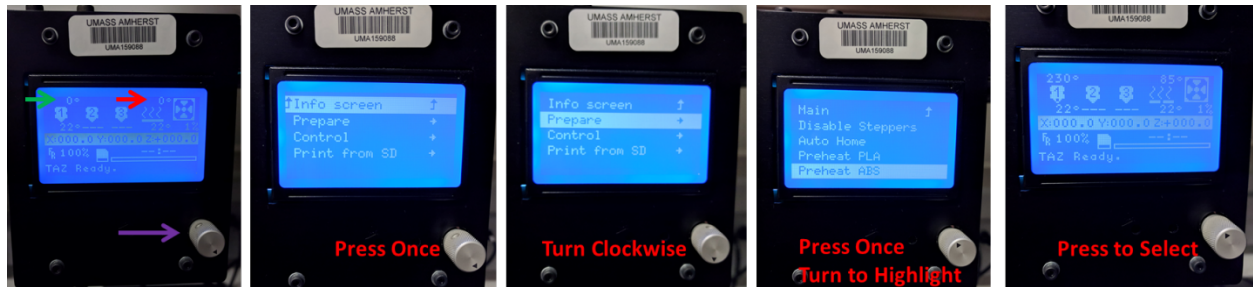


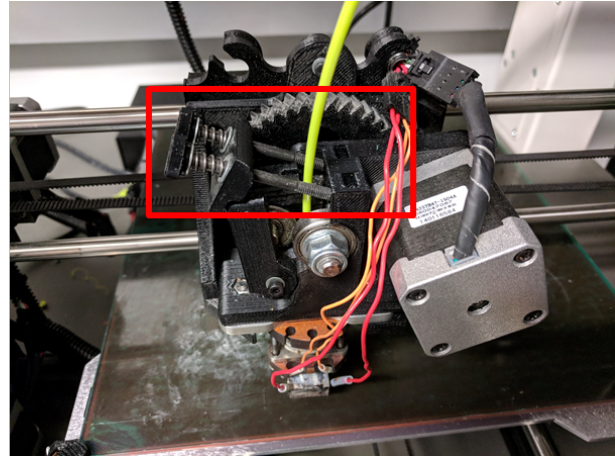
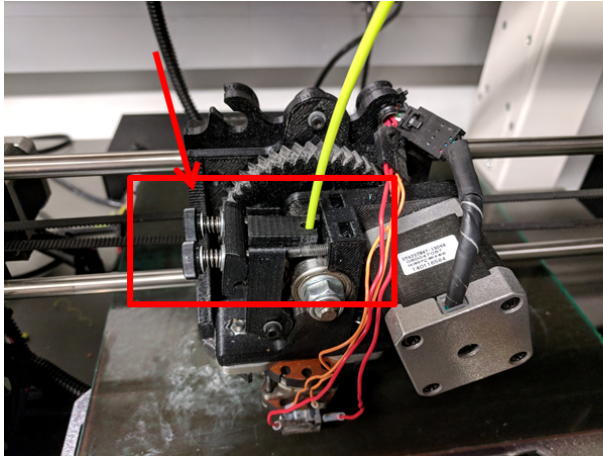
3D Printing with the Lulzbot 4

