



---

# MELBOURNE SCHOOL OF ENGINEERING

---

---

## 3D Printing Tips and Tricks

---

Vincent Khau  
Eleanor Lourey  
Stefan Napau

Version 1 - April 11, 2017

# Contents

<b>1</b>	<b>Why Use 3D Printing?</b>	<b>1</b>
<b>2</b>	<b>General Computer Aided Design (CAD) Advice</b>	<b>2</b>
2.1	Common CAD Pitfalls . . . . .	2
2.2	Model Resolution . . . . .	2
<b>3</b>	<b>3D Printing Tips and Tricks</b>	<b>4</b>
3.1	Model Orientation . . . . .	4
3.2	Support Material and Overhang . . . . .	5
3.3	Infill . . . . .	5
3.4	Layer Height . . . . .	6
3.5	Shells . . . . .	7
<b>4</b>	<b>Troubleshooting with 3D Printing</b>	<b>8</b>
4.1	Warping in 3D Printing . . . . .	8
<b>5</b>	<b>Questions/Feedback/Contact</b>	<b>8</b>

## 1 Why Use 3D Printing?

3D printing is an emerging technology that is being increasingly employed as a rapid prototyping technique in many companies around the world - in some cases, it is even the technology used to produce the final product (for example, 3D printed jet engine turbines). Despite being popularised in the early to mid 2000s, 3D printing has actually been around since 1984 and has since allowed engineers to create designs that would have otherwise been very difficult to manufacture by traditional techniques. It is a comparatively low-cost manufacturing technique for parts that need to be printed on demand for short run productions and thus increases the accessibility of rapid prototyping.

## 2 General Computer Aided Design (CAD) Advice

To get started in Computer Aided Design software, there are free CAD workshops organised by Research Platforms Services at the University of Melbourne - <http://melbourne.resbaz.edu.au/>

The following section provides some pointers on CAD files and some common errors that may result in a file that cannot be processed. Solutions to these errors will not be documented as each CAD file will require different tools and processes to fix and hence it is up to the user to submit a suitable **.print** file to the 3D Innovation Centre.

### 2.1 Common CAD Pitfalls

To 3D print a CAD file, it needs to be converted into an **.stl** (stereolithography/standard tessellation language) file - there are many other options available, although **.stl** is the most commonly accepted by conventional 3D printing software. This process is automatically done by most CAD software and it interprets the 3D part as a mesh made out of small triangles (the size of these triangles depends on the level of specified accuracy). The **.stl** file will be converted into a **.print** file via **MakerBot Print** that translates the triangular mesh into many slices: eg. a 100mm tall bunny at a 0.2mm z-resolution consists of 500 layers. These slices are essentially layers of GCode that instruct the 3D printer of the cartesian coordinates where the nozzle and build platform need to be in order for the model to be constructed.

In order to have a working **.stl** file, the part has to be 'manifold'. A manifold part is a combination of geometries that can be physically recreated: this means that the geometry cannot intersect one another. Each surface has to have a thickness and two geometries cannot share just one edge or point.

### 2.2 Model Resolution

When exporting a CAD model as a **.stl** file, there are a few options for the resolution of the model. Lower resolutions mean that the number of polygons in the triangular mesh are lower than a high resolution **.stl** - however the file size is lower overall.

