Additive manufacturing with Autodesk Fusion 360

This document accompanies the *Design considerations for Additive manufacturing* lesson in the *Additive manufacturing with Autodesk® Fusion 360®* course.

It highlights some of the common pitfalls when designing for additive manufacturing and describes how to avoid them using Fusion 360.

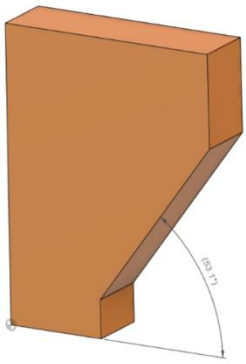
Considerations when designing for additive manufacturing

Additive manufacturing is a very powerful technology that enables you to relatively quickly make your designs a reality. However, like all manufacturing methods, there are manufacturing constraints of the technology to consider when designing your parts. Just because you can design something, does not necessarily mean you can make it.

These constraints you encounter will depend on the additive technology type and specific machine that you are using, so it is always best to check the recommendations from the printer manufacturers.

Overhangs

One of the main considerations for an additive build is which faces of the model will need to be supported. For most additive technologies, the material will not need to be supported if it is built on top of the previous layer. This means that if building directly upwards, you will not usually need supports. Building out to a specific angle, called the overhang angle, also usually removes the need for supports. If the overhang angle required to build your part is too large (usually more than 45 degrees), then the material will need to be supported.

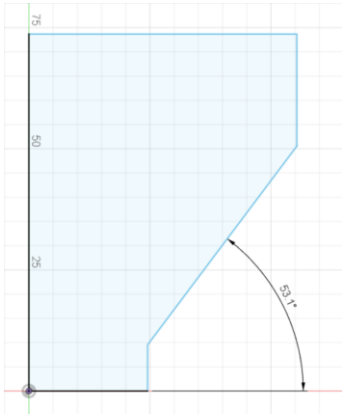


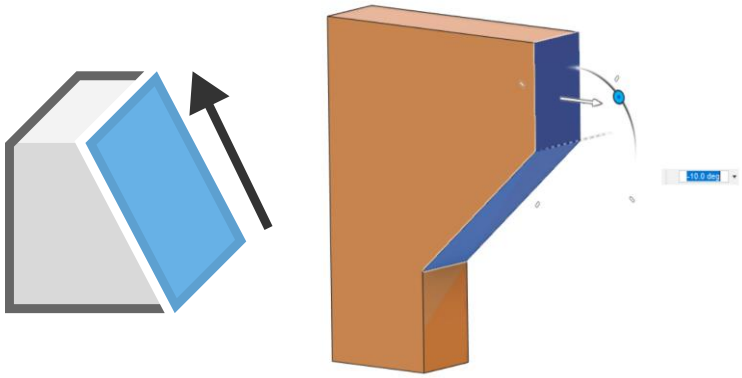
To reduce the amount of support structures you are using, perhaps to keep the build time or material use to a minimum, or to reduce the amount of post-processing time involved with removing these supports, it could be necessary to make sure you are not going over the critical overhang angle.

When designing using Fusion 360, you can ensure that you don't exceed the overhang angle by:

- Defining any sketches that will create an angled face to be less than the critical overhang angle, or
- Avoiding draft angles that are larger than the overhang angle when using the Draft tool.

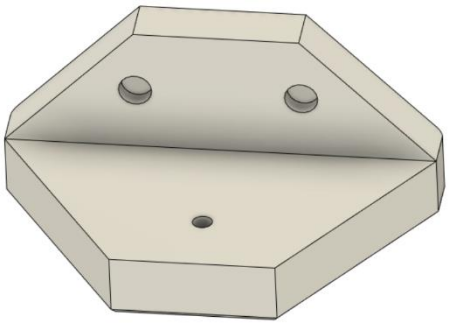
You can also use the Draft tool to edit the model to decrease the angle of existing faces if necessary. This also works for any imported models.

