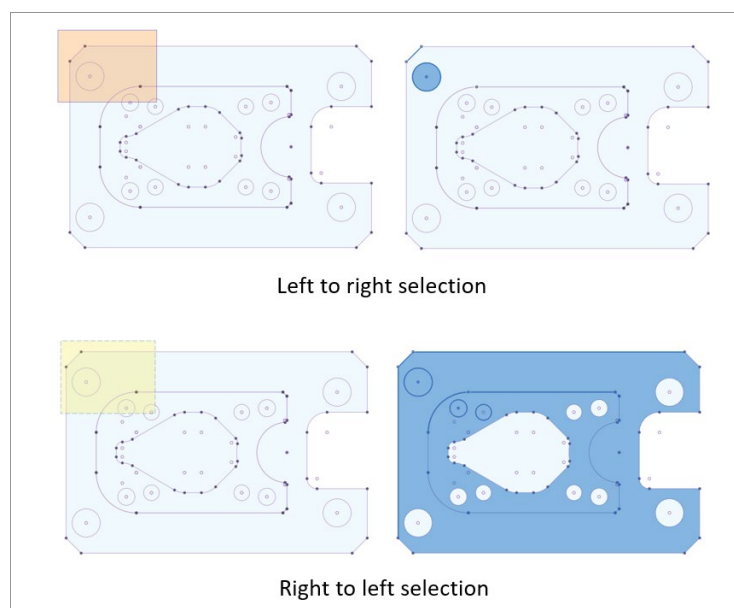
3. Design once, pattern many times.

The easiest way to save time is to use the pattern command wherever you can. If you have similar features, bodies or components in your design, then you should always be thinking how you can pattern (or mirror) this. Not only will it save you time but it makes for a 'lighter' file, which will keep Autodesk Fusion running faster than if you model the features separately.
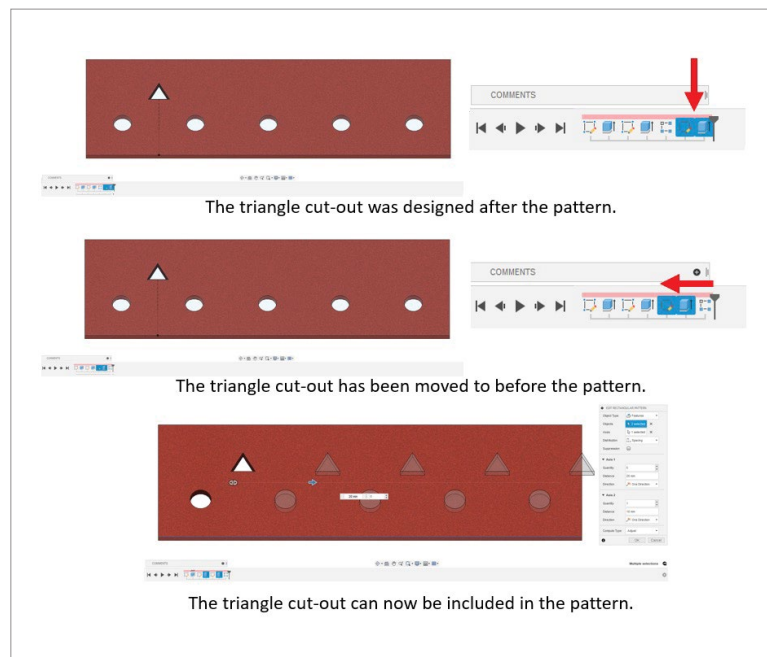
Note that you can also pattern in the sketch environment, but more in the sketch environment will make the file 'heavier' and therefore Autodesk Fusion will run slower. Th best practice is to do as much as you can with bodies and features rather than sketches.
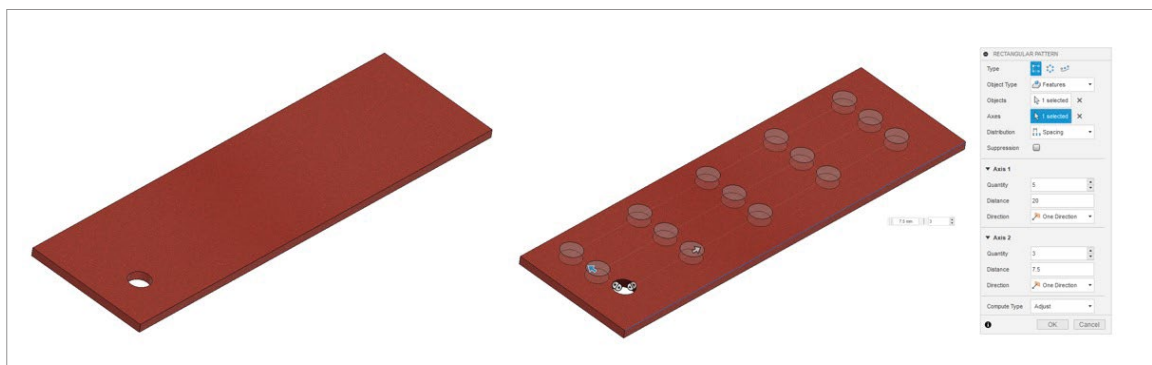
Here's a video from our team that delves deeper in the pattern capabilities of Autodesk Fusion:
https://www.youtube.com/watch?v=GppbvHEZFYc

4. Make use of the timeline's rearranging functions.

If you haven't already, turn on the timeline function which appears at the bottom of the screen. Not only is this a good way of scrolling back to various points in your design history, but you can also drag and drop the order in which features were made. This becomes particularly handy if you want to add a feature to a pattern/mirror you did earlier. Note however that it won't let you completely reorder the history of features being made. E.g., you cannot move an extrude to before the sketch that it references from.