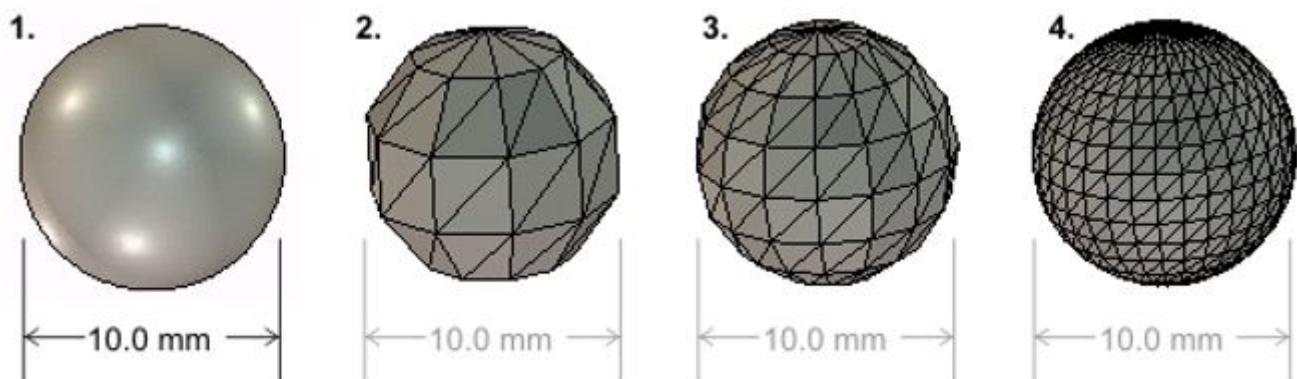


Figure 1: Model Resolution Differences

Although an in-depth understanding of .stl model resolution is not necessary, Figures 2a and 2b are provided for interest.

- Surface deviation is a measure of how much the tessellated object deviates from the true surface; it measures the edge of a facet to the actual surface.
- Angular deviation compares the surface normal of two adjacent faces and cannot deviate more than the specified amount.

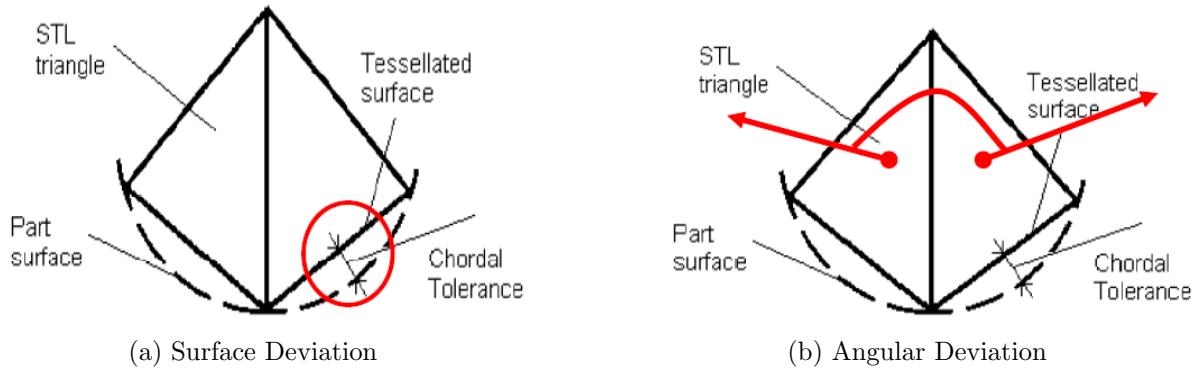


Figure 2: Model Resolution Settings

## 3 3D Printing Tips and Tricks

### 3.1 Model Orientation

When orientating CAD models in the **MakerBot Print** software, there are a few rules of thumb to follow. A tutorial of how to use **MakerBot Print** is not provided here, but can be easily found online.

When orientating models, care should be taken into considering if support material will be generated (explained in more detail in Section 3.2)- each layer needs to be printed on a pre-existing layer although there are some exceptions to this. Exploring Example 1 in Figure 3, it can be seen that the Bad Orientation has overhanging sections that will require support material to be printed in order to support the overhang - if support is not selected, failed overhangs may result as per Figure 4. A simple reorientation will resolve this issue. In some cases, there may be no optimal print orientation and support material may need to be generated no matter how the CAD model is orientated.

