



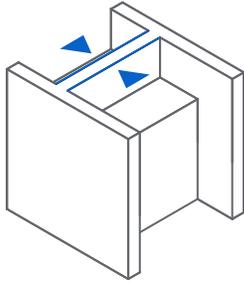
## DESIGN GUIDE

# Formlabs and Fusion 360 Design Guide

A good print starts with a well-designed model that is optimized for 3D printing. We've created this design guide to outline some of the minimum features you can expect from the Form 1+ and Form 2, and then how Fusion 360 can be used to define these features and adjust models where necessary.

Please note: The guidelines below have been developed using Formlabs Clear resin printing at 100 microns. You may experience slight variations with other Formlabs Resins and other layer heights.

# Minimum Wall Thicknesses

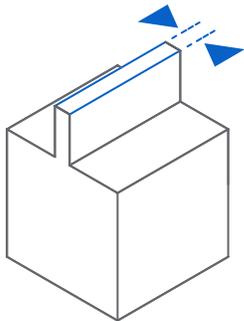


## MINIMUM SUPPORTED WALL THICKNESS

**Recommended: 0.4 mm**

A supported wall is one that is connected to other walls on two or more sides. A supported wall smaller than 0.4 mm may warp during the peel process.

**Note:** Care should be taken when washing thin walls, as they may absorb IPA and swell during the cleaning process, leading to deformation of the part. Minimizing time immersed in IPA will limit this effect.



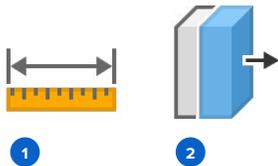
## MINIMUM UNSUPPORTED WALL THICKNESS

**Recommended: 0.6 mm**

An unsupported wall is one that is connected to other walls on fewer than two sides. An unsupported wall that is smaller than 0.6 mm may warp or detach from the model during printing.

During the design stage in Fusion 360, it is important to make sure no walls are created below these sizes, 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** 1.

If it is found a wall thickness is too small, using the **Press Pull tool** 2 is a very effective way of quickly editing surfaces to add (in this case) or remove material.



► Fig. 1 Measuring a wall thickness with the Measure tool.

► Fig. 2 Adding material to a thin walled section using the Press Pull tool.

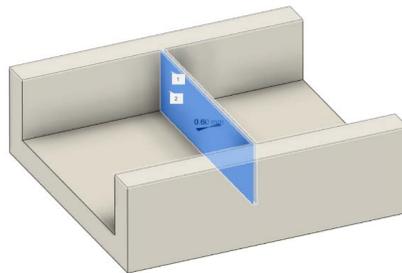


Fig. 1

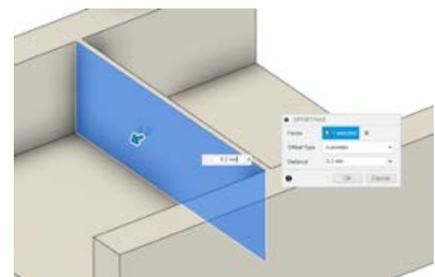
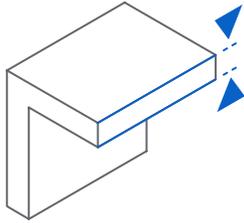


Fig. 2

# Overhangs

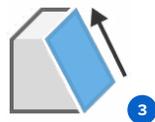
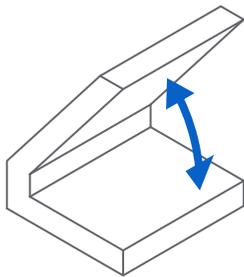


## MAXIMUM UNSUPPORTED OVERHANG LENGTH

**Recommended: 1.0 mm**

An overhang refers to a part of the model that sticks out horizontally parallel to the build platform. Printing such features without supports is discouraged, as the layers cannot maintain their structure. Horizontal overhangs will be slightly deformed beyond 1 mm and become increasingly deformed as the length of the overhang increases. You can turn on “internal supports” in PreForm to ensure your overhangs are supported.

If you want 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 might be better to simply edit the model instead so that this overhang length value is not exceeded. This is another case where the Press Pull tool in Fusion 360 could be very useful to reduce the size of the unsupported overhang.



## MINIMUM UNSUPPORTED OVERHANG ANGLE

**Recommended: 19° from level** (35 mm long × 10 mm wide × 3 mm thick)

The overhang angle refers to the angle from horizontal that the overhang sticks out. Printing at an angle less than 19° could cause the overhang to break off the model during the peel process. Rotate your part so flat surfaces can be held up by supports if they are not already self-supporting. See the “Print Flat Surfaces at an Angle” section of our Model Orientation article.