Alyssa Schwartz on 7/2/2018

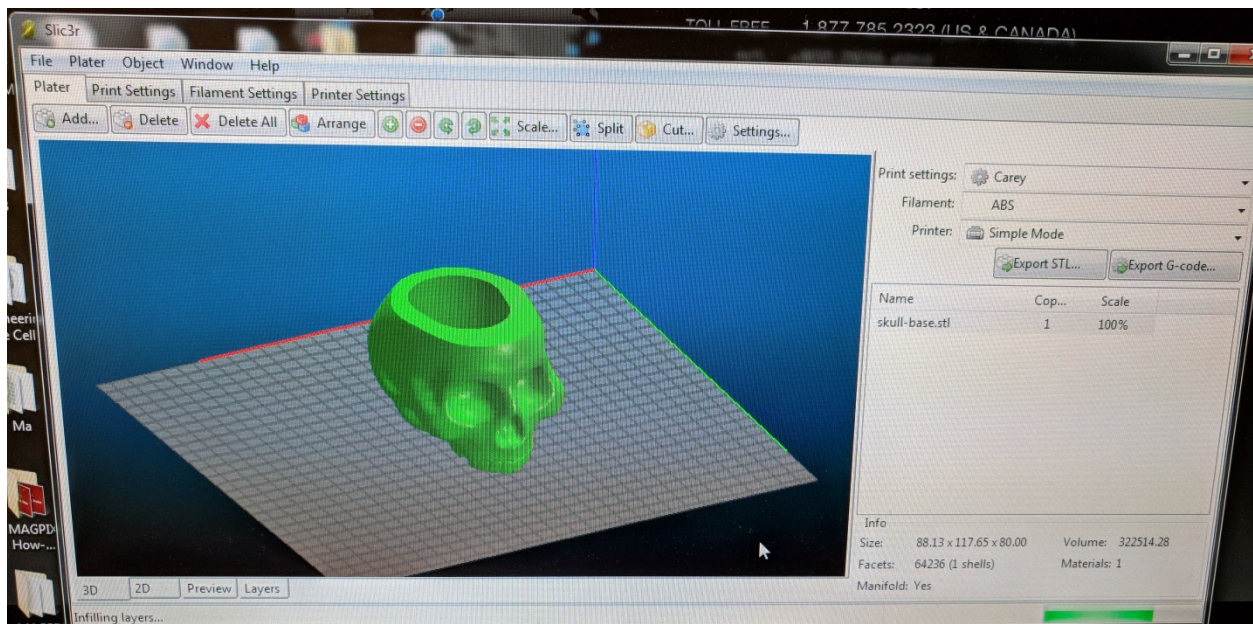
1. Find or Design your Object. This can come from almost any source, but needs to be in a .STL file format.
 - a. Thingiverse.com
 - b. <https://3dprint.nih.gov/>
 - c. Design in any CAD software. One example of this is FreeCAD – here are some tutorials for that particular software.
<https://www.youtube.com/channel/UCuwmvJXdUCik-Z6goNzTxKQ/videos>
<https://www.youtube.com/user/MarthamEngineering/videos>
<https://www.youtube.com/user/FreeCADNews>
 - d. There are many other options (ex. If you take 8 pictures of an object to capture all 360 views there are online programs that can reconstruct the object... or person)
2. Choose your plastic:
 - a. ABS (acrylonitrile butadiene styrene): Easy to print with, Tends to make stable prints, surface can be smoothed using acetone, can warp during printing as the lower layers cool. Print temp = 230°C, bed temp = 90°C-100°C.
 - b. PLA (polylactic acid): Easy to print with, Tends to make stable prints, Cools quickly which is good for objects with overhangs, less likely to warp, not soluble in acetone. Print temp = 180°C-200°C, bed temp = 60°C.
 - c. T-glase/PETT (polyethylene terephthalate): More difficult to print with, but is good for objects with overhangs. Minimal warping. FDA approved if you want something food grade. Print temp = 230°C, bed temp = 80°C.
 - d. HIPS (high impact polystyrene): Easy to print, good with overhangs, can warp some. Print temp = 230°C, Bed temp = 100°C.
3. Turn on the printer. There are 2 switches that need to be on:
 - a. Behind the printer there is a box with a large red power switch (part of the power cord).
 - b. On the left front of the printer, there is black power switch that needs to be turned on.
4. Preheat printer by following the steps below. Note that when you connect the computer to the printer it may reset the set point to 0°C, and you will need to set it to preheat again to make sure it continues heating. The green arrow is the nozzle set point, the red arrow is the bed set point, and the purple arrow is the silver control knob.
 - a. Press the silver knob in to get to the main menu
 - b. Turn the silver knob to scroll through the menu, land on Prepare and press the silver knob again to select.
 - c. Scroll to either Preheat PLA or Preheat ABS depending on the temperature you need. The PLA set point is 180°C and the ABS set point is 230°C.



5. If you need to change your plastic, heat the printer to the temperature needed for the hotter of the 2 plastics that you are switching between. Once it is fully warmed up, loosen the screws that hold the plastic against the feed wheel and gently pull the plastic out of the print head, from the top. Feed the end of the new plastic into the print head, pushing gently until you see a small amount of the old plastic come out of the nozzle. Then tighten the screws to hold the plastic in place. Use the extrude feature (in increments of about 50mm) until the extruded bit is completely the color of your new plastic.