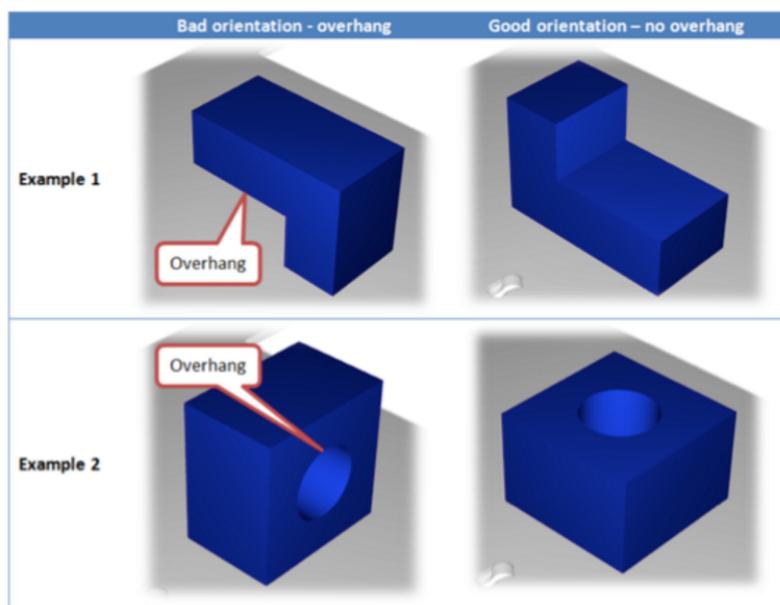


Figure 3: Examples of Good and Bad Model Orientations

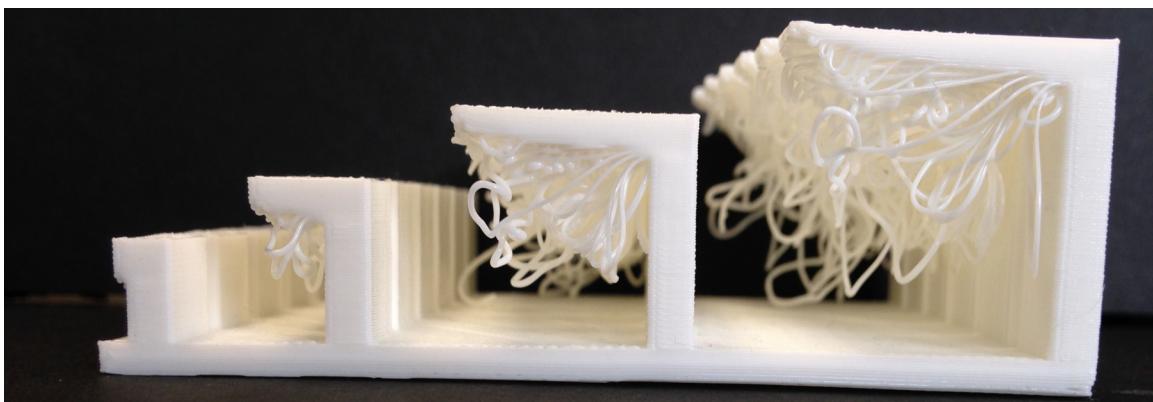


Figure 4: Failed Overhangs with No Support

There are exceptions to this and general guidelines can be used to determine how much of an angle can be printed without support, or how much of a horizontal bridge can be constructed without support material/failed overhangs. This is documented in Section 3.2.

If strength is a requirement in the design, model orientation is an important consideration when 3D printing. Generally speaking, the build direction (the z-axis which controls the height of the build layers) will be the weakest under bending due to the comparatively low bond strength between layers. Re-orientating the model such that the bending stresses act to bend the filament layers would be best.

### 3.2 Support Material and Overhang

Support material may be required for the print if part of the print will not be building upon existing layers. This is referred to as either overhang (where the print is extended out to the side, beyond the borders of the layer underneath) or bridging (where the print layer is supported by layers underneath on either ends). If the support option is selected, material with a light fill will be printed underneath these areas which can be removed once the print is completed.

For overhang, printing with PLA will accommodate designs with an overhang angle **65 degrees** from the vertical. A design with an angle greater than this will require support in order to print. For bridging the 3D printers at the 3D Innovation Centre are able to accommodate up to a **5cm horizontal bridge** with no support. This can be seen in Figures 5a and 5b. Printing with support requires more printing material, so will take longer and be more costly.



(a)



(b)

Figure 5: Angles and Horizontal Overhang with No Support Material

A preference for support material can be **indicated** in the submitted .print file, however it is up to the operator's discretion as to whether support is required or used. When the job has been successfully completed and harvested, the support material will still be attached to the part. The print will also be printed on a **raft** (a light printed base attached to the print). Both raft and support can be removed by gently pulling this away from the print. The 3D Innovation Centre **DOES NOT** remove support/raft material **NOR** take any responsibility for fragile designs that may break during the removal of the support/raft material.

### 3.3 Infill

Infill refers to the density of the material used in filling the printed object. The percentage infill specifies how dense the print needs to be and high percentages consume more material, as well as taking longer to print (illustrated in Figure 6). Most prints do not require a high infill unless they will be undergoing high concentrated stresses. The 3D Innovation Centre standard is to set the **infill to 10%**. If a higher infill is required, the desired infill may be requested as long as the reason for using the requested infill are provided - please fill in the 'Notes' section of the Job Request. If infill has been set higher than 10% without a reasonable justification, the operators will automatically change the setting to meet the standard.

