

# A Tutorial on 3D Printing Robot Parts

## Why We Use 3D Printing.

3D Printing provides an excellent platform on which to rapidly prototype and produce working parts for relatively low-stress tasks. This tutorial will explain how we, the Marianopolis CRC Team, applied this piece of technology to enhance our workflows by resolving atypical problems through solutions made accessible through 3D Printing.

Before we continue, please note that FDM 3D Printing is not without its hazards. As the writer and an avid tinkerer, myself, I must urge that FDM 3D Printers be handled with utmost care to avoid health issues from fumes, and severe injuries or even death from electrical shocks from the exposed electronics on some of these machines (particularly ones from China). In addition, they aren't fast: a single 20-teeth Idler Pulley takes 5-10 minutes to print while a helmet takes several days to print. Also note that these printers are basically making plastic spaghetti walls, so you can't expect a printed duplicate of a metal part to perform anywhere nearly as well as the original.

All that said, we still used FDM printing because said spaghetti walls are surprisingly strong when thick, and 3D Printers are significantly more accurate than humans if precision tools, such as lathes or mills, are unavailable. With the right tools, 3D Printed parts can be perfect for many mechanical purposes. It remains an versatile and powerful tool.

## The Tools.

3D Printing is actually relatively accessible today due to lower costs and its incorporation with devices available to all. To design and print parts through your 3D Printer, you will need four things: a 3D Modelling Software, a Computer for that Software, a Printer, and some nice tools for finishing the printed object – namely a sharp scraper, some thin pliers, and some thin flush cutters.

## Phase 1: Part Designing on the Surface Level – CADing

The first thing to do for designing a part is to, well, design the part! You will need to design your 3-dimensional object through Computer-Aided Design (CAD) Software. For beginners, I recommend TinkerCAD for creating 3D Objects for printing. When designing your part, it helps a lot to visualize the requirements for your part and work around those. Though there is not one single best way to CAD, I recommend that you take measurements of the interacting parts – from their size, their shape, to the distances between one another - and build a basic geometry surrounding those measurements first, then cut away at the part until you have a simple, yet effective model that considers the requirements (interacting parts, like screws needing holes), the constraints (limited space), the assembly (convenient things like adding holes for screw drivers to access screw positions), and the printing (you must have a relatively large flat surface on that acts as the base for the print). Do keep in mind that 3D Printing requires tolerances on the level of a few hundred micrometres – that is to say that features of your object may be parts of a millimetre more or less wide. If your design has holes, I suggest tolerances of 0.2 mm to 0.4 mm. Once you are satisfied with your part, you can now export the shape as an .stl or .obj file for usage in a Slicer program. You have only created the skin of your object. Next is the internal structure.

## Phase 2: Part Designing on the Structural Level - Slicing

You now need to “Slice” your part. Slicers are programs that take your 3-dimensional skin and slice them into vertical layers for printing. For this, I recommend using Cura, which is a versatile Slicer with many adjustable print settings. When setting up, you must select the plastic filament you will be using. I recommend PLA filament for beginners as it is easy to print with and produces rigid, albeit brittle, parts - though you should keep in mind that it starts deforming at 70 degrees Celsius. In Cura, you must select pre-configured settings for using PLA in the filaments section. This will load good nozzle and bed temperatures for printing with PLA, although you should verify that it is within the filament manufacturer’s recommended temperatures. From here, you should also adjust these settings:

- Use 0.3mm Layer Height and Base Layer Height;
- Change the number of walls from 2 (default) to 4 (strong enough for many small things);
- Change the number of bottom and top layers to 4, such that it is 1.2mm thick; and
- Enable the printing of outside walls before inside walls in the Shell section for accuracy.

## Phase 3: Printer Operation – Printing

Operating a 3D Printer is relatively simple when the printer is tuned properly. If the printer has not been tuned, tune it. Maintenance is of utmost importance to ensuring the quality of produced parts. If the printer is in good shape, you can begin the print by saving the slicer’s .gcode file onto an SD Card, inserting it into your FDM printer, and activating it as the print. Be sure to remain present for at least all of the printer’s heating time and for the first layer. The first layer of a print is the most important layer as it will decide whether or not the print results in a success or failure. If the plastic coming from the nozzle appears to be pressed onto the bed platform, but not transparent, then the bed is at a good height. If this is not the case, you should stop the print so that you can adjust the tension of the bed springs until the right height is reached. Be careful about the remaining heat on the bed! At high temperatures, it can burn.

Some printers do not have a heated print bed. This is a problem as it limits the capabilities of printers. As such, I recommend obtaining a printer with a heated bed or a heated bed or mat to upgrade a non-heated bed printer. Adding a glass plate about 3mm thick will also greatly increase the quality of prints as a glass bed heated to the glass-transition temperature of plastics will allow plastics to stick on while warm and pop off when cooled.

Once the printer is printing with its first layer perfected, the printer can now be left to finish the print on its own. During this time, you may notice that PLA filament has the smell of cooking oil (you might also just not smell it). The fumes from PLA have not been noted as toxic. However, if you use other filaments, this cannot be guaranteed. Styrene fumes from ABS filament, for example, is carcinogenic. Try to keep the printer in another room!

## Phase 4: Part Finishing and Using

Once the print is complete, you should let the print bed cool down 15-25 degrees Celsius and afterwards use the scraper to lift the printed object off the bed. If you printed with supports or have minor imperfection you’d like to get rid of, you can remove them using the pliers and flush cutters suggested earlier. And with all that done, you now have a functional part fully design. Nice job!