The triangle cut-out was designed after the pattern.

The triangle cut-out has been moved to before the pattern.

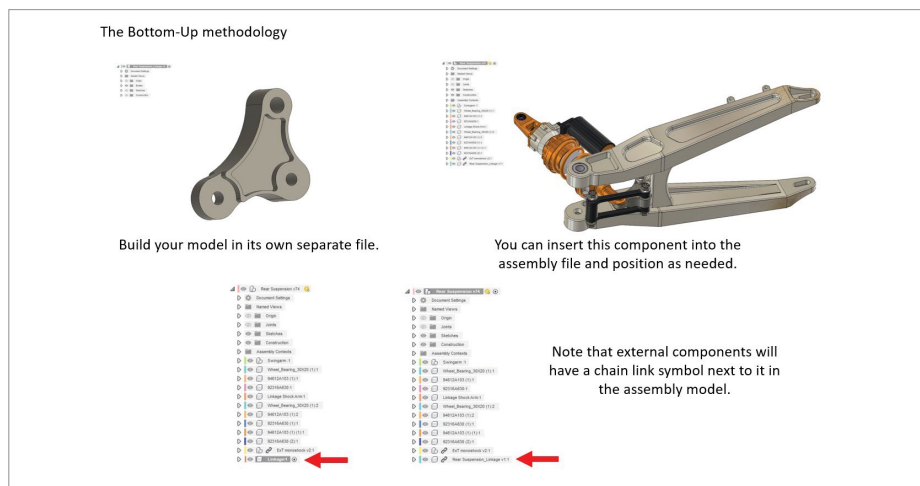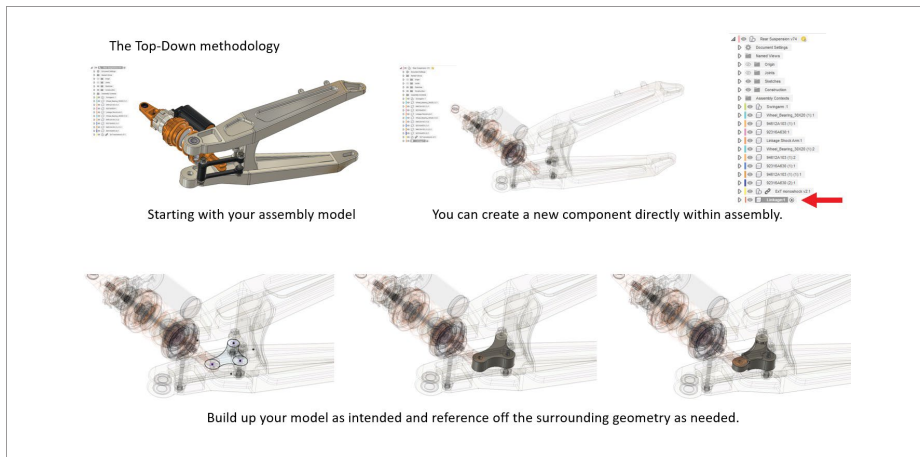The triangle cut-out can now be included in the pattern.

5. Know the difference between 'Top-Down' and 'Bottom-Up' modelling.

There are 2 different design methodologies to how you can design a part in CAD and Autodesk Fusion allows you to do a hybrid of both. They are the Top-Down and Bottom-Up design methodologies.

Bottom-Up is the more traditional way of designing where you would design every component independently in its own separate file. Then to make an assembly, you would create a separate assembly file and insert all the individual components into that. In Autodesk Fusion we call these external components.

The Top-Down methodology is where you would design a new component within the same file as the assembly. In Autodesk Fusion we call these internal components.

The Top-Down methodology

Starting with your assembly model

You can create a new component directly within assembly.

Build up your model as intended and reference off the surrounding geometry as needed.

The Bottom-Up methodology

Build your model in its own separate file.

You can insert this component into the assembly file and position as needed.

Note that external components will have a chain link symbol next to it in the assembly model.

Both have their merits. The Bottom-Up methodology allows you to use that part in other files and allows you to manage that part independently if your using a PDM/PLM system. The Top-Down methodology allows for quicker designing. You can see the new part into the context of the assembly and ensure that this part will have the right dimensions and fit into the right space. This method is particularly useful in the initial concepting phase. You can of course save/derive this file out into its own independent file if you need to use it in other assemblies.

Both of these are valid design methodologies so do whatever works best with your workflows.

