

# Tinker Tuesday Project - 3D Printing Basics

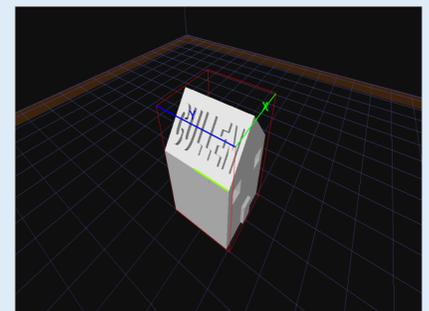
1. Select a file to print from Thingiverse.com or another website with downloadable 3D print files, or export a design you have made using Tinkercad or another 3D design program. You'll need a file with the extension .STL.



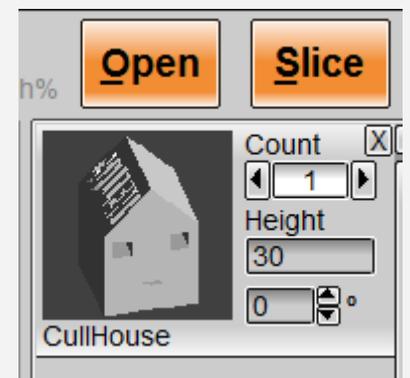
2. Open the KISSlicer program. Click the "Open" button near the top right corner of the program and select the .STL file you would like to print.



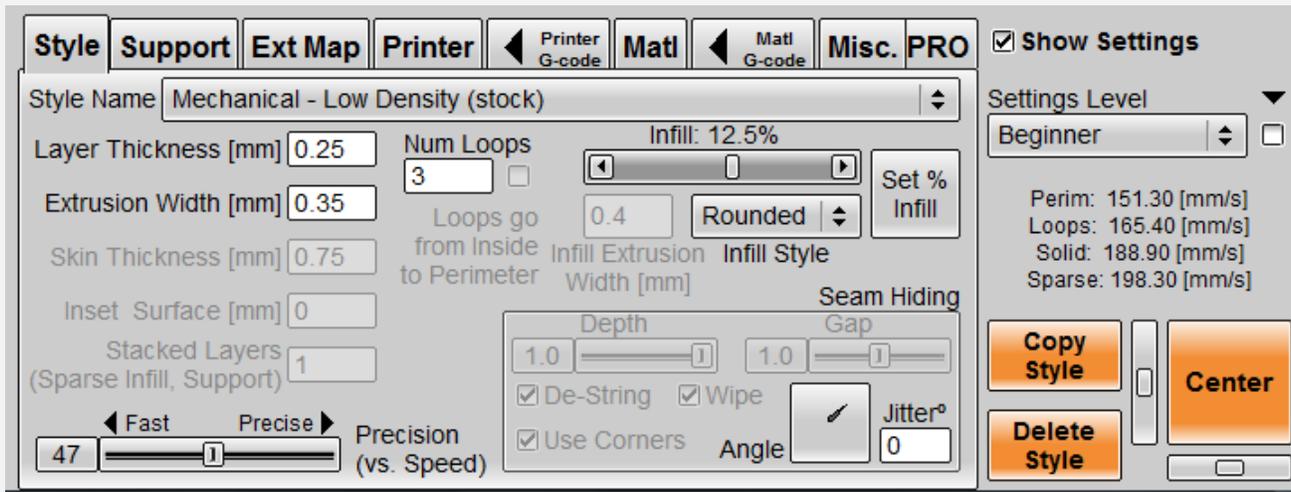
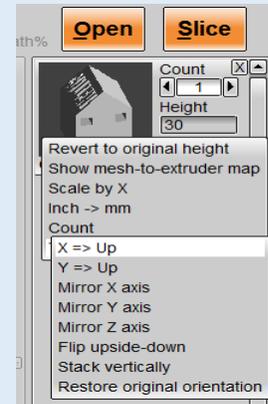
3. Your object will be imported into KISSlicer. You can zoom in or out with the scroll wheel on your mouse. You can rotate your view around the object by holding down a right click and moving the mouse.