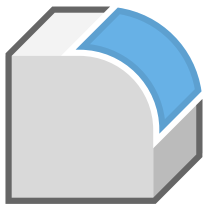


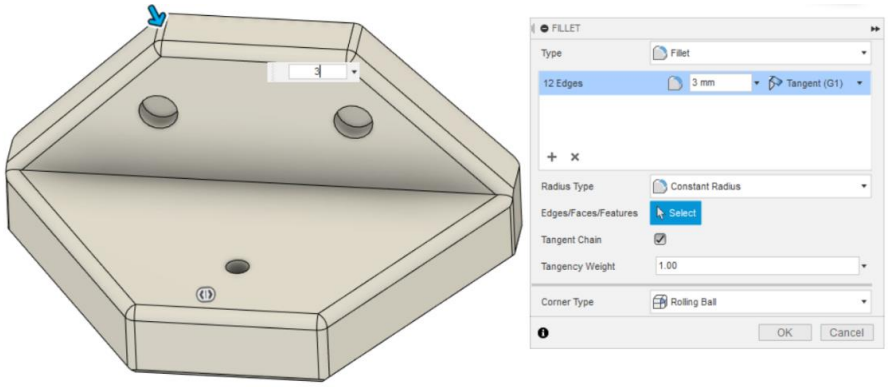


Features

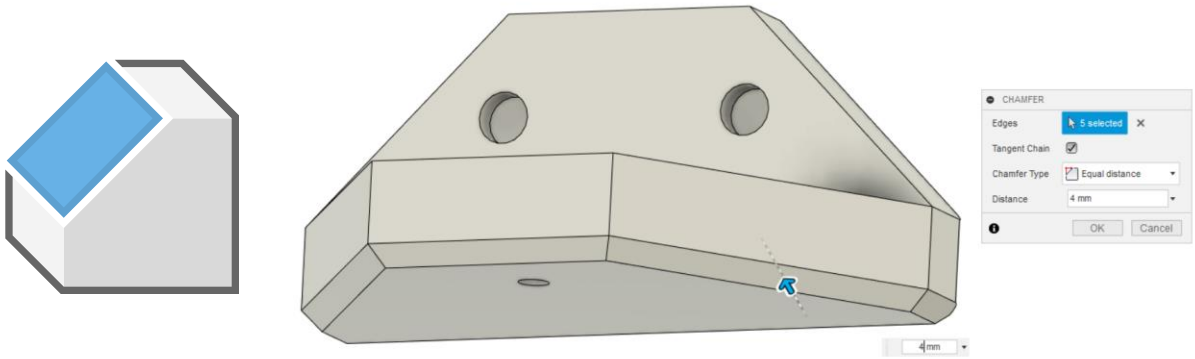
Some types of features will print better than others. Sharp edges or corners should be avoided as these are more susceptible to warping. Instead fillets should be used where possible. When using Fusion 360 you can use the Fillet tool to easily replace any sharp edges with a rounded fillet.







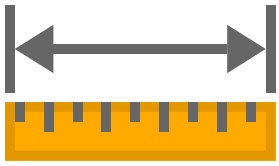
The exception to this rule is if the fillet is curving onto the build plate then instead a chamfer should be used. This is because a fillet is more prone to warping and peeling away from the build plate. You can use the Chamfer tool in Fusion 360 to easily add these into your model instead. Where necessary, it is possible to delete any fillets first, by either selecting the faces from the model or the Fillet item from the Timeline.