Check out this video to find out more on the different design methodologies:
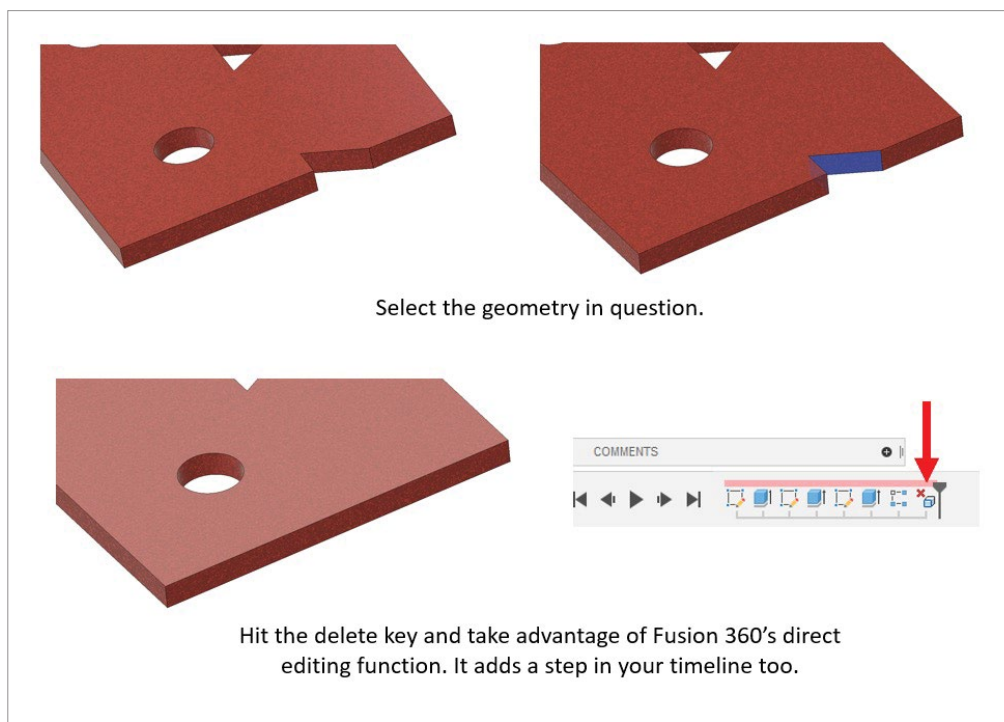https://www.youtube.com/watch?v=f1Nx0kd-cJE

Intermediate Level:

Now once you're a bit more familiar with the basics of modelling you'll be looking for more ways to improve your skillset. Let's look at some tips that would be useful for users who want to take their skill up a level.

### 6. Use direct editing to quickly clean up designs.

If you have any obscure faces, edges or other 'artifacts' that appear in your design, then you can get rid of them using the direct editing functionality. Simply select the object on your model and hit delete. Autodesk Fusion will then 'rebuild' the geometry around it for you. Not only will this be quicker than trying manually track down and edit whatever sketch profile or extrusion created this artifact in the first place but it also becomes invaluable when dealing with imported STL (or other CAD files) that have no design history. You can do much more such as move, rotate and grow the objects too.

Select the geometry in question.

Hit the delete key and take advantage of Fusion 360's direct editing function. It adds a step in your timeline too.

Check out this quick tip for more details on what can be done with the direct editing capabilities:
https://www.youtube.com/watch?v=meq-fNmh4iI.

7. Use the information button to open the Autodesk Fusion documentation.

When you come across a function that looks useful but you're not entirely sure how to use it. Click the 'i' symbol located at the bottom of each feature menu (and then select more information) to open up the Autodesk Fusion documentation on our webpage. Here you'll get an in-depth guide on how to use each function of the feature in question. There's even some handy videos in there too. It's a great way to learn how to do more inside Autodesk Fusion.
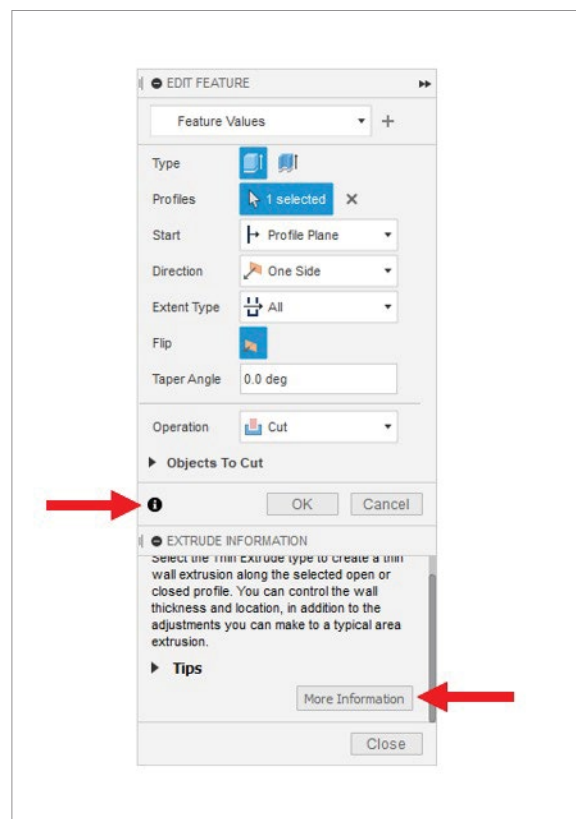
## 8. How to change the selection choice when dimensioning an arc.

