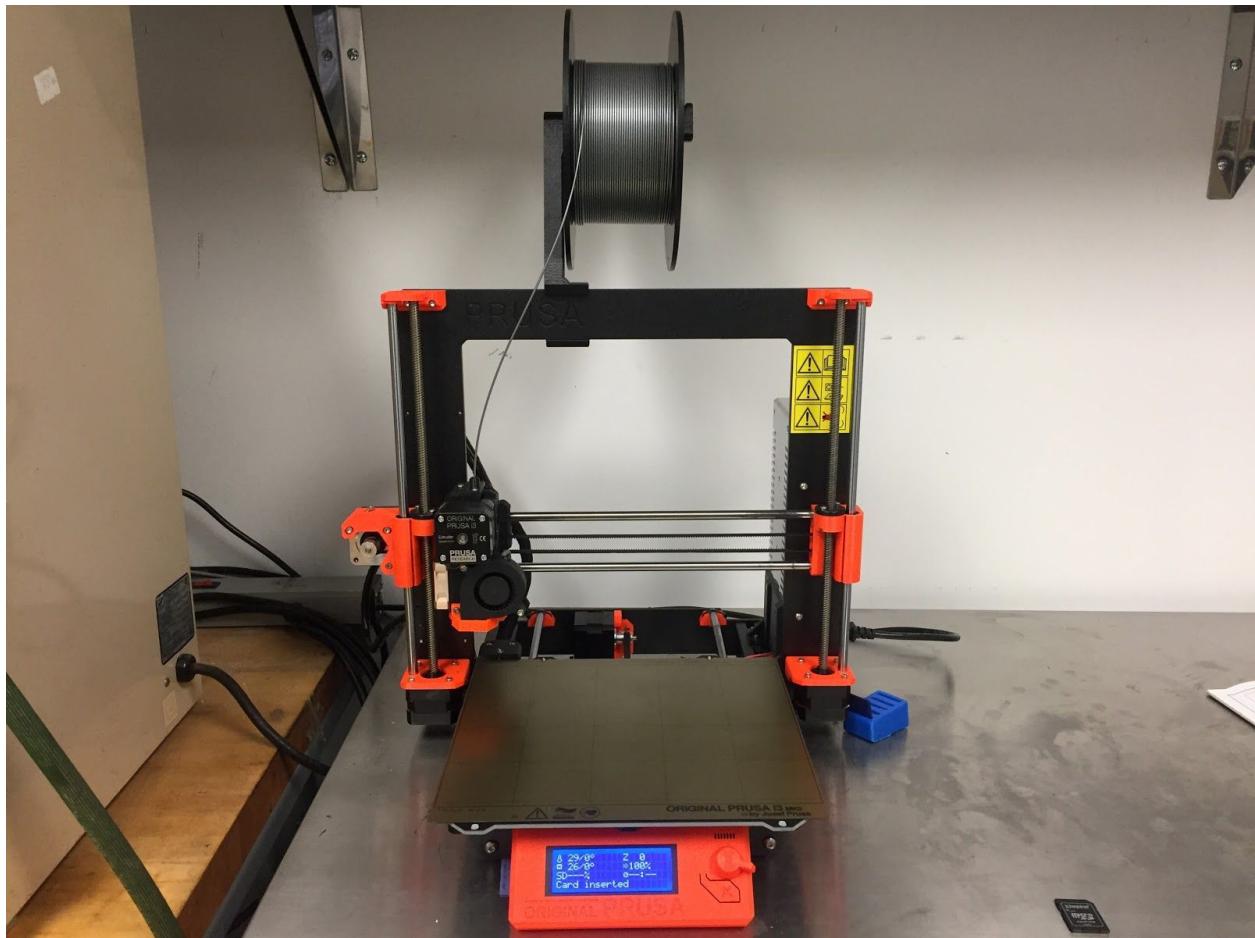


How to use the Prusa 3D Printer



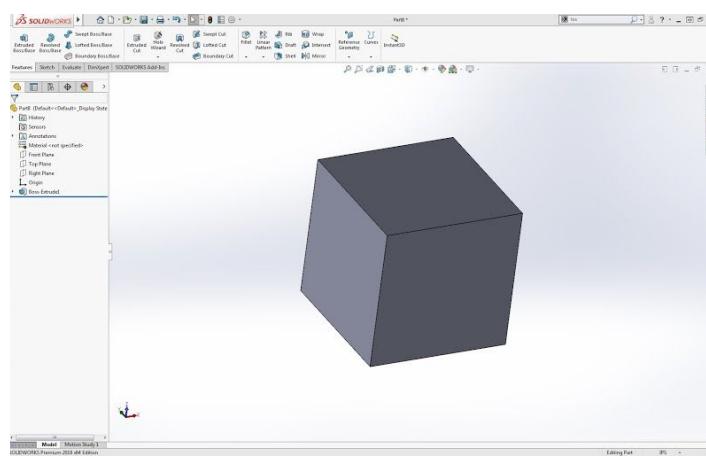
This tutorial will teach you how to use the Prusa 3D Printer:

Things you will need:

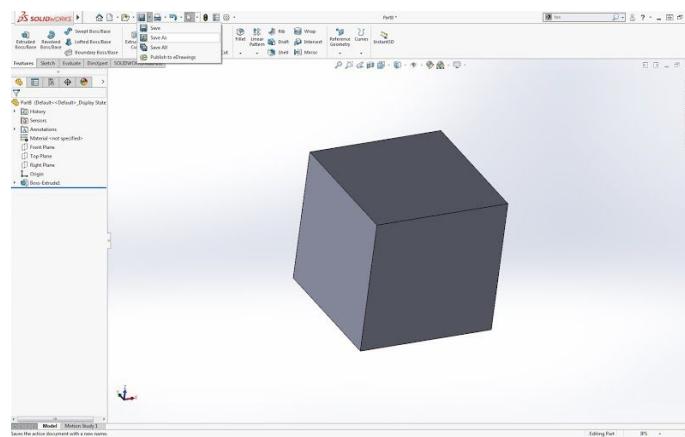
- An SD card
- An adaptor so your computer can read that SD card
- PrusaControl (if you do not have it you can download it at this link: <https://prusacontrol.org/#download>)
- A CAD program and a file that you would like to print

How to save your project in Solidworks:

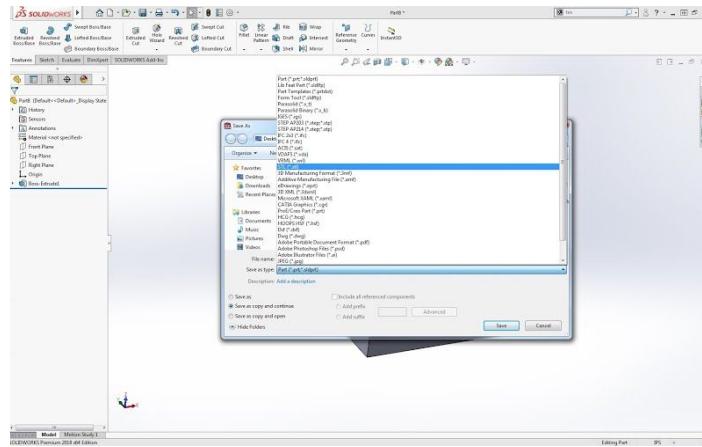
First open your project in Solidworks.