If you cannot or want to avoid rotating the part or using supports, when designing using Fusion 360 you can ensure the overhang angle is not exceeded by defining any sketches that will create an angled face to be less than the minimum unsupported overhang angle, or when using the **Draft tool** <sup>3</sup> not adding any draft angles that are less than the minimum overhang angle. You can also use the Draft tool to edit the model to increase the angle of existing faces if necessary. This also works for any imported models.

► Fig. 3 Using sketch dimensions to define overhang angles

► Fig. 4 Editing the overhang angle on a model using the Draft tool

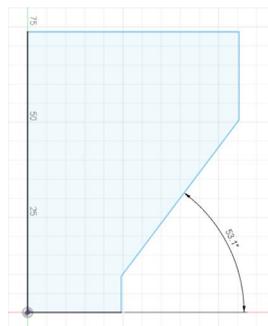


Fig. 3

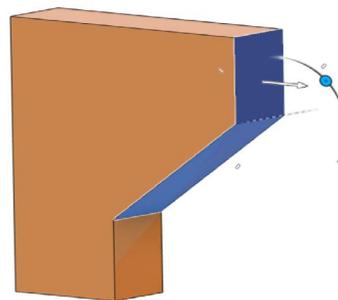
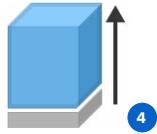
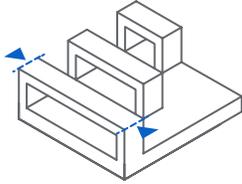


Fig. 4

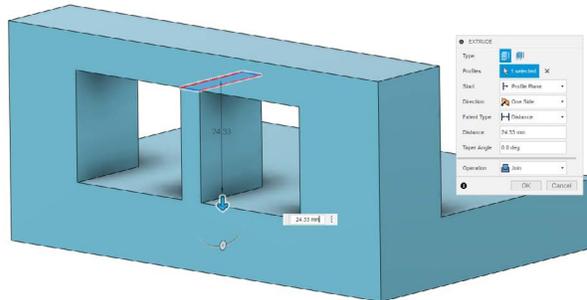


## MAXIMUM HORIZONTAL SUPPORT SPAN/BRIDGE

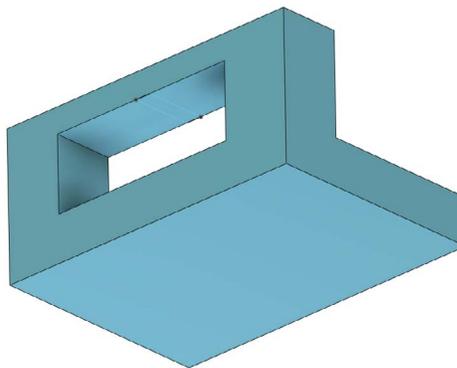
**Recommended: 21 mm** (5 mm width × 3 mm thick)

A span is the distance between two intermediate supports of a structure. While printing horizontal spans is discouraged, certain geometries print well. For a 5 mm wide and 3 mm thick beam, spans longer than 21 mm are likely to fail. Wider beams must be kept shorter to avoid breaking during the peeling process.

With Fusion 360 another good workflow for editing this type of feature is the **Extrude command** 4. If a span or bridge exceeding the maximum length cannot be avoided in the design and you would like to reduce the use of supports you could directly model small supports to break up the structure and give it support at points along the span, without needing to support the whole length. For this you could sketch rectangles on the surface then Extrude to create solid geometry and join these to the model. You will just need to bear in mind the post-processing then required to remove these areas.

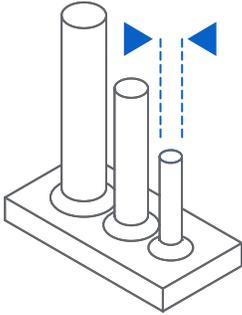


► Manually adding in support for the span/bridge using a sketch with the Extrude tool.



► Unsupported horizontal span/bridge exceeding maximum length.