When in the sketch environment, you may find yourself needing to put a dimension on an arc. Sometimes you will want to dimension it from the centre of the arc and other times you might want to go off the outer edge. By default Autodesk Fusion will choose the centre of the arc, but as you're dimensioning, if you right click on the arc, you'll bring up the option to change it to reference off the tangent of the arc. Useful if you wish to specify total length of a slot for example.
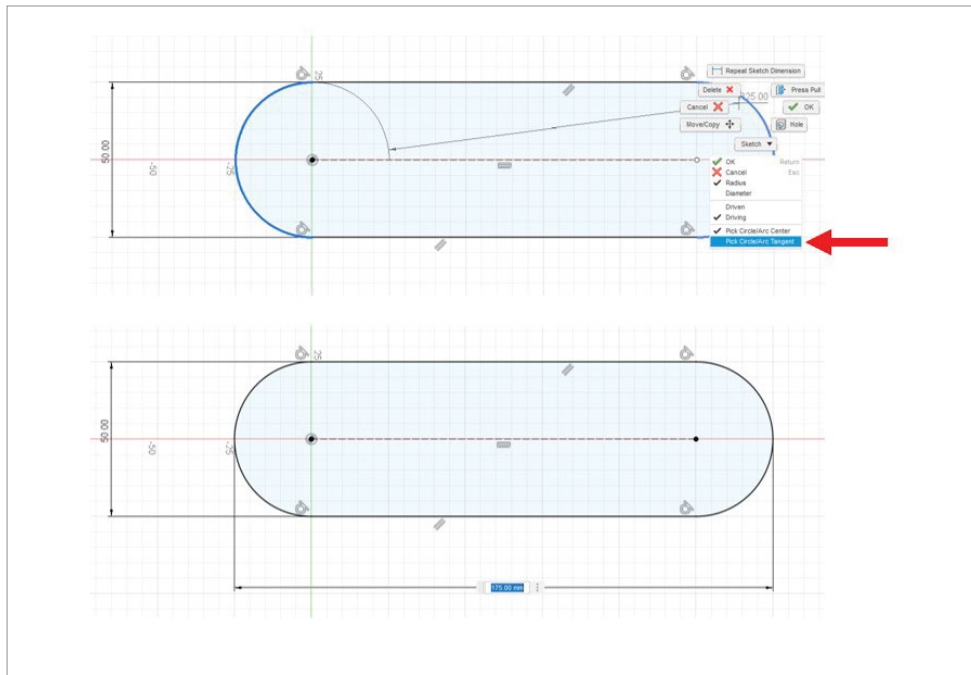
Check out this video for a more detailed guide on how to sketch to the tangent side of an arc:
https://www.youtube.com/watch?v=2q4sPvSULSc.

## 9. Use 'Show Dimension' to change sizes in 3D.

Sometimes you might want to make dimension changes whilst viewing your model in 3D. The benefit is that this will update the model instantly each time rather than having you jump back into the sketch. To do this, right click on the sketch in question and choosing 'Show Dimension'. This will show the dimensions out of the sketch environment for you to edit as and when you want.
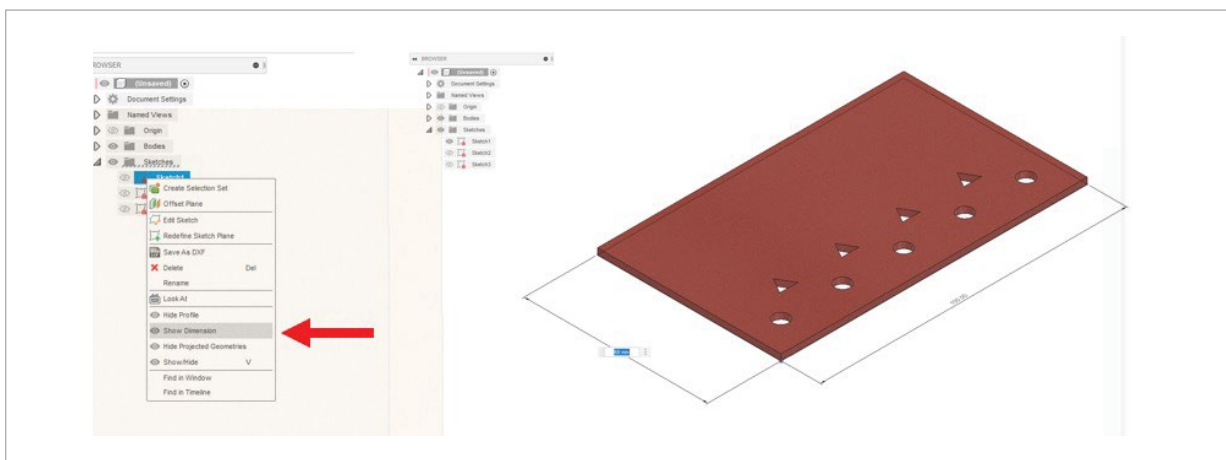
Check out this video for a more detailed guide on how to show live sketch dimensions:
https://www.youtube.com/watch?v=M23ndogd-5I.

