# Create Your First 3D Print

1. Set Temperature

Set the hot end and print surface for ABS or PLA plastic and turn both on. Temperature settings for ABS should be set at **230°C** for the hot end and **85°C** for print surface; **185°C** for the hot end and **55°C** for print surface for PLA. These temperatures are for LulzBot filaments. You may need to adjust the temperature depending on the filament source, color and type. Click the **Motors Off** button.

2. Load Filament

A Load the plastic filament into the extruder by gently squeezing both the idler screws and the plastic clip together and pull upwards to release the idler. The idler can be rotated downward, allowing access to the hobbed bolt and filament feed hole. Once the hot end is up to extrusion temperature, remove the short length of ABS that is left in the extruder for shipping. If you are using PLA, you may need to raise the temperature to **230°C** to remove the ABS filament and then drop the temperature back down to **185°C**.
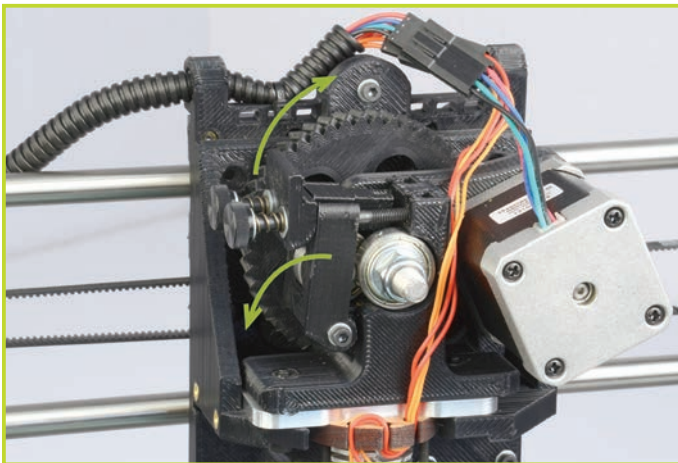
B Feed the end of the plastic filament into the filament feed hole and push the filament through the extruder by slowly pushing the filament down into the hot end.
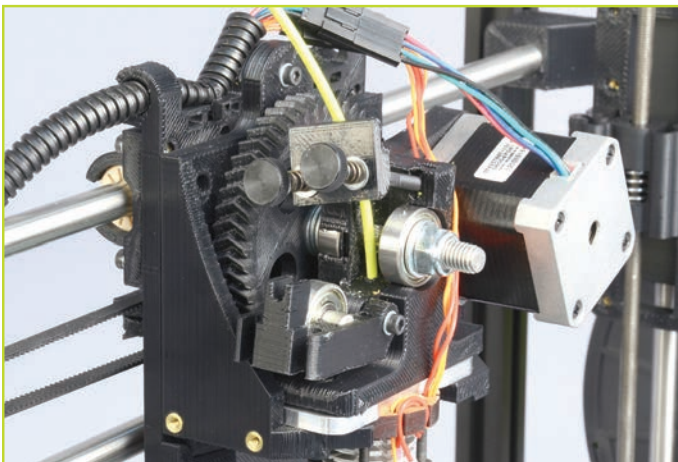

C Once the filament extrudes a small amount out of the nozzle, raise the idler and slide the two idler bolts and plate back into place. Tighten the two idler bolts. Tighten the two screws until the top of the thumbscrews are about 10mm away from the plastic clip.

D Use the **Extrude** button in Printrun to test the extruder. Slowly tighten the two extruder thumbscrews while extruding until you achieve reliable, repeatable extrusion. If the extrusion stalls, you may need to open the extruder and trim off any filament with a chewed out section.

3. Home Printer

A Use the home buttons to home the X axis and Y axis, then the Z axis. When the Z axis is at home, the nozzle tip should be right above the glass.
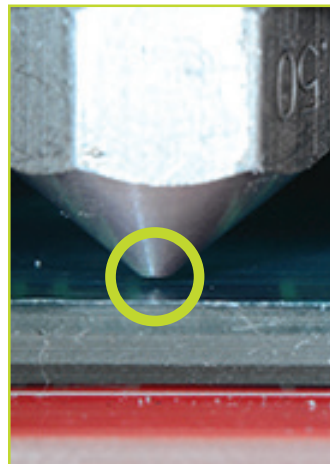
B To lower or raise the Z home height, adjust the Z end stop trigger, which is on the far left of the printer, mounted on the X axis motor mount. Lower it by turning clockwise and raise it by turning counter-clockwise. Once you have homed the axes and the hot end and bed have reached the correct temperature, it is time to print!