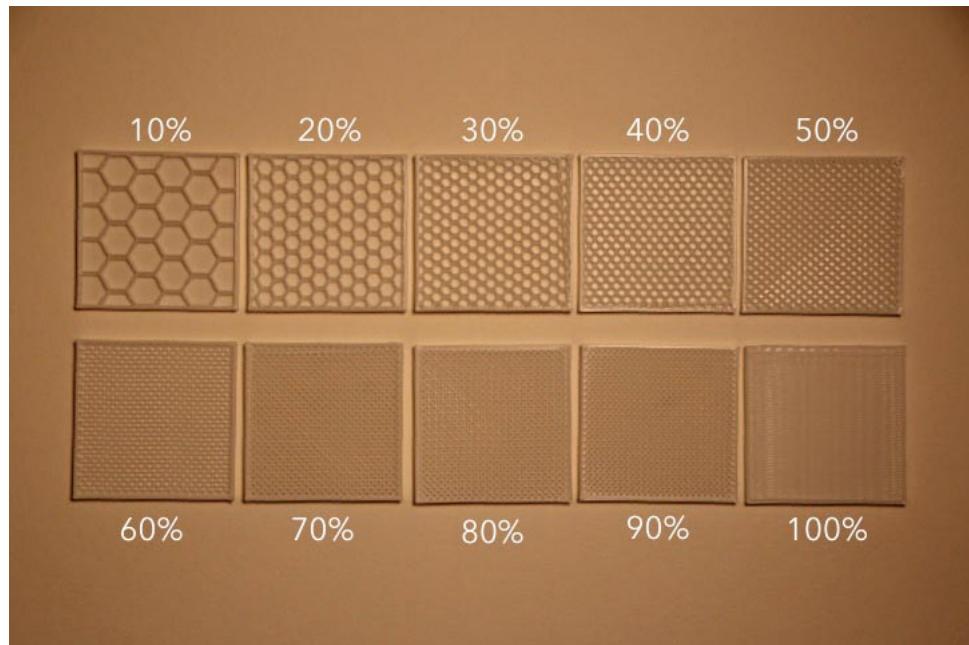


Figure 6: Varying Percentage Infill

### 3.4 Layer Height

The layer height is the thickness of each printed layer. Prints with a greater layer height will therefore have less accuracy and detail in the print than prints with a lower layer height - this can be seen in Figure 7. However, printing with a low layer height will take much longer to print and require more material, increasing the cost of the print. The standard in the 3D Innovation Centre is to print with a **0.2mm layer height**. If a print is required with a lower layer height, the desired layer height must be requested in the notes attached to the print request. If sufficient explanation is not provided, the operators will automatically change the setting to meet the standard.



Figure 7: Varying Layer Heights

### 3.5 Shells

The shell is the outer wall of the printed object. Shells refers to the number of these printed walls - depicted in Figure 8. A print with a higher number of shells will therefore be able to withstand more force applied to the outside of the print. The standard for printing is **2 shells**. Once again, a higher number of shells will generally not be required, but if requested, reasons must be provided within the print request notes otherwise the operators will automatically change the setting to meet the standard.