10. Use McMaster-Carr components for fasteners.

Life's too short to model nuts and bolts. Instead upload the fastener you need from the 'Insert McMaster-Carr Component' function. Here you get access to McMaster-Carr's comprehensive catalogue of 3D modelled fasteners. You can easily throw them into your design and save hours of modelling time.
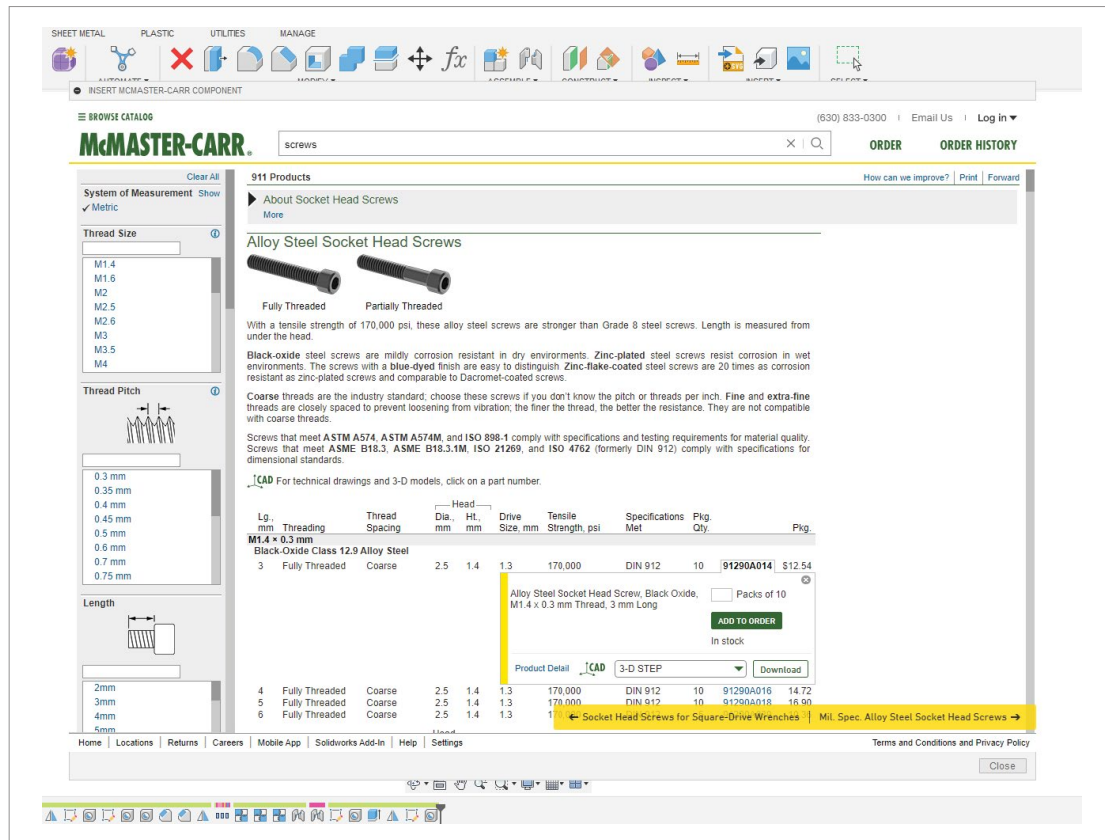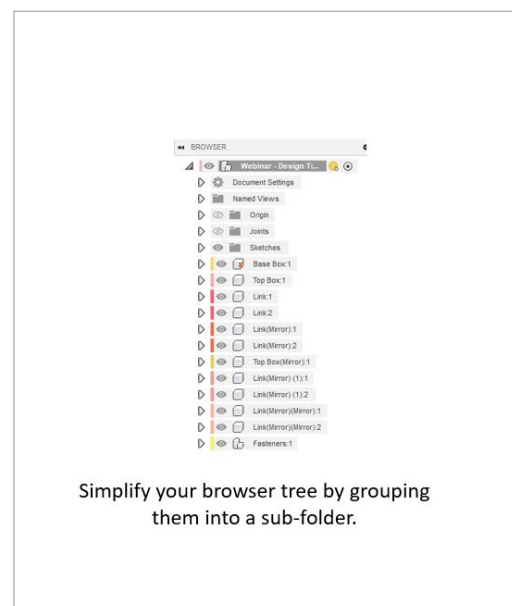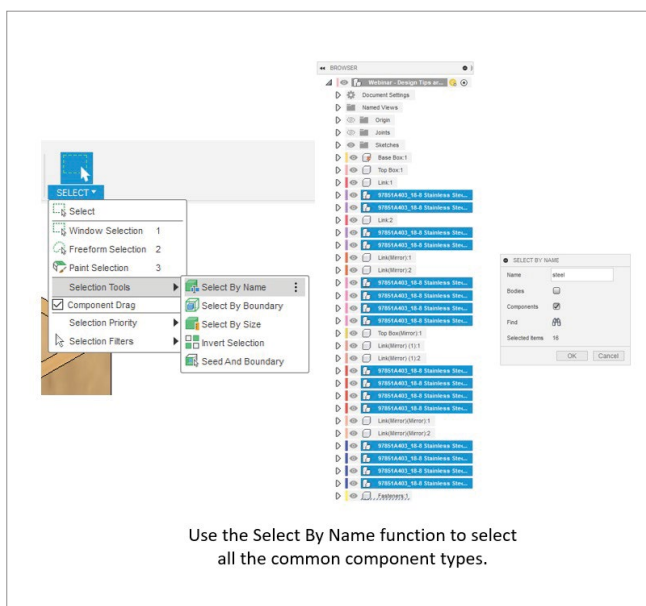

Check out this webinar on getting the most out of Mcmaster-Carr components:
https://www.youtube.com/watch?v=l4oak9He6Z8.

