So you want to print something?

- Get **.STL** model file from an online database (Thingiverse, GrabCAD, etc.), or design your own! (TinkerCAD, Fusion360, OnShape, etc..)
- Open STL file in Slicer software (Cura, Slic3r, etc.)
  - Slicer software converts the STL into gcode (machine specific instructions) that control the motion of the motors to build the print in layers.
  - I use Cura, I recommend it as it is free and has a lot of support from the open-source communities.

- Ensure slicer software is configured to physical printer settings (often presets can include specifications of the printer, i.e. Cura has Ender3 Pro preset).
  - Main thing: ensure print bed size is 220 x 220 x 250 mm (8.6” x 8.6” x 9.5”)
  - Second most important: **PLASTIC** settings are important. Usually PLA is the most common. Ensure settings for PLA are selected (Extruder/Hotend: 190-210 deg C, Bed temp: ~60 deg C).
    - ABS printing is difficult since it is so temperature sensitive. Trade off is that ABS can be a lot stronger than PLA. However, PETG is an alternative to ABS that is easier to print and is stronger than PLA.

- Orient STL model in slicer software so that the **flattest surface sits on the print bed**. This ensures easier prints, less support material used, and less headaches for you! Two ways to do this. In Cura:
  - The left hand panel has settings to manually rotate the model on the print bed, or
  - Have the left hand panel module automatically lay the model flat.
  - In position module, ensure z-position is 0mm (flat on the print bed)

- In Slicer, there are many settings to choose from. The main ones include:
  - **Print Quality / Layer height:**
    - The lower this value is, the higher quality of the print. This offset with a much longer print time. (~0.1mm should be the minimum to try to print at).
    - A **standard print is 0.2-0.25 mm** layer height, but don’t be afraid to try up to ~.3 mm for prints where precision doesn’t matter as much (phone stands, etc).
  - **Support:** If you have a print with overhangs this will be your friend.
    - Two types: Normal and Tree. They function in similar fashions but the tree uses less material and can be easier to remove than normal support.
    - Everywhere vs. Touching build-plate (print bed).
  - **Bed Adhesion:**
    - Skirt - Not too important to functionality, just ensures print bed is level and prepares for print by making a few lines around the print. OK to turn off, but it does give a preview before printing starts to let you know whether the print will be successful or if you’ll have to reconfigure and restart if unsuccessful.
    - Brim - Important if you are trying to print something with a small base, ensures adhesion is kept throughout the duration of the print.
    - Raft - Useful if you are having troubles with adhesion otherwise. Builds a surface for the actual print to rest on. Plastic adheres more easily to plastic than it does to a print-bed.
  - **Infill:** Higher percent means more material inside the print surfaces, results in a stiffer / stronger finished product. Play around with these, I usually try for 25-50%.

Before you print:

- **Ensure print bed / surface is level / calibrated**
    - Four knobs at every corner under print bed tighter/loosen springs that lower/raise print surface.
    - First: Under **Prepare** menu -> Autohome, disable steppers.
        - Lowers extruder head (Z-axis) and pushes print bed backwards (Y-axis) to **Home** position
        - X-axis motor (side to side) has to be pushed manually all the way to the left until clicking the limit switch reaching its **Home** position.
    - After everything is in **Home**:
        - Stick a piece of paper under the extruder / hot end.
        - Loosen screw (clockwise) to raise the print bed at each corner until there is tension when trying to move the paper.
        - Careful to make sure the extruder does not scratch the print surface too much. If so, tighten screws (counter-clockwise) to lower the print surface for easier travel.
    - **Has to be done semi-frequently. And it may take a lot of tries.**
        - Usually it stays level after a few prints, but things may move. So do it.
        - I didn't get a printer that auto levels, those are expensive… sorry lol.

- **Changing Filament**
    - Preheat hotend / extruder (under preheat PLA settings in **Prepare menu**).
    - There is a spring + handle on the extruder motor that holds a bearing keeping the filament in place as it goes through the filament feed tube.
        - Push the handle to open the space between the bearing and gear allowing for the filament to come loose.
    - Manually pull out the filament from the extruder head, ensuring there are no crazy tangles in the filament spool when you roll back.
    - Replace with new filament spool through the extruder inlet, ensuring you are pushing the handle to open a space between the bearing and gear. (Basically do the last steps backwards)
        - Might be necessary to use wire cutter pliers (the blue one close to the machine) to taper filament so the gear are able to grab and feed the filament through the feeder (Bowden) tube.
- **Loading Sliced Model (gcode) to Printer**
    - SD Card Method:
        - Simply connect SD card to computer using SD card adaptor, load gcode file (NOT STL). Safely eject.
        - Insert SD card to printer (bottom left of printer)
        - Click printer menu wheel, select **PRINT FROM TF**, choose your file, sit back and watch the printer print.
    - Octopi Method (**Remote (local) printing,** work in progress, try at your own risk):
        - Connect RaspberryPi in white case to power supply (mini usb) from outlet, use USB out from Pi to printer in (bottom left).
            - Pi hosts a local server on the wifi that loads a print interface, allowing remote printing from the local network.
        - On a web browser go to: http://octopi.local/
            - User: octopi
            - Pass: 2053
        - Bottom left of webpage, upload gcode file. From there, you can start the print, stop the print, monitor progress. Be on standby in case you need to physically turn off the printer.

While Printing:

- Observe for any initial issues when starting the print; most print problems begin at the first few layers.
- Adhesion: print “initializes” with a test line that clears dried plastic in the extruder
    - If the plastic doesn’t stick - STOP the print (click the knob, select STOP print). Auto Home the printer, clear any plastic on the print bed, and recalibrate the print bed. (i.e. level the bed).
- If you need a precise print - you can slow down the feed rate (FR) found on the control panel (bottom left of screen) by turning the knob and bumping it down to 75-85%. After the first few layers, I recommend bumping it back up to 100%.
    - If you are not too worried about the precision of your print (printing a stand or shelf, for example) - you can speed it up by bumping up the FR to ~125%.
    - If you want a balance, leaving it at 100% FR is fine.
- Watch for tangles in the filament. You might hear a grinding between the extruder head when it is not able to pass filament. Manually loosen the filament spool to fix.
- Check every 15 mins to ensure no errors happen, material is not wasted.
- When print is done - remove magnetic bed to remove your print.
    - Post-processing might be necessary, especially if your print used support material. The pliers are your best friend.

Things to Remember:

- Google and Youtube (CHEP is a really good YT channel) are your best friends
- Don’t be afraid to mess up! Failure is the first step to learning.
- Do not leave unattended until you feel confident your print will not mess up.
- Call or message me whenever anything comes up - and show me your prints!