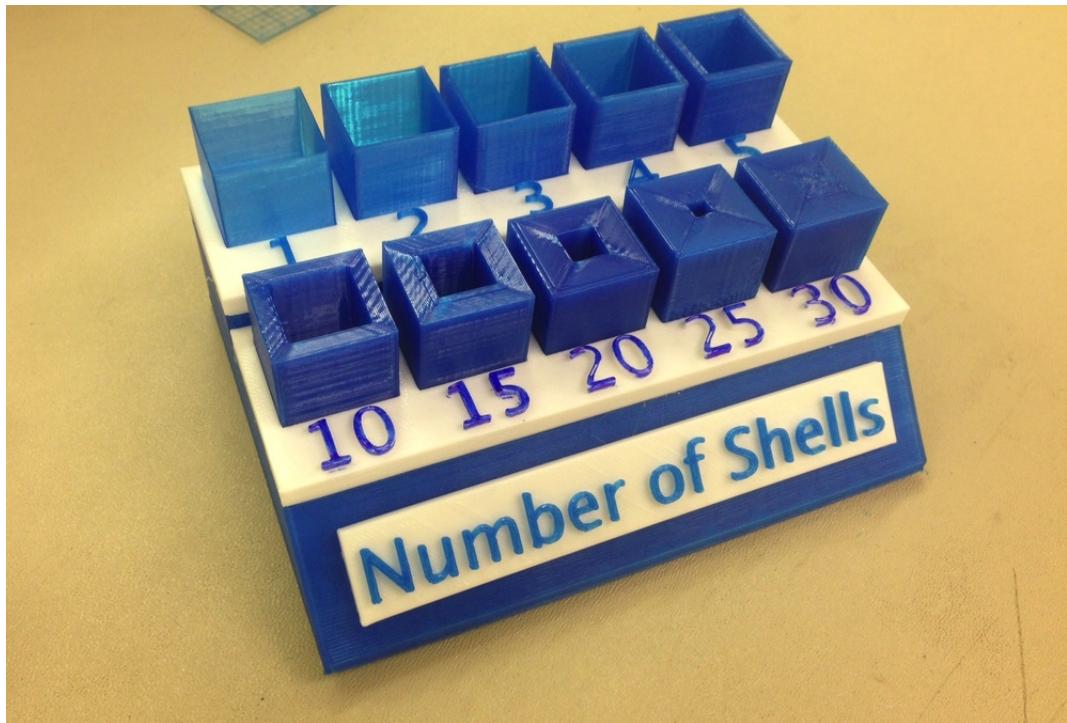


Figure 8: Varying Number of Shells

## 4 Troubleshooting with 3D Printing

Although there are a number of problems that can arise with 3D printing, the operators at the Engineering Workshop will be responsible for mitigating the large majority of these issues. What the user will need to be aware of though, is warping.

### 4.1 Warping in 3D Printing

Warping is an issue that can occur in 3D printing. This can be seen in Figure 9. As the extruded plastic cools and the hot nozzle moves around the build plate, thermal gradients can be induced - when the thermoplastic is heated it expands slightly, following this, the material cools and contracts. Room temperature fluctuations can cause layer separation, print lifting and warping and may happen with larger prints, or having multiple objects on one print bed.

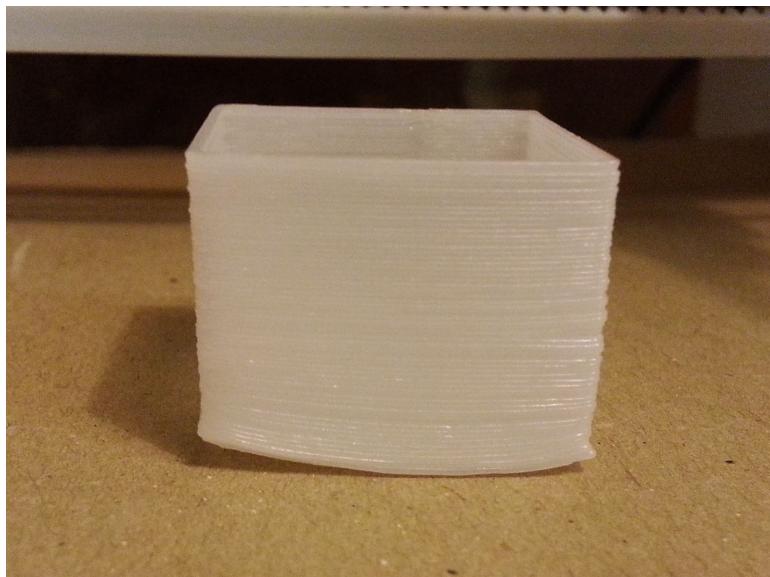


Figure 9: A Warped Part

## 5 Questions/Feedback/Contact

Please direct all questions regarding the 3D printing process to [MSE-3Dprint@unimelb.edu.au](mailto:MSE-3Dprint@unimelb.edu.au). Please direct any questions about the Engineering Workshop to Kevin Smeaton at [k.s@unimelb.edu.au](mailto:k.s@unimelb.edu.au). Please note that the Engineering Workshop is open between **9-5pm** and closed between **1-2pm** Monday-Friday. Job requests are processed and harvested from **9-10am** and **4-5pm** Monday-Friday. **NO** access to the 3D Innovation Centre is allowed without prior approval of the operators.