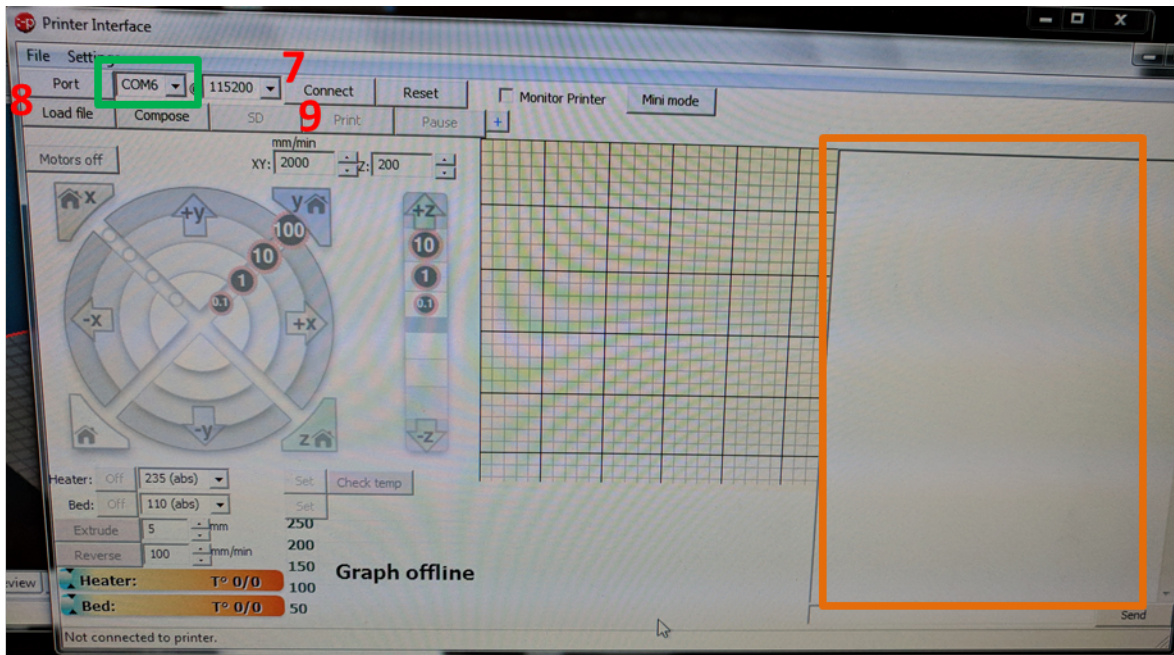


6. To prepare your object for printing, open the program Slic3r, and either click *Add* and browse to find your item, or you can drag and drop it into the window. Under print settings you can set the nozzle and bed temperatures, the infill, and any supports. Then click “Export G-code” to generate your ready-to-print file.



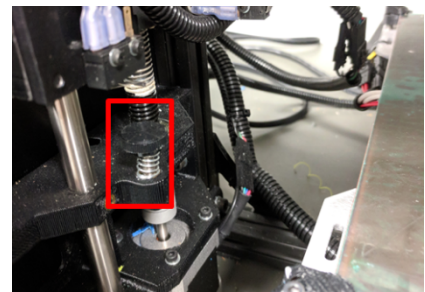
7. Open the printer interface program and connect to the printer using the connect button. Make sure the port (green box) listed is COM6, otherwise it may not connect. See image below step 9.

8. Click Load File and navigate through the menus to select the .GCODE file that you would like to print.
9. Hit Print. At this point, on the right side of the Printer Interface program, the printer will start displaying lines of information (orange box). If each line includes the parameter "W:" followed by either "??" or a number 1-9, then the printer is ready to go and just waiting until it is sufficiently close to the set point to start extruding. If this parameter is missing, you will need to repeat step 4, and only step 4, after which the printer will probably respond and add the "W:" parameter. If the "W:" is still missing, try preheating to the other plastic, then immediately preheating to your desired plastic.