# Minimum Detail Size



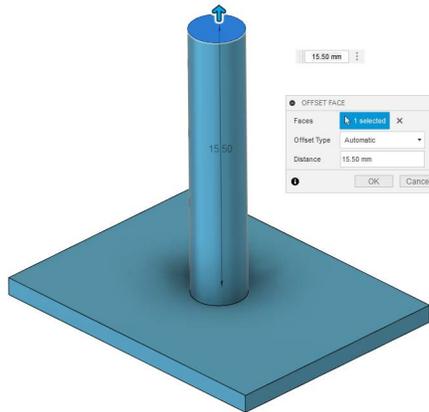
## MINIMUM VERTICAL-WIRE DIAMETER

**Recommended: 0.3 mm** (7 mm tall) **to 1.5 mm** (30 mm tall)

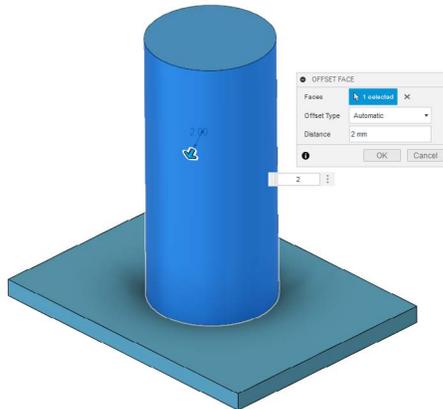
A wire is a feature whose length is greater than two times its width. The ratio is key to printing wires; at 0.3 mm thickness you can print up to 7 mm tall before you start to see waving. 1.5 mm wires can get up to 30 mm tall without defects.

**Note:** Similar to washing thin walls, extra care should be taken when washing small wires, as they are weakened by IPA and can easily be damaged. Minimizing the time the part is immersed in IPA will limit this effect.

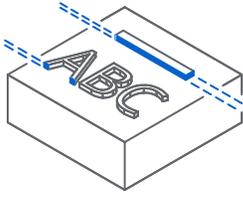
With Fusion 360 the **Measure tool** can be used to check a wire is not outside of the recommended ratio. If it is the Press Pull tool is again an easy way to edit the wire, by either reducing the height or increasing the diameter.



► Reducing the height of a vertical-wire using the Press Pull tool.



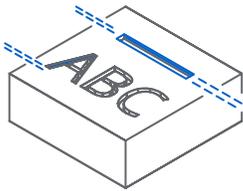
► Increasing the diameter of a vertical-wire using the Press Pull tool.



### MINIMUM EMBOSSED DETAIL

**Recommended: 0.1 mm**

Embossed details are shallow raised features on your model, such as text. Details smaller than 0.1 mm in thickness and in height may not be visible on your print.



### MINIMUM ENGRAVED DETAIL

**Recommended: 0.4 mm**

Engraved details are imprinted or recessed features on your model. Details recessed less than 0.4 mm in thickness and in height may not be visible because they will be fused with the rest of the model during the print process.

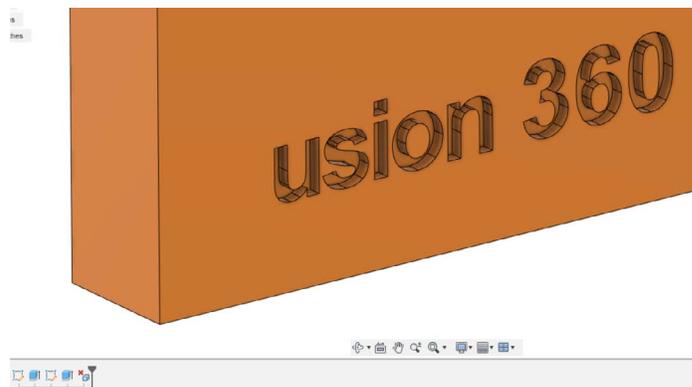


When designing with Fusion 360 either embossed or engraved detail can be added using the **Emboss**  command, by raising or recessing a sketch profile relative to faces of a solid body, including curved faces. If it is not possible if it is not possible to achieve these minimum size values it could be best to simply delete these features if they are very fine. Using Fusion 360 it is very easy to simply select faces and delete, and the material around the faces will be automatically filled in. This deletion itself can then be removed later using the Timeline to bring back the detailed surfaces.