11. Put similar components into a sub folder (bonus: use the Select by Name function).

You may find yourself in a situation with 100s of components in your build tree (especially if you upload loads of fasteners from McMaster-Carr as mentioned above). This can clutter your design tree and make it hard to find the components you need. Where possible it's good practice to group similar components into a separate sub-component folder. You can simply create a new folder and drag and drop these components into. Additionally, you can use the 'Select By Name' function under the 'Select' options to select all of the components in one go that contain a common word.

Use the Select By Name function to select all the common component types.

Simplify your browser tree by grouping them into a sub-folder.

This video on bodies vs components will help to explain more on this subject:
https://www.youtube.com/watch?v=lczBmirc5Kk.

Advanced:

The final few tips will help those who are more advanced in CAD and who really want to maximise their modelling capabilities inside Autodesk Fusion.
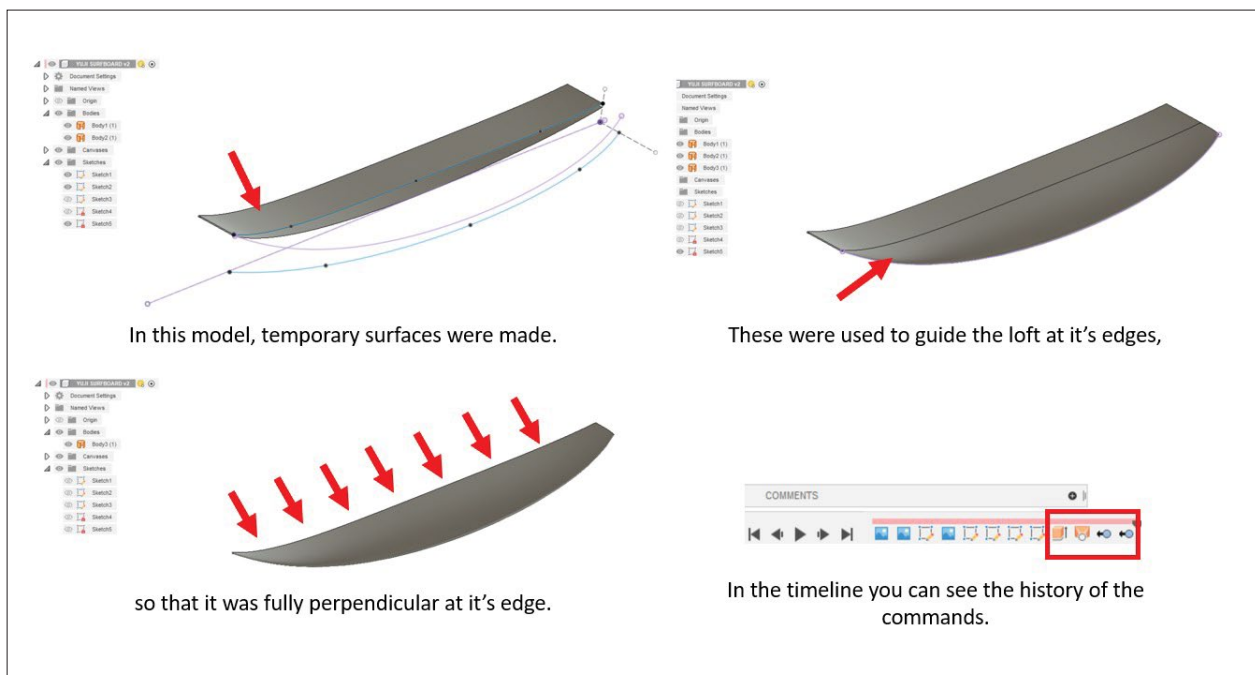
12. Know the difference between delete and remove.

On the surface, both of these commands will do the same thing; they'll both get rid of a body or component from your model. However, they perform different roles.