4. A thumbnail image of your object will be shown in the panel on the right side of the program. The height is displayed in millimeters and may be changed to your desired value. If you hover the cursor over the thumbnail image, you'll be shown the X, Y, and Z dimensions (in millimeters) of your object.

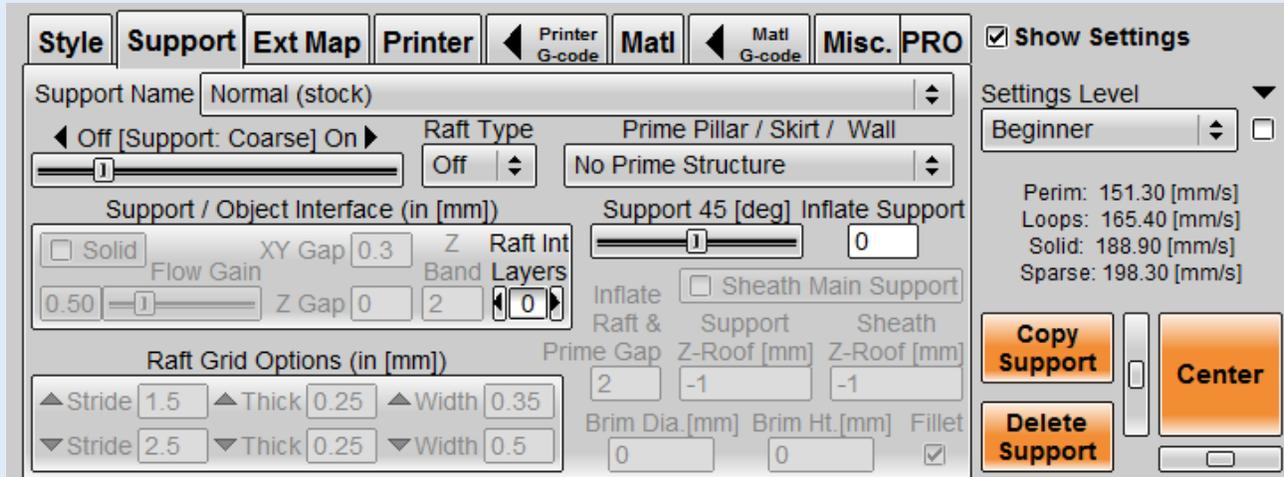


5. Right clicking on the thumbnail image brings up a menu with additional options. “Scale by X,” which allows you to scale your object up or down in size, and “X=>Up” and “Y=>Up,” which allow you to rotate your object along the X or Y axis, are the options you’ll likely use the most.



6. Below the display of your object is a menu area with multiple tabs. Each tab contains a different category of settings. While there are many settings that can be adjusted, for a basic print, you need only concern yourself with a few. On the “Style,” tab, none of the settings necessarily need to be changed to ensure a successful print. One you may consider changing is the “Infill” setting, which determines how solidly filled your printed object will be. A higher infill percentage results in a more solid object, but it will also increase the weight and print time.

You can increase the precision of your print with the slider in the bottom left area of the menu, but doing so can increase print time significantly.



7. There are a couple of important settings to consider on the “Support” tab, the first of which is “Support Name.” If your object has any overhanging parts (e.g., a dragon with wings), it will need supports in order to print successfully. Select “Normal (stock).” The slider below the “Support Name” menu determines the density of the support material that will be printed. In most cases, “Coarse” will work well, but you may wish to move the slider to the right to print finer supports for more complex objects. If your object doesn’t have any overhanging areas and you don’t need supports, select “None” from the “Support Name” menu.

The other setting to consider is “Raft Type.” If your object has a flat or mostly flat base, it doesn’t need a raft. If your object’s base consists of thin parts (e.g., the legs of an animal), select the raft type “Grid.”

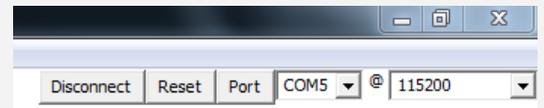
8. Once you have adjusted all relevant settings to your liking, click the “Slice” button in the top right corner of the program. Depending on the size and complexity of your object, this may take up to a few minutes. Once the slicing is complete, your object will display as green.