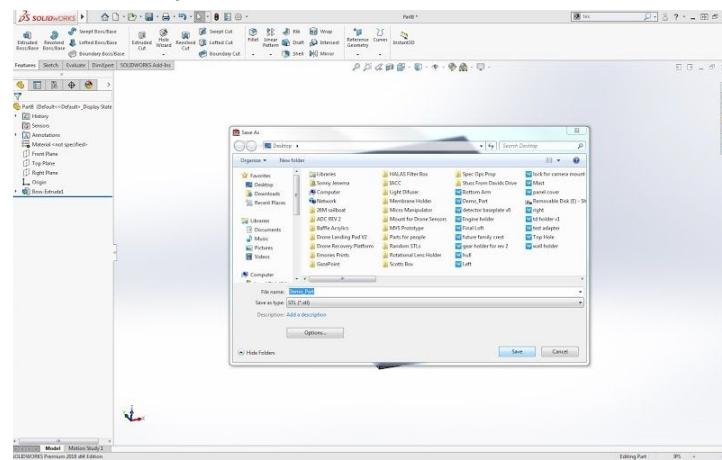
Click the save options button and select save as.



Once the window pops up, save the file type as stl.

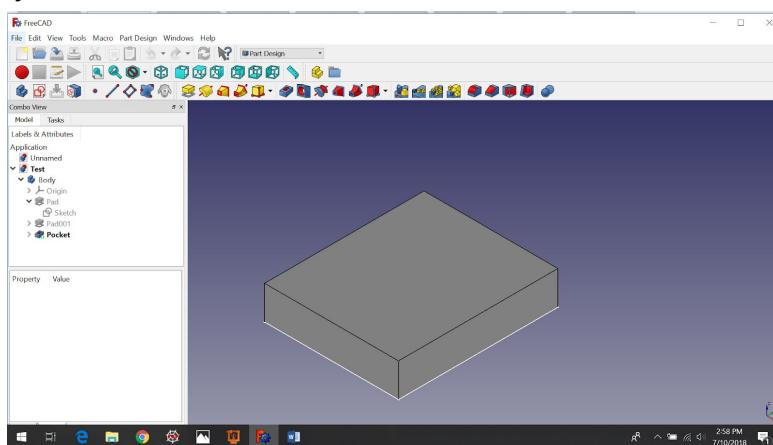


Once you have selected the file type, save it.

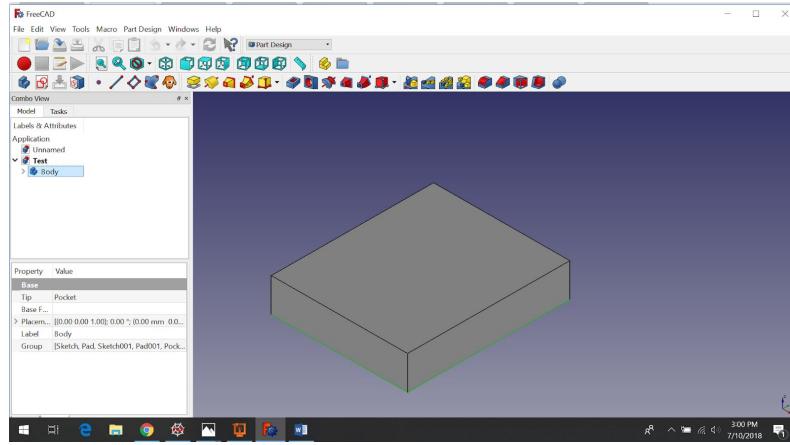


How to save your project in FreeCAD:

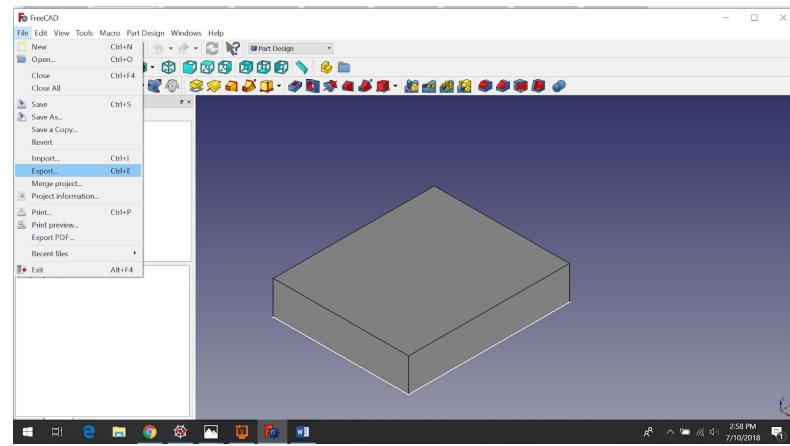
First open your project in FreeCAD.



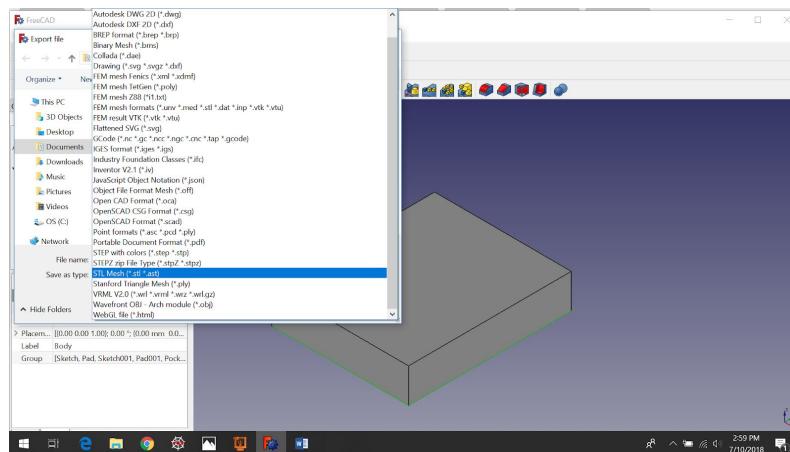
Select the parts you would like to print.