The Delete command will get rid of all traces of that object and anything that was made using a particular function will also be deleted. It also does not leave a record of this in your timeline.
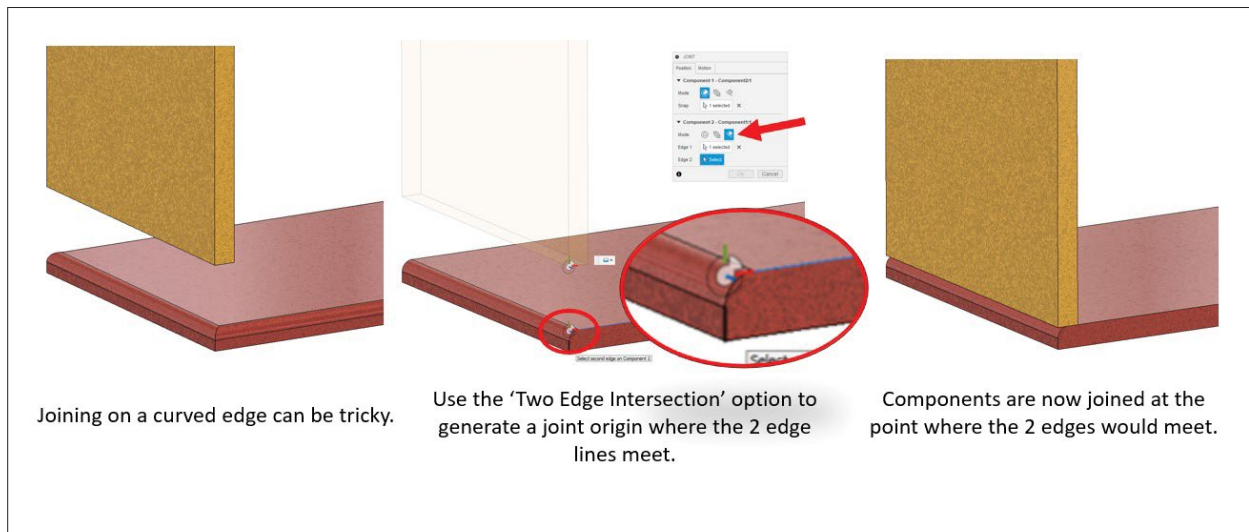
The Remove command can be considered as a 'soft delete'. It will remove a body or component but will create a feature in your timeline that you can roll back to before. It also keeps the build history of the body or component in question.

When would you use Remove? In some instances you might was to create some temporary reference geometry which aids in the design of something else. Deleting this will remove those references and therefore cause issues for the parts referencing off them, but Removing it instead will keep all the references in place but get rid of it from the screen.

In this model, temporary surfaces were made.

These were used to guide the loft at it's edges,

so that it was fully perpendicular at it's edge.

In the timeline you can see the history of the commands.

13. How to join to a curved edge.

A common question that is asked how is to place a joint when the edge is curved or filleted. The solution is simple (when you know where to look). Within the Joint command, you'll see the mode option for 'Two Edge Intersection', this is what you use and it will place a joint origin where 2 lines intersect. That joint origin can then be used to join with another component.

Joining on a curved edge can be tricky.

Use the 'Two Edge Intersection' option to generate a joint origin where the 2 edge lines meet.

Components are now joined at the point where the 2 edges would meet.

14. Animate joints in motion study space.

Once you have a series of joints, you may wondering how you can animate these to show the combined motion. Most users jump to the animation workspace but in here you would have to redefine the joints and connections again manually which would be slow and time-consuming. Instead our recommendation is to use the 'Motion Study' function found within Assemble tab. This uses your existing defined joints and allows you to control and manipulate the motion over a period of time. You can even render this to showcase as a video.

Use Motion to control when and by how much the joint moves.
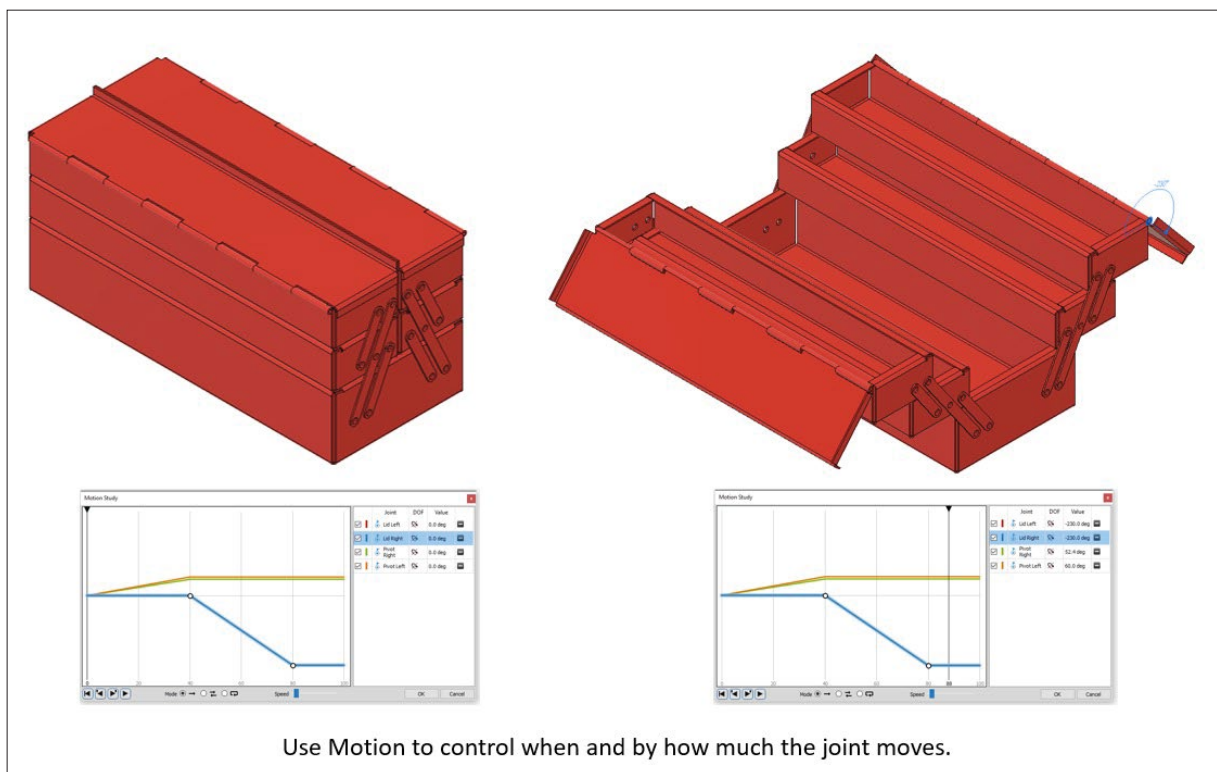
Check our this webinar to get a deep dive on joints:
https://www.youtube.com/watch?v=cJRnNUBIHBE.