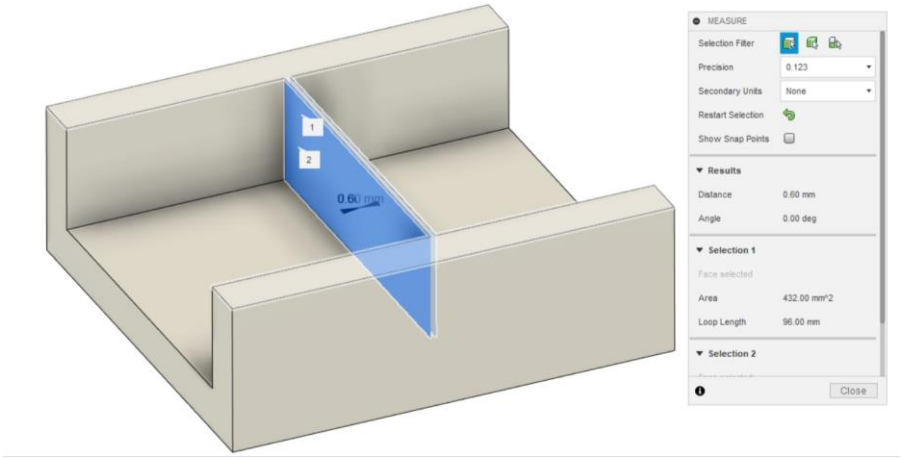


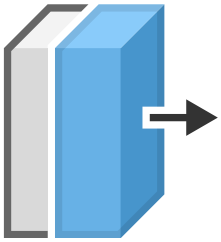
Minimum detail size

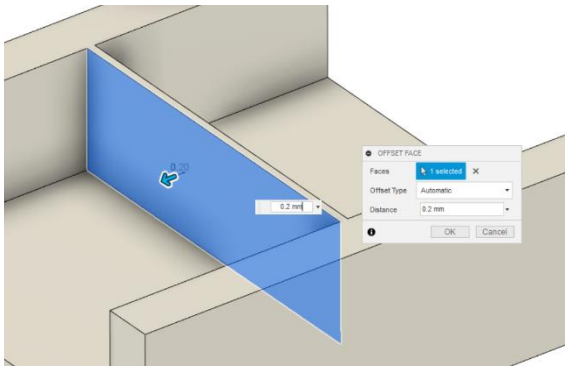
Although very fine detail on a model might be a nice addition, it may not always be possible to achieve using additive manufacturing. Depending on the additive technology type, the minimum size of detail that can be reached could be down to the nozzle size of the printer, the layer height, or for example the laser beam width. Therefore, it is important to know the minimum detail size for the machine you plan to print on. When you know this value, you will need to consider it when designing or editing a model for additive manufacturing.

For example, there will be a minimum wall thickness achievable, which will be equal to this minimum detail size. During the design stage in Fusion 360, it could be important to make sure no walls are created below this size, by controlling the dimensions of any sketch items. Alternatively, a model may need to be checked for this which can be done using the Measure tool.



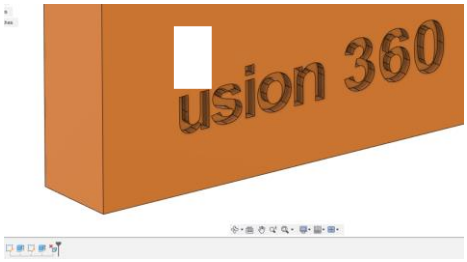






If it is found the wall thickness is too small, using the Press Pull tool is a very effective way of quickly editing surfaces to add (in this case) or remove material.

Another example of when it is important to consider the minimum detail size is when using text, small slots or embossed features, so you should consider the minimum detail size value when using the sketching tools in Fusion 360 for these features. Alternatively, the Measure tool is also for these types of feature, and the Press Pull tool may be useful to edit them if needed. It is also worth remembering that any detail will add time to the build, so it could be best to simply delete these features if they are very fine. Using Fusion 360 it is very easy to select faces and delete, and the material around the faces will be automatically filled in. If required, you can later revert any deletions using the Timeline:

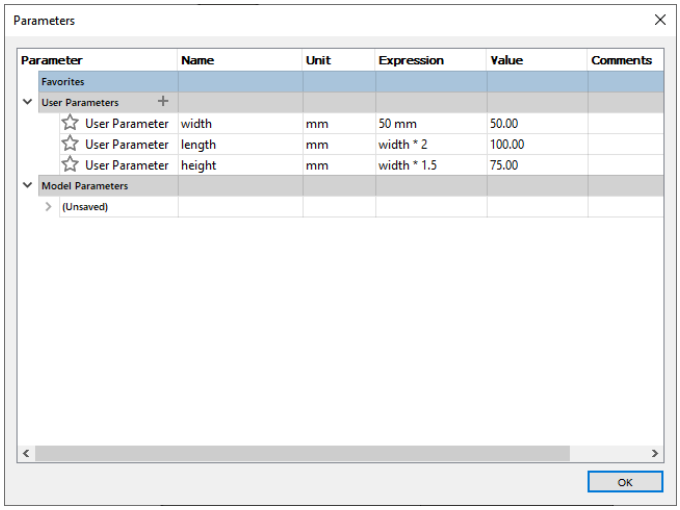


Prototyping

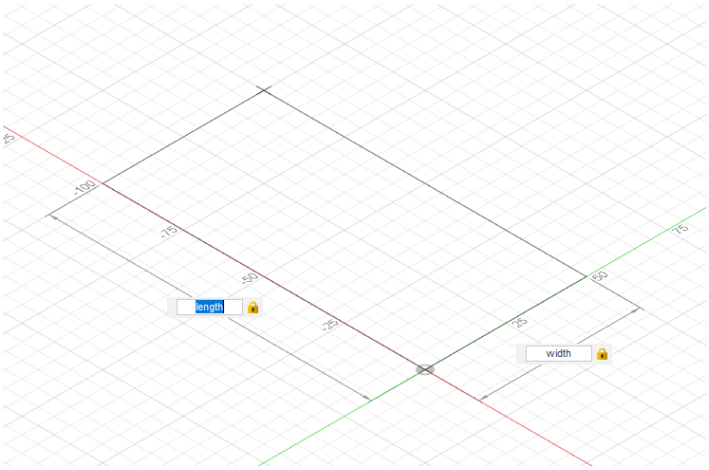
One of the biggest advantages and more widely used applications of additive manufacturing is the ability to rapidly create prototypes of a design, allowing you to conceptualize and make quick edits if required before committing to mass production or to a more complex and perhaps more expensive form of manufacturing. However, if design adjustments are needed you don't want to have to start from the beginning of the process again.

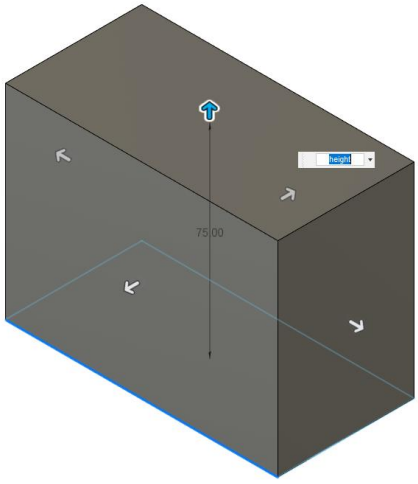
Using Fusion 360, you can quickly edit a CAD model using the Timeline. The Timeline enables you to edit any feature or sketch that has been created during the design process, and your edits will propagate to the rest of the model, updating the surrounding geometry where appropriate.

To have more control over this, you can also use User Parameters. These allow you to define dimensions and other parameters, by assigning them a parameter name and value. You can then reference these parameters when defining other dimensions. For example, if we look at creating a simple rectangular box we can define the width as a user parameter and give it a specific value. We can then define the length as another user parameter and define this as width multiplied by 2. Similarly, we can create a third user parameter for the height and define this as width multiplied by 1.5.