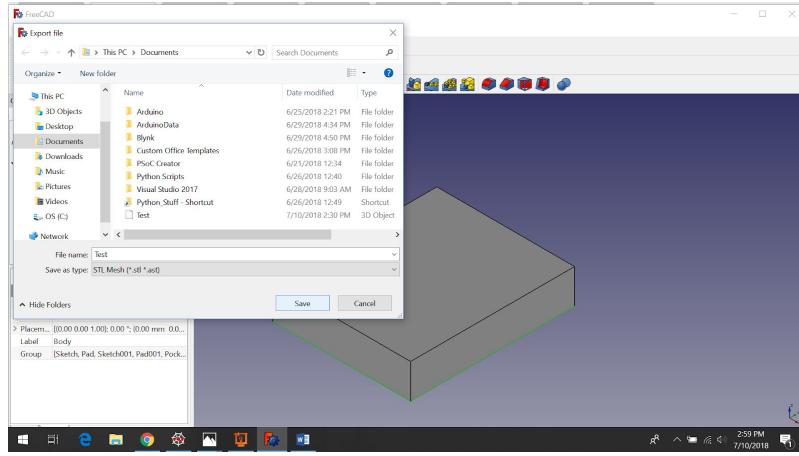
Click on the file menu in the upper right hand corner, then select the export option.



Change the file type to stl.

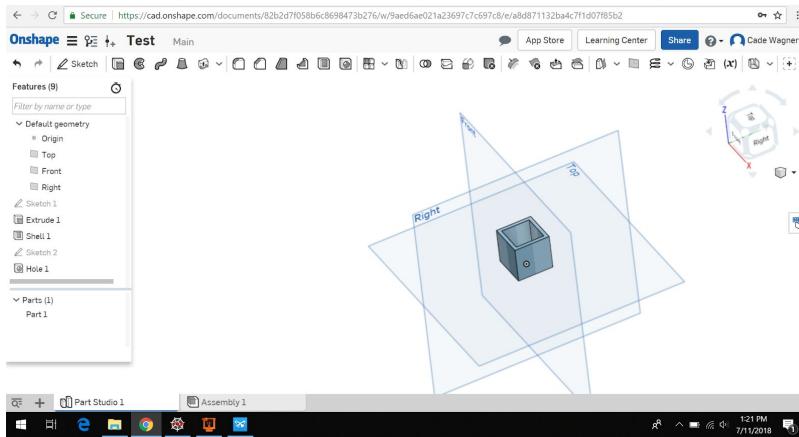


Save the file.

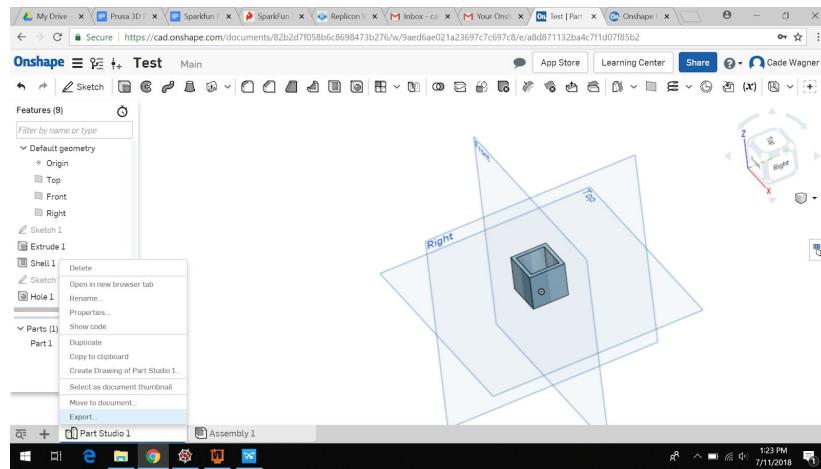


How to save your project in OnShape.

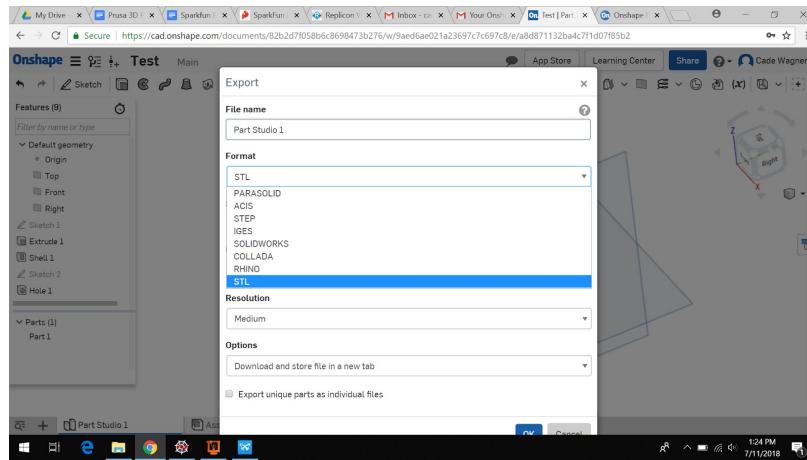
When you open your file in OnShape it should look like this.



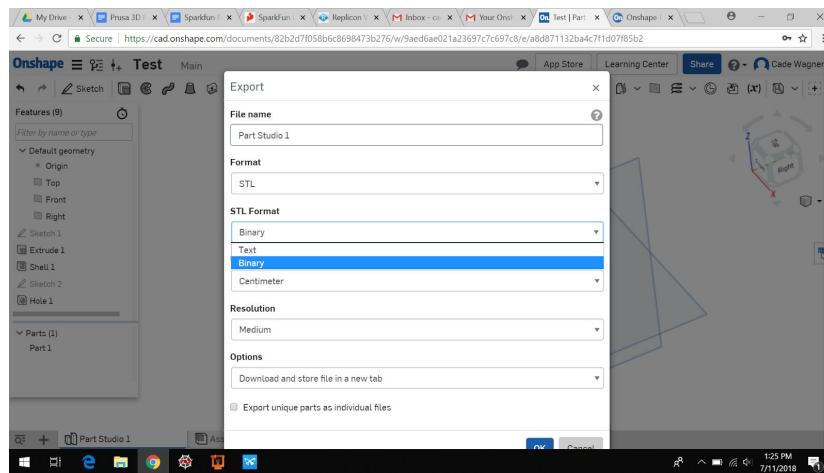
Right click on the tab that you would like to print in the bottom left hand corner and select the export option.