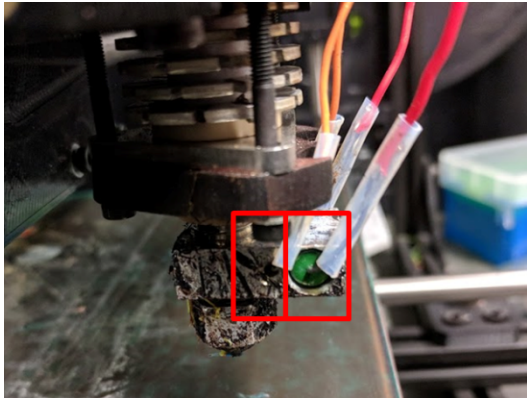


10. Troubleshooting:

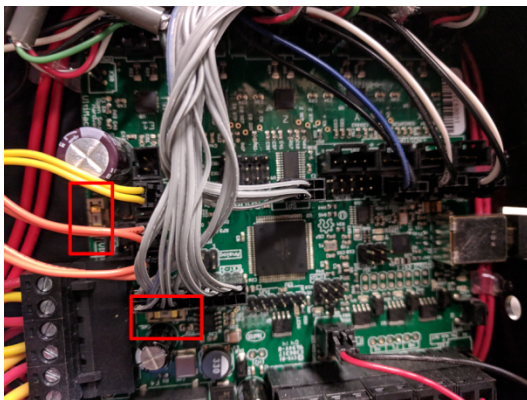
- a. If your print does not stick to the bed, here are several options that you can try:
 - i. Increase or decrease the temperature of the bed. (increasing the temp will result in more uniform adhesion, but if it gets too hot, the plastic will stay melted and not adhere either)
 - ii. Add a *very thin* layer of glue to the bed by using a glue stick
 - iii. Adjust the Z-homing height so the plastic is pressed more or less onto the bed during extrusion. This can be done via the screw to the left of the bed. As the screw is screwed in or unscrewed, it moves up and down, which is sensed by the Z-homing component, and changes the base height of the nozzle relative to the bed.
 - iv. In slic3r, you can move the print to a new location on the bed that may have a less damaged piece of green PET film.



- b. If your overhangs are falling apart, add supports in the 'print setting' tab in Slic3r. There are many support settings – you may need to do some troubleshooting on what works best for your individual print.
- c. If the screen doesn't turn on or the nozzle or bed will not heat up,
 - i. The resistor or thermistor may be blown. The red box on the left in the image is the thermistor, which provides feedback to the system about the current nozzle temperature. If this is broken, the print head won't heat up for safety reasons, because it may get too hot. The red box on the right is the resistor that provides heat to the system. If this is broken the system can't heat up. Neither of these pieces are expensive (<\$5) and each has a life expectancy of 3-6 months.



- ii. One of the cords might be unplugged – check and re-plug any flexible wires.
 - iii. A fuse might be blown. If you unscrew and open the box on the left of the printer (where the second power button is), this is what it looks like inside. On occasion, a fuse might be blown that impacts the blue screen or the nozzle/bed heating. If you think this is the issue, carefully replace the fuse (red boxes below) with a pair of tweezers.



- d. If the plastic gets stuck when you try to change it, let it melt more at a higher temperature. Try wiggling the plastic feed as you pull up to remove it from the print head. Sometimes the printer is inexplicably feisty and trying again tomorrow will yield better results.

- e. If the printer head is moving in the shape of your print, but nothing is coming out, check to make sure that your nozzle temperature is at or above the melting point for your plastic! If it is, you may need to wait for more of the plastic to melt, because there may be a jam. If the plastic was recently changed from one with a higher melting temperature, turn the nozzle temp up to that of the original plastic and wait a while to let it melt. Occasionally, you may need to apply some pressure to the plastic feed, but be VERY CAREFUL if you do this: too much force will break the entire print head. If you try to extrude plastic before the nozzle has reached the set temperature, the notched wheel that moves the plastic feed down into the nozzle will tear up the plastic, and will no longer be able to grip and push the plastic. In this case, pull the whole feed out of the print head, trim several inches off (there will be a noticeable wearing down of the plastic where it meets the feed wheel if this is the problem).
- f. If the software will not connect to the printer, try restarting the software (or if that doesn't work, restart the computer). The printer must be turned on before the software is opened. Alternatively, the wrong port might be selected, ensure the port reads "COM6". If the computer still will not connect to the printer, you can load your print onto the SD card (which lives to the left of the small blue screen in the printer) and select "print from SD".

11. Other notes:

- a. Occasionally, the green PET film that coats the bed will need to be replaced. Peel off the old/cut up film, and carefully replace with a new one. After you do this, use the blue screen on the printer to perform the bed-leveling protocol.
- b. When a print is completed, the nozzle will automatically cool back down, but the bed will not. Make sure to turn off the printer and cooling the bed will help in getting the final object off of the bed.