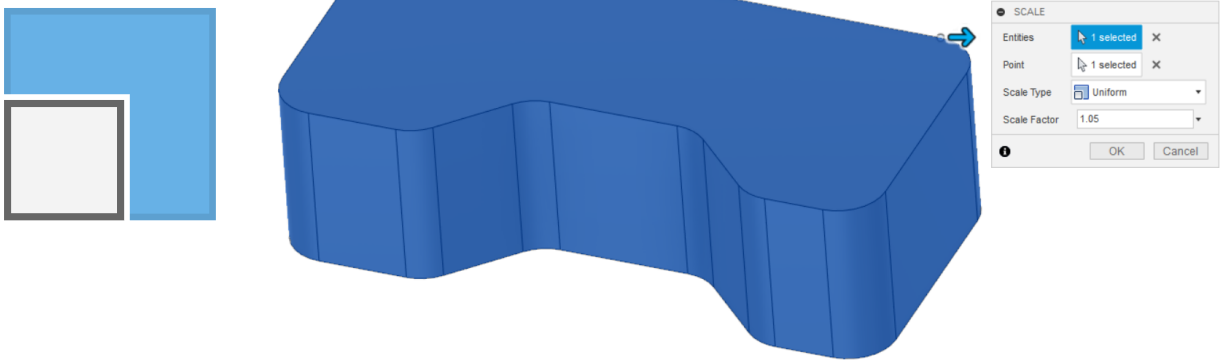
This means that if we edit the width value the length and height will automatically update, so that we don't need to go back and edit every dimension and value. Making use of these User Parameters when designing your model can be incredibly powerful, especially if carrying out an iterative prototyping process with additive manufacturing. The User Parameters can be defined at the point of creating a sketch or can be created and edited from the Change Parameters dialog.





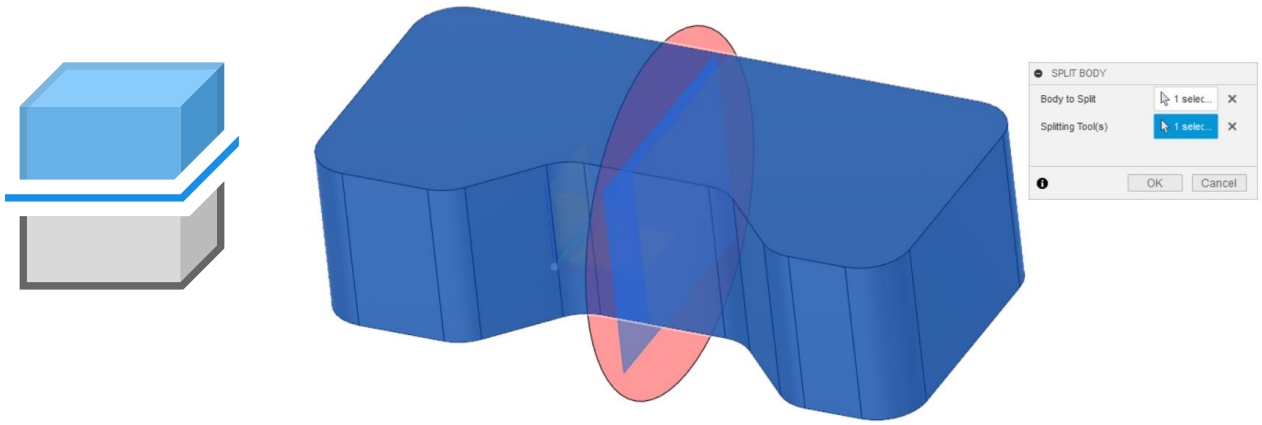
Whole-model edits

In all methods of additive manufacturing, there is some form of warping and shrinkage due to the loss of heat from the material, and some models may appear to be printed undersize due to this shrinkage. If the fit of your finished part is critical, then you may need to adjust for this shrinkage. One way to do this while using Fusion 360 is to scale your model up to account for the expected shrinking. You can do this using the Scale tool.