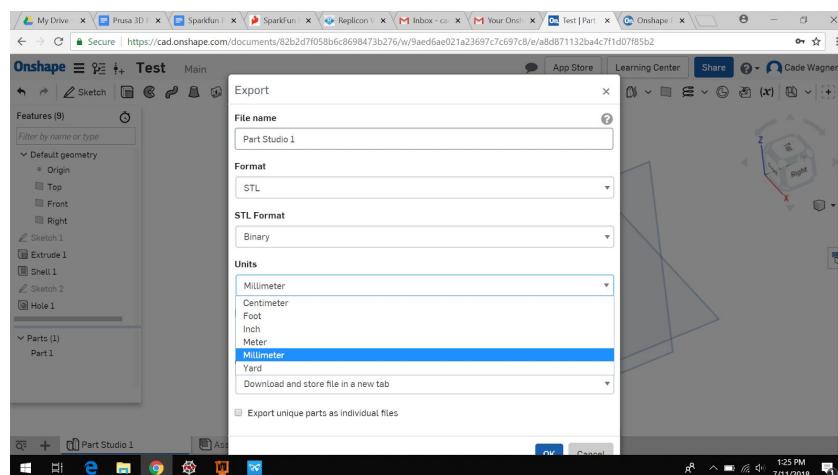
Once the window pops up change your file format to stl.



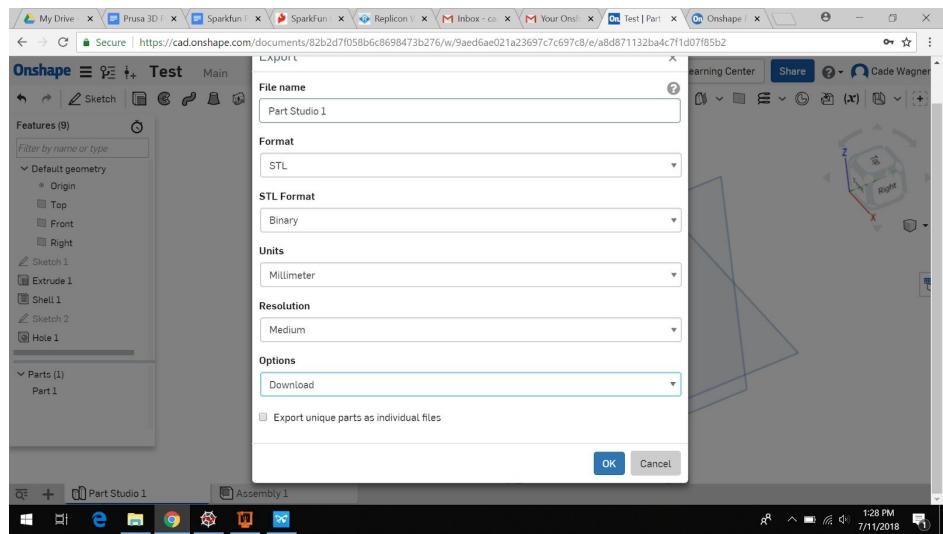
Make sure that the stl format is set to binary.



The next thing is to set the units. If you made your drawing in inches set the units to millimeters and vice versa. I know that seems weird but this will work when the file is imported into Prusa Control.



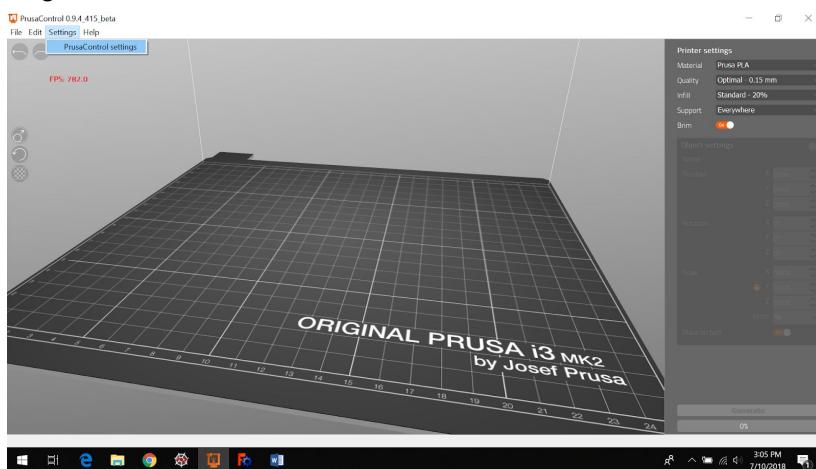
Make sure that your resolution is set to medium unless you need high quality, and that under options you have selected download. Hit ok to download the file onto your computer.



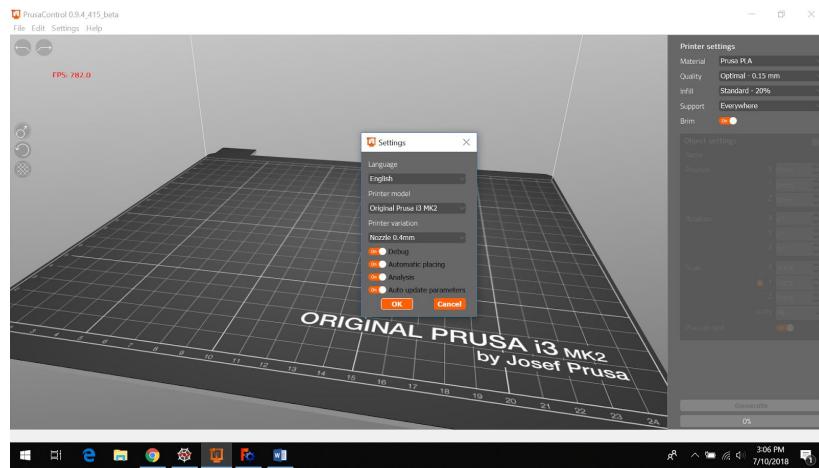
Now that you have your file saved it is time to compile it.

Open up the application PrusaControl.

If this is your first time using PrusaControl, click on the settings menu, then select the PrusaControl settings tab.