15. Use the parameter table to make quickly editable designs.

One of the best features for creating design variations and making quick design changes is the parameter table. Here you can setup custom parameters (dimensions etc.) with their own name which you can then use throughout your model. Common examples would be distances referencing the 'Height', 'Width' and 'Length' of your part. Then if you wish to make quick changes, simply change the values from the parameter table, hit enter and watch your model rebuild itself to your new specifications in seconds. Use this in any instance where you have the same needing to be modelled in multiple sizes.

Note as well you can get really deep with the types of parameters you use and you're not just limited to distances or single values. Feel free to explore using equations and parameters relating to 'Quantities of', 'Angles' and much more.

Add custom parameters into the parameter table, then use them throughout your modelling features.

Adjust these parameter values as needed and see your model update.
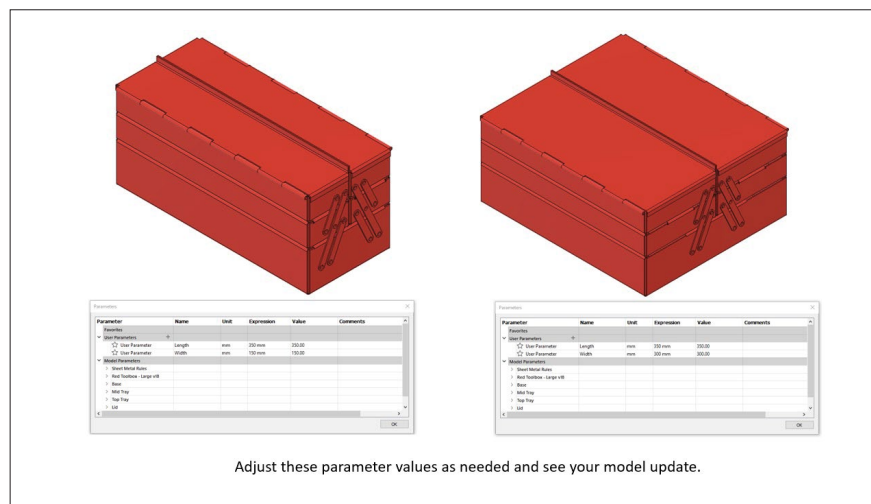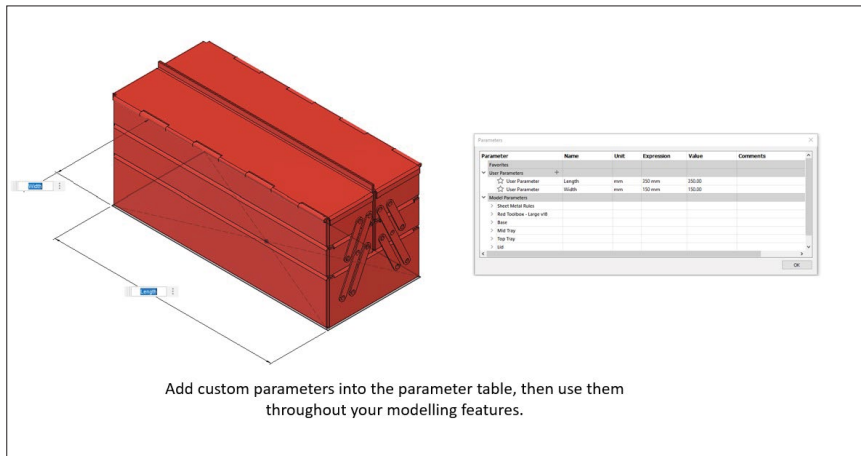
Check out this webinar for and in depth explanation into using the parameter table:
https://www.youtube.com/watch?v=TGtBcI-nC34.

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We hope this guide has been useful and you've been able to take away something that can help you in your CAD career. Autodesk Fusion is a constantly evolving product and new capabilities are always being added, therefore the instructions in this guide may not always be 100% accurate. Check out website for the latest information.

For purchasing

Phone: 1800-314-451
Email: fusion360anz@autodesk.com

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