Z END STOP TRIGGER ADJUSTMENT

Z END STOP SWITCH
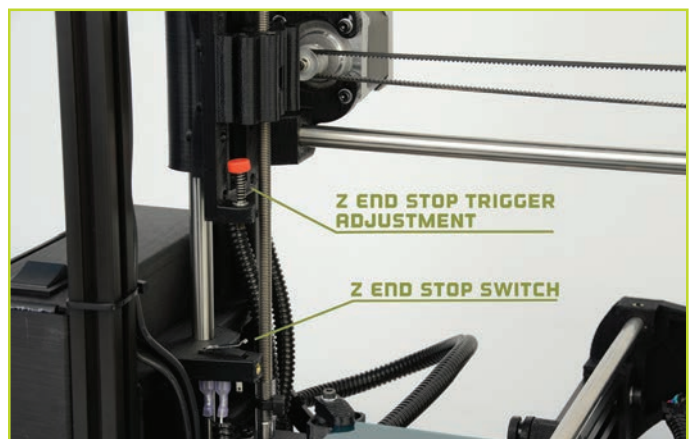
4. Z Print Height

A Load the **bed_level.gcode** file, which can be found in the calibration directory on the included SD card or at: **http://www.LulzBot.com/support/downloads**. Once you have downloaded the file, press the **Load file** button in Printrun.

B Highlight the **bed_level.gcode** file and select the **Open** button. It should appear in the Printrun G-Code viewer. Press the **Print** button to begin the print.

C When the print starts, check to ensure a good first layer adhesion. If the first layer is too high or low, pause the print and adjust the Z end stop trigger. Raise/lower a corner to minimize/increase the extrusion width to match the others. Remove any printed material and press **Restart**.

| VERY LOW | LOW | PERFECT | HIGH | VERY HIGH |
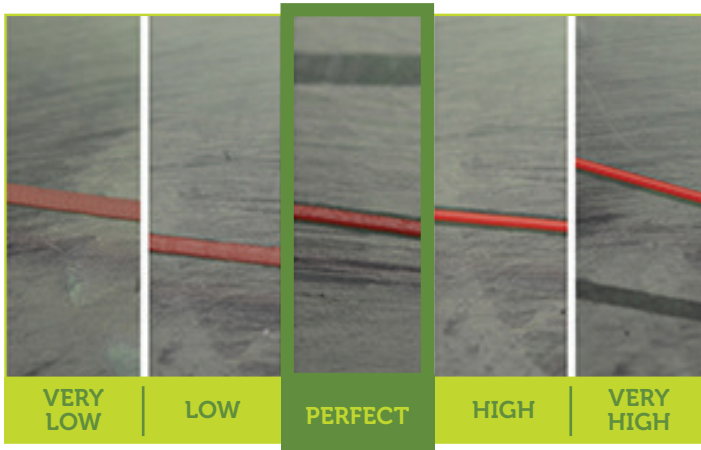
Your First Octopus!
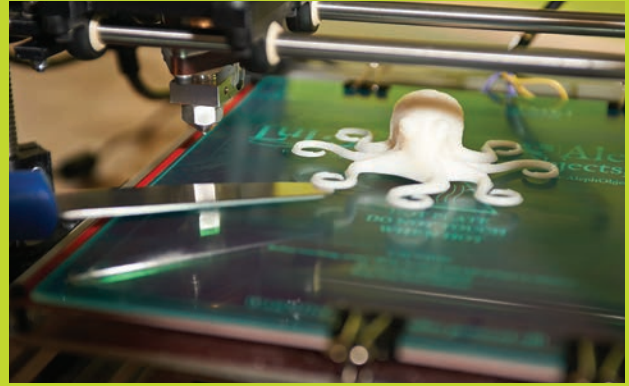
Load the **octopus.gcode** file:
**http://download.lulzbot.com/TAZ/3.0/novelties/**

Load the file in Pronterface, bring the hot end **(230°C ABS/185°C PLA)** and the heated bed **(85°C ABS/60°C PLA)** up to printing temperature.

Once the printer is at the appropriate temperature, press the **Print** button to begin the print.

5. Remove Part

A After the part is finished, the hot end will automatically cool down. If you are printing PLA, you will need to turn the heated bed off. Once the bed cools, pop the part off the surface with the included clam knife. If your part is large, you may need to pry at multiple points.

B Reset the heated bed to the correct temperature before starting the next print.
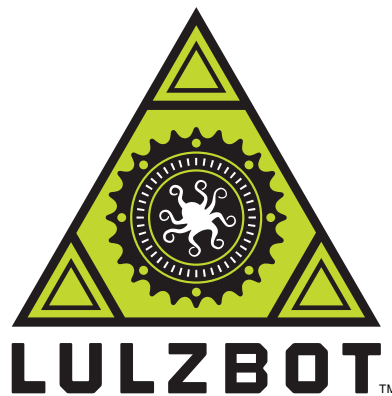
LULZBOT™

www.LulzBot.com

For more information, call **+1-970-377-1111** or email **support@lulzbot.com**

ALEPH OBJECTS® INCORPORATED

Buy printers, parts, and plastics at **LulzBot.com**