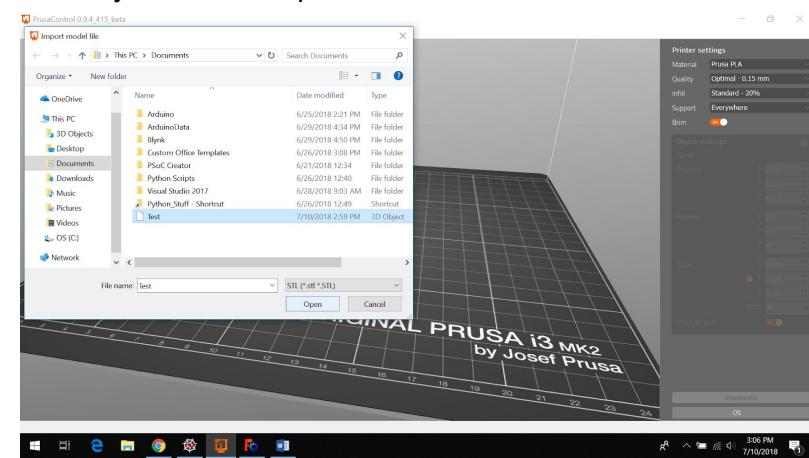
Set your settings so that they look like this.



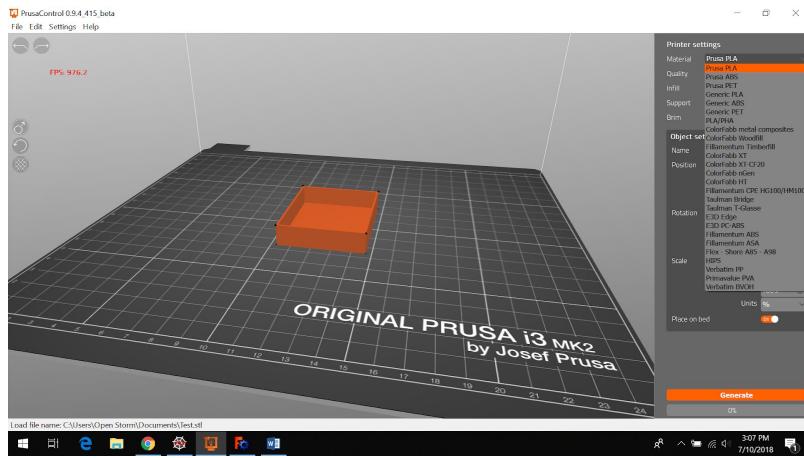
Then click on the file menu and select the tab called import model file.



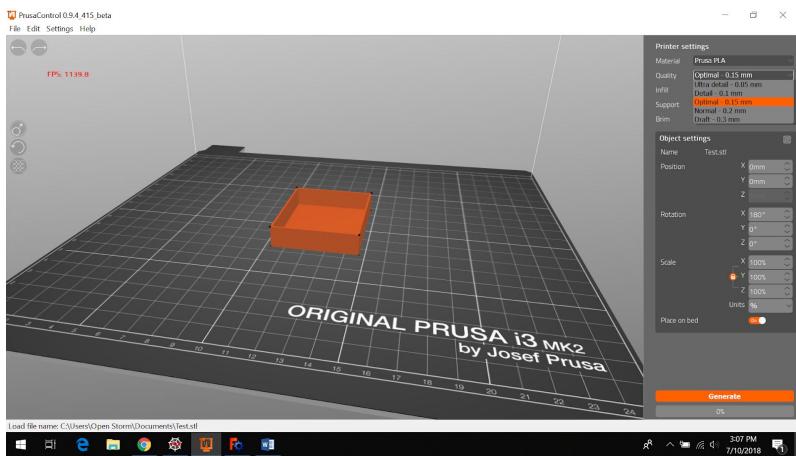
Select the .slt version of your file and open it.



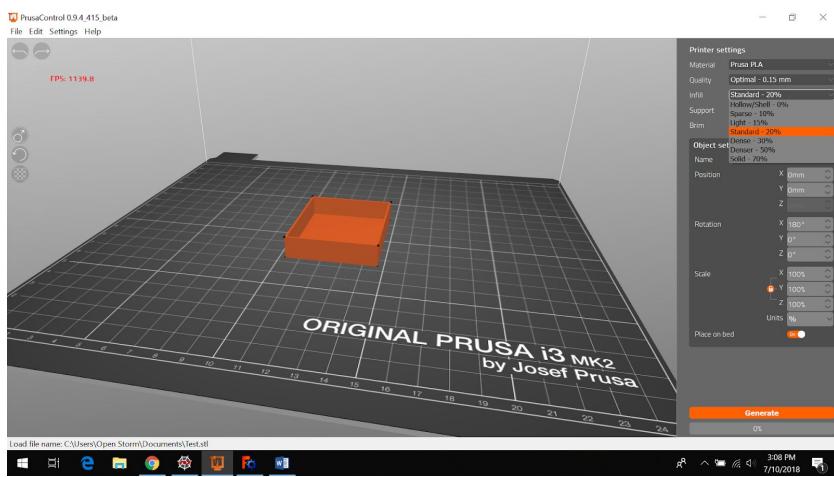
Make sure that your material is set to the correct type, it should generally be Prusa PLA unless you have changed the spool on the machine.



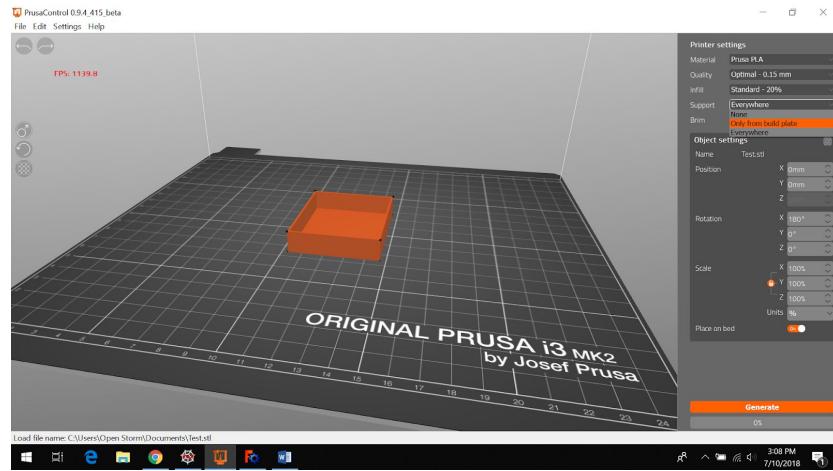
Make sure that your quality is set to the correct level, it should generally be set to optimal (0.15 mm) unless told otherwise.