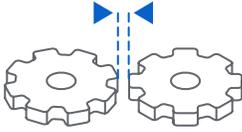
► *Selecting faces to be deleted.*



► *After deleting the faces the material is automatically filled in, and a Delete item is seen in the Timeline.*



# Minimum Clearance

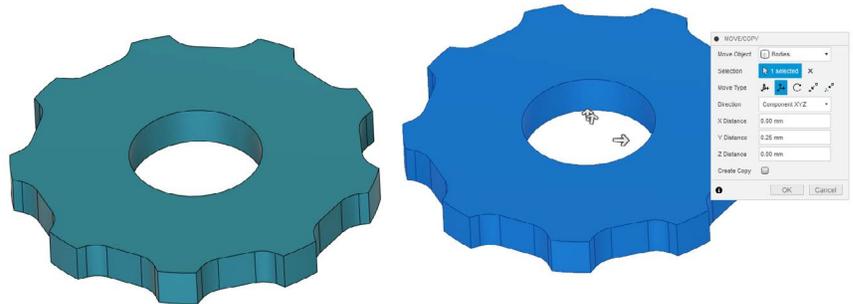


## MINIMUM CLEARANCE

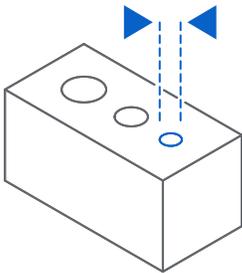
**Recommended: 0.5 mm**

Clearance is the amount of distance needed between two moving parts of a model (e.g., the distance between gears or joints). A clearance of less than 0.5 mm may cause parts to fuse.

When arranging components in Fusion 360 the Measure tool will again be useful here to check the clearance is not below 0.5mm. If a part needs more clearance the **Move** command can be used to reposition it.



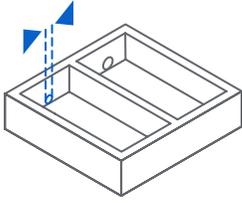
► Using the Move command to reposition models to ensure there is minimum clearance.



## MINIMUM HOLE DIAMETER

**Recommended: 0.5 mm**

Holes with a diameter less than 0.5 mm in the x, y, and z axes may close off during printing.

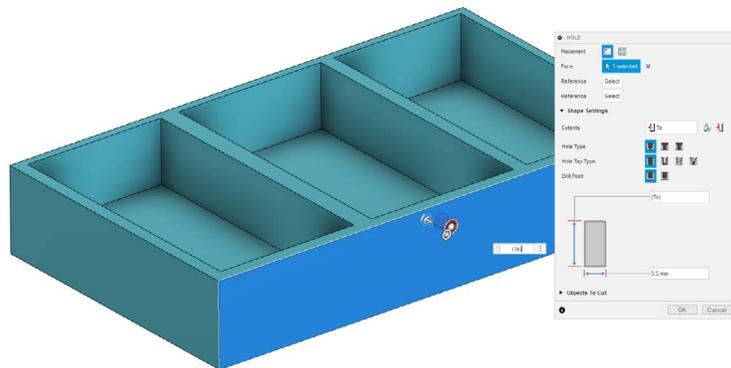


## MINIMUM DRAIN HOLE DIAMETER

### Recommended: 3.5 mm diameter

Drain holes are recommended for resin to escape in models that are a fully enclosed cavity (like a hollow sphere or hollow cylinder printed directly on the build platform). Without drain holes of at least 3.5 mm in diameter, the part may trap resin and lead to an explosion of the print. Although drain holes are recommended, they are not entirely necessary when using a Form 2 printer.

For both of these hole types the Measure and Press Pull tools workflow is again useful for checking and making any edits to a model in Fusion 360. Alternatively these diameters can be set during the modeling stages, either when dimensioning circle sketches and using Extrude to create holes with these profiles, or by using the Hole tool and dimensioning these features to be above these minimum sizes depending on their purpose.



► Using the Hole tool to add a drain hole into the design.

## 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.

## 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 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.

By using Fusion 360 you have the ability to quickly make edits to a CAD model. The **Timeline** enables you to be able to edit to any feature or sketch that has been created during the design process, and these edits will then be carried through downstream to the rest of the model, updating the surrounding geometry where appropriate.

To have even greater control over this you can utilize **User Parameters**. These will 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 this 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 will be some form of warping and shrinkage due to the loss of heat from the material. For many models they 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. From this 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 this needs to be done, **Split Body** can be a useful tool in Fusion 360. This will allow you to break a body down into separate parts.

This is also a useful method for splitting a model up into separate parts in order to make them easier to print than the original full model would be. These parts can then be assembled after production. For example, if trying to print a sphere this may require high support structure use.

If you instead split this into two semi-spheres these would not require any support structures, and depending on the additive technology used could lead to better surface finish quality and a lower build time.

## Post-Print Assembly

If the model being produced is to be part of an assembly, there may be some edits to make to the design to account for this. The first 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 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 in reality 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 can simply be 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.