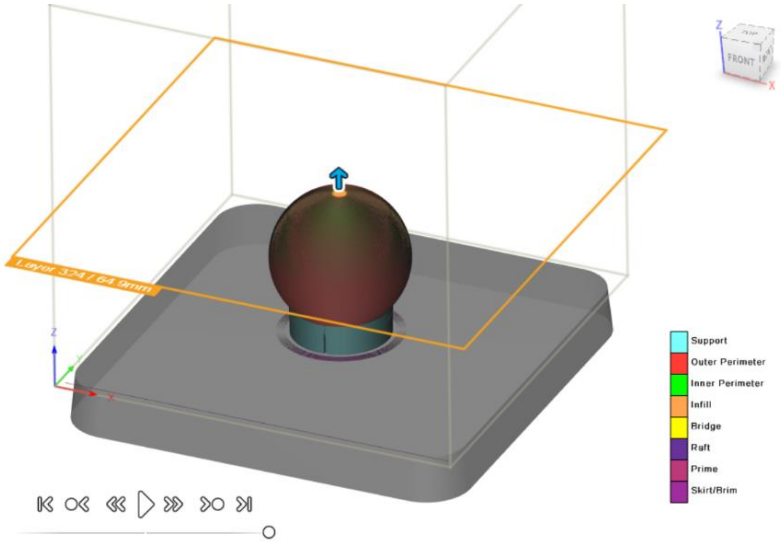


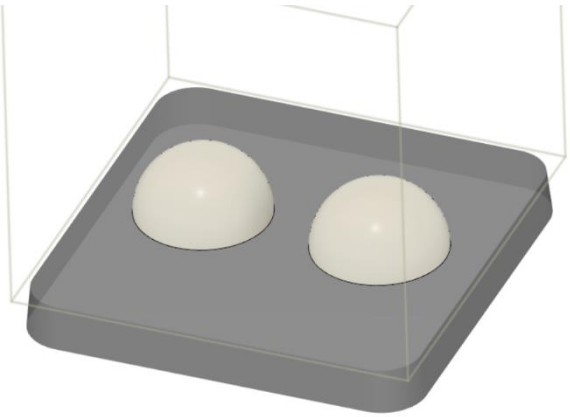
To work out the factor to scale your model by, the best thing to do is print a test part, measure some features and make note of the actual printed size versus the nominal CAD model size. You can then work out your scale factor to use for future prints while using the same print parameters.

Another factor to consider is the size of your design compared to the size of your build plate. If your model is quite large, you may not be able to print it in one go depending on the machine you are using. You may be better breaking the model down into smaller parts and printing these separately. If you need to do this, the Split Body tool can be useful, as it allows you to break a body down into separate parts.



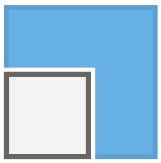
Sometimes it may also be useful to split a model into separate parts, as smaller parts can be easier to print than the full model and the smaller parts can be assembled after production. For example, printing a sphere as one part may require a lot of support structures.





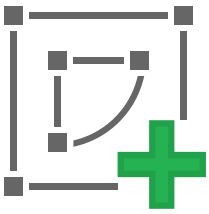
Splitting the sphere into two semi-spheres, however, would not require any support structures, and depending on the additive technology used could also lead to better surface finish quality and a lower build time.

Post-print assembly

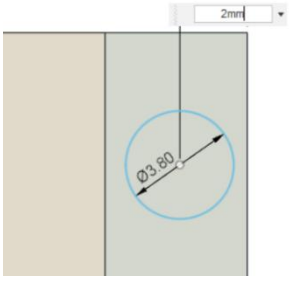


If the model being printed is to be part of an assembly, you may need to make some edits to account for this. The first edit could be to use the scaling method mentioned earlier to ensure the part will be printed to tolerance, so that it will fit with the other components of the assembly as required.

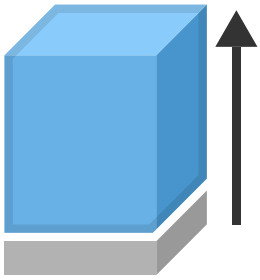
It may also be the case that the components will be held together by bolts or otherwise use a thread. For this it could be tempting to try and print the thread as part of the model, but it can work better to add these in afterwards by using metal heat-set inserts. These inserts can be heated up then pushed into the printed model, melting the material around them and then cooling again to set them in place. These will be stronger and more durable than printing the threads or tapping the printed material, which is another option.

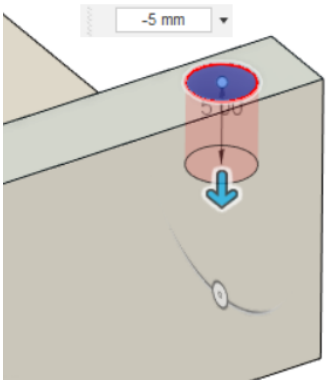


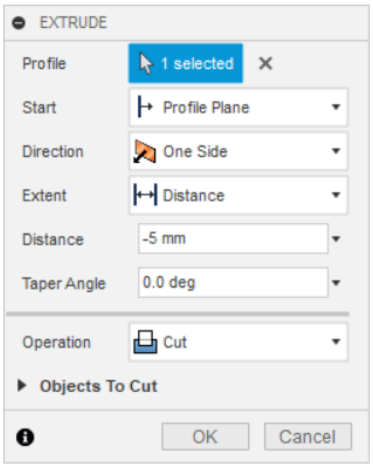
It is best to put cut-outs into your design to house and locate these inserts in. When using Fusion 360 these can easily be added in by creating a Sketch item and drawing a circle with a slightly smaller diameter than the insert to be used. It may be useful to dimension the centre of the circle from other sketch items to position it correctly.



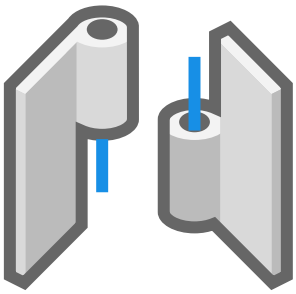
The Extrude tool can then be used to take the circular profile and use it to cut material out of the model.

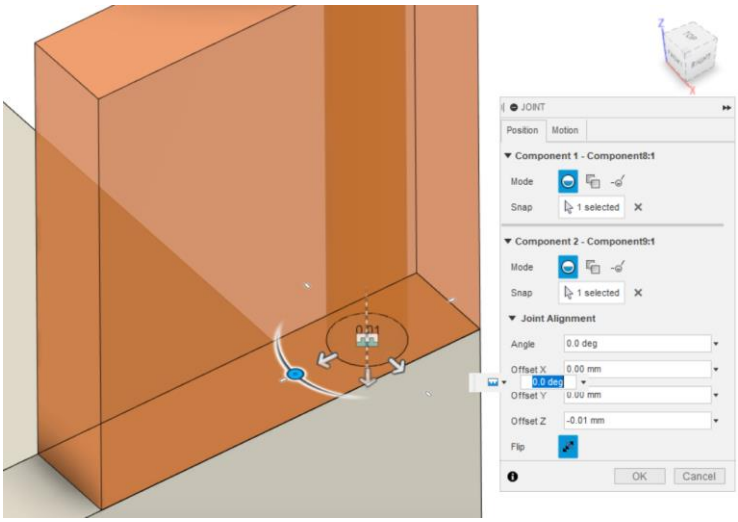






An alternative solution for joining components in an assembly is to glue them together, which can be a quick and effective method depending on the material used. To ensure the components align accurately when doing this the use of dowel pins is a good technique. Again, you will need to add cut-outs to your model in which to insert these dowel pins, and the combination of a Sketch and Extrude is a simple way to do this. Whether using dowel pins or a bolt or screw with a thread it will be important to make sure the positions of these line up between the components to be assembled. The best way to do this using Fusion 360 would be to use the Joint tool, which will allow you to position the components relative to one another in an assembly.





Once they are in position you can carry out the sketching and extrusion to add in any necessary cut-outs on multiple components at the same time, to ensure they will align as needed and the printed parts will assemble correctly.