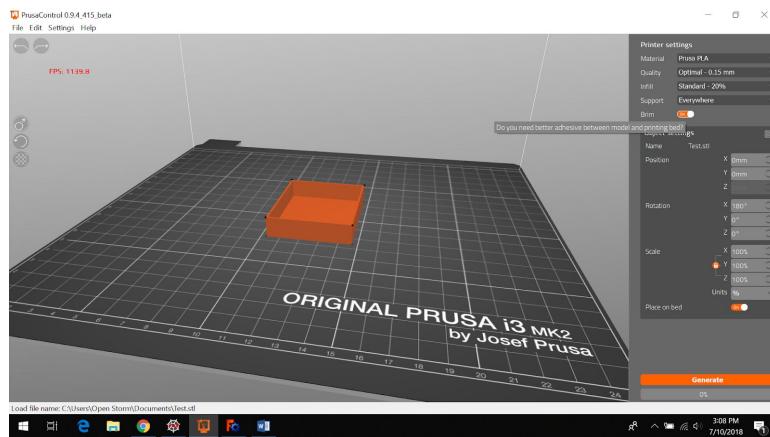
Make sure that your infill is set to the correct amount, it should generally be standard (20%) unless instructed otherwise.



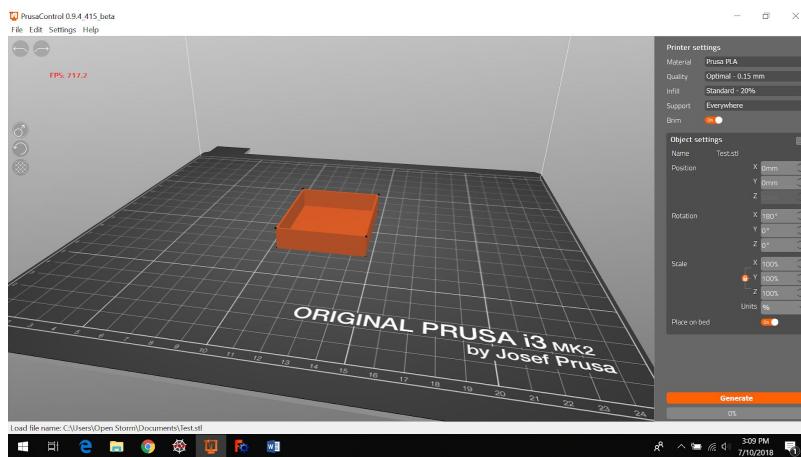
Make sure that your support is set to only from build plate unless it is an overly complex structure.



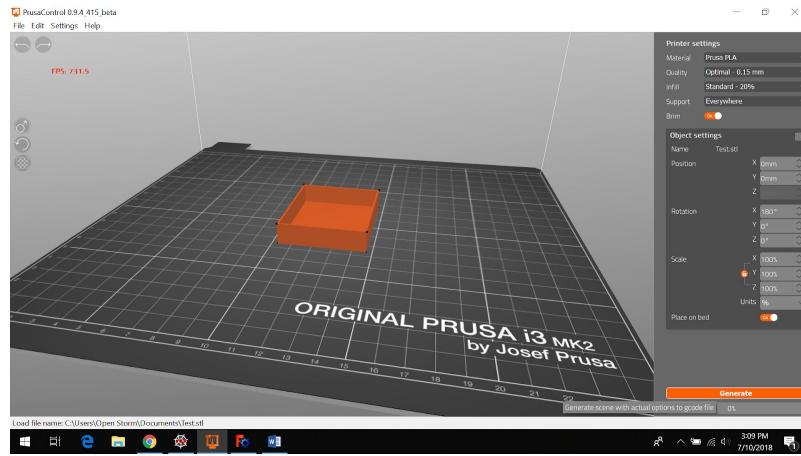
Make sure you set your brim to on. This will put a thin layer around your object which will help it stick to the build plate.



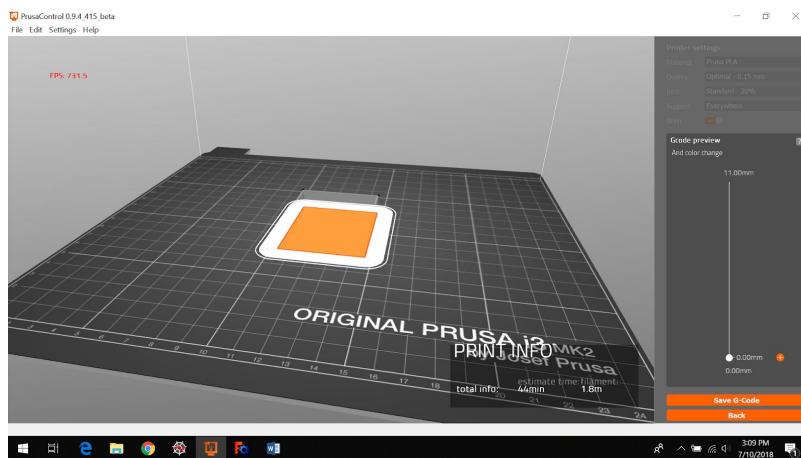
You can use the object settings to adjust the position, orientation, and scale of the object on the print board.



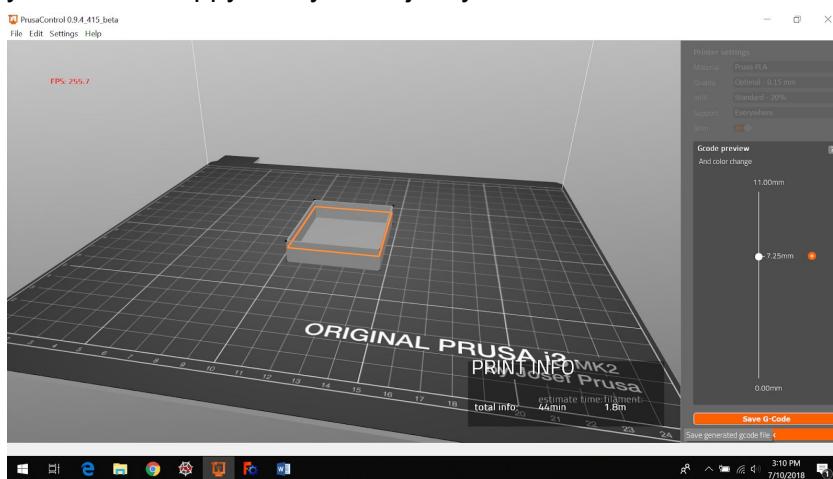
Once you have finished adjusting your object settings, click the generate button in the bottom right hand corner.



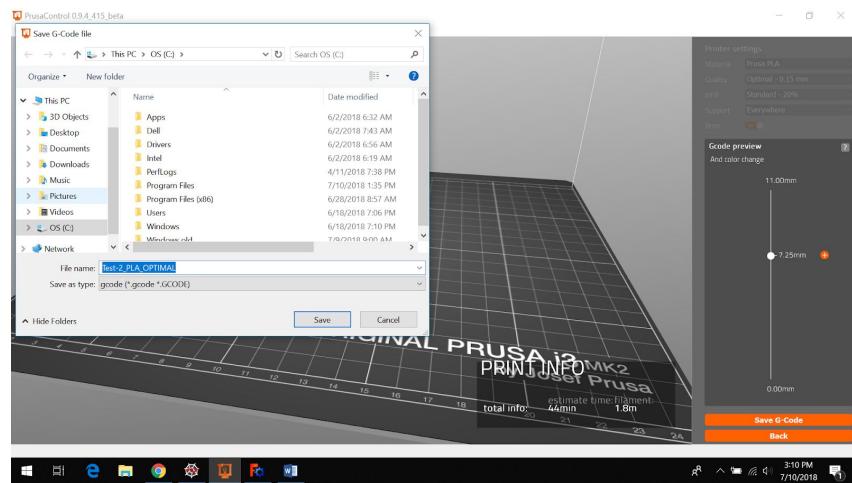
You can then look through the object's profile by adjusting the slider. The orange represents your object, the green represents supports, and the white represents the brim.



Once you are satisfied with your object, click on the generate G-code button in the bottom right hand corner. If you are not happy with your object you can click the cancel button to restart.



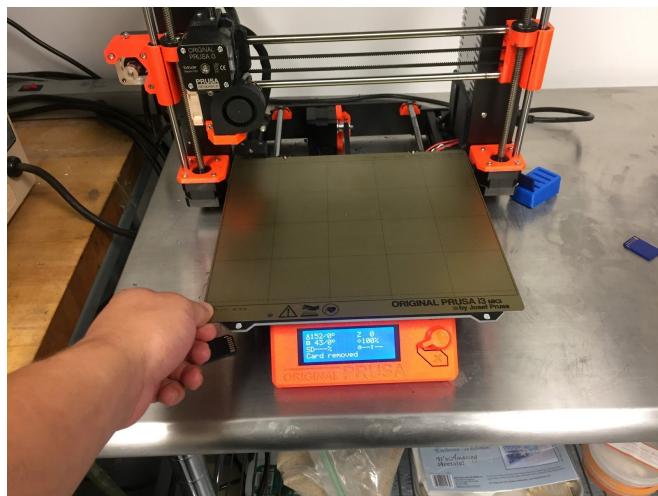
Then save the G-code file to an SD card.



Eject the SD card from your computer and take it over to the 3D printer.



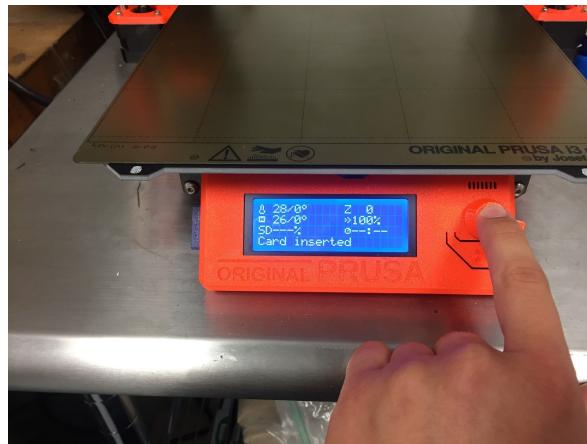
When you first approach the Prusa printer make sure that the gold colored build plate is what you see, not a black plate.



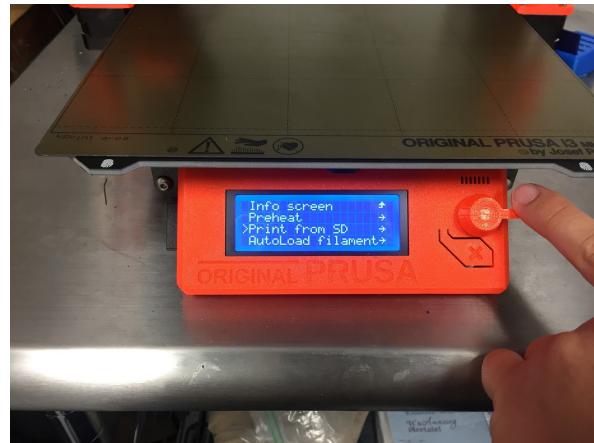
Insert the SD card into the left side of the 3D printer display. The exposed metal terminals of the SD card should face you when you insert it into the printer.



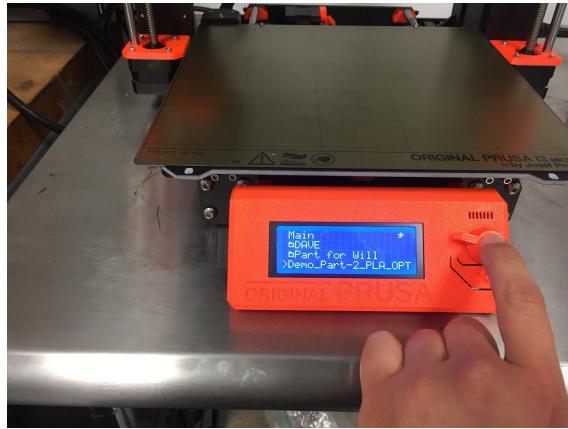
Once the SD card has been plugged in, a loading bar will appear on the screen. Once the SD card has finished loading, press the knob.



Turn the knob to scroll. Scroll down to print from SD and press the knob to select it.



Use the knob to scroll through the list of files and once you have found yours press the knob to select your file.



The following screen should then appear indicating that the printer is heating up.



Hit the knob again to pull up a list of settings. Scroll down to tune and press the knob again to select it.



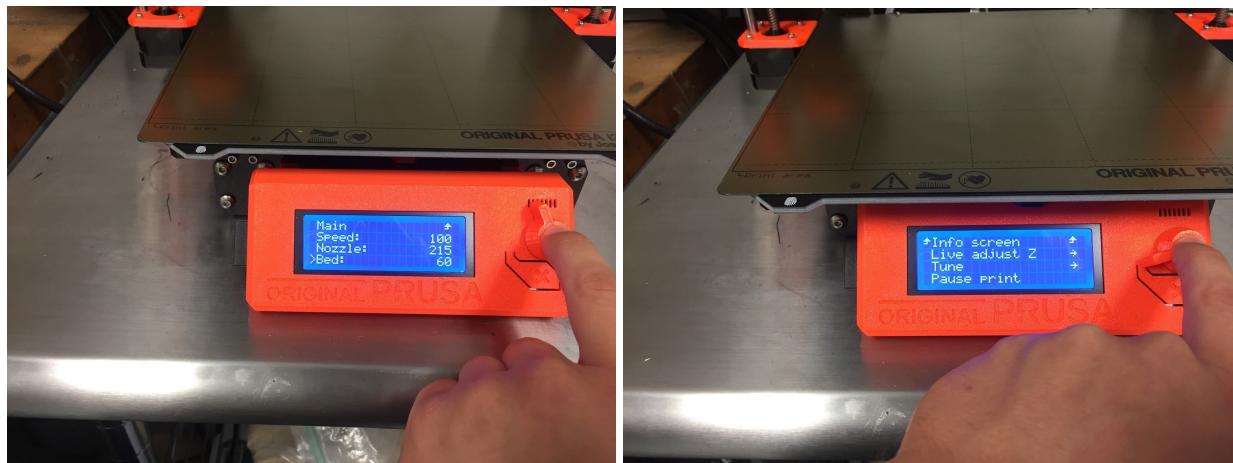
Then scroll down to the option called bed and select it by pressing the knob.



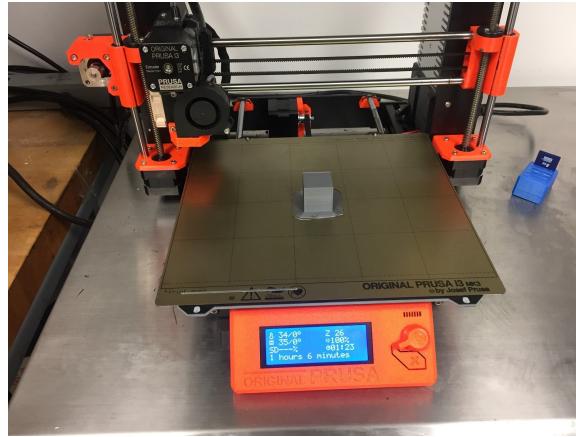
Once you have selected the bed option turn the knob to set it to 65. Once you have the bed set, press the knob again to lock in that setting. (This bed setting is specific for using Prusa PLA).



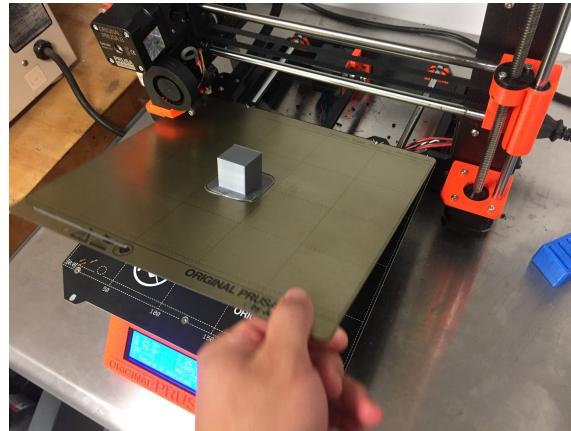
Select main using the knob. Then select info screen to get back to the information screen of your print.



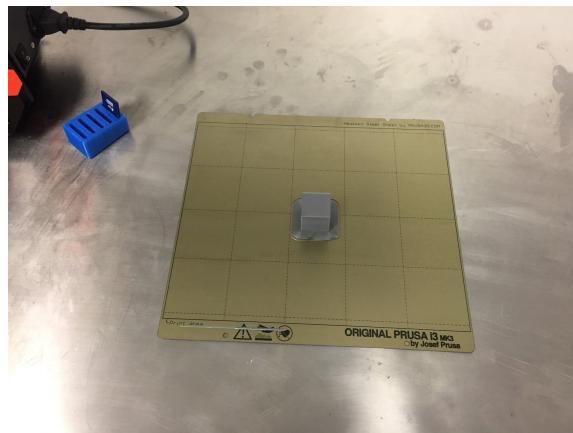
Now wait for your print to complete, you can tell that your print is complete when the printer returns to the far left corner.



Once your print is complete carefully remove the magnetic print plate by grabbing it in the corner.



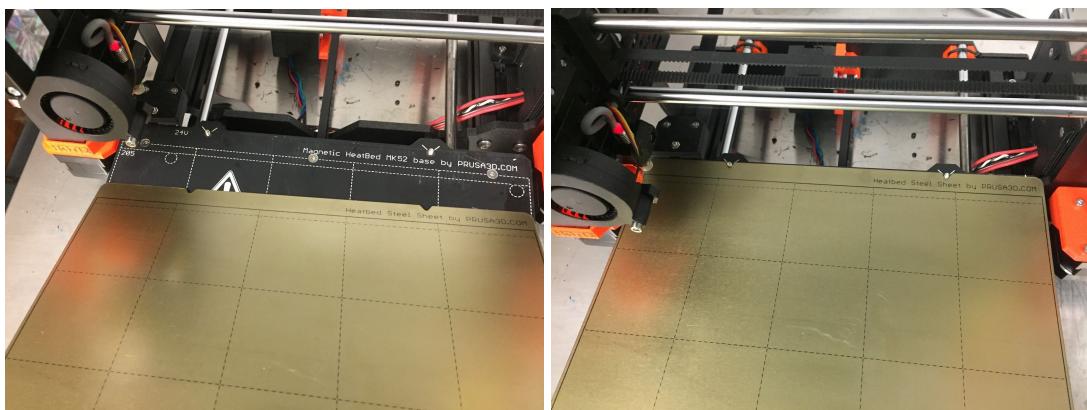
Set the magnetic print plate on the table for 30-60 seconds to allow the heat to flow out of the piece and the print plate.



Flex the magnetic print plate so that your piece pops off. If there is any left over residue on the build plate, use rubbing alcohol to get it off. Make sure to wipe off the build plate when you are done cleaning it.



Once you have taken your part off the build plate, make sure you put it back. There are two small notches on the back of the build plate, and two pins that they will align with.



If you ever need to cancel a print once it has started press the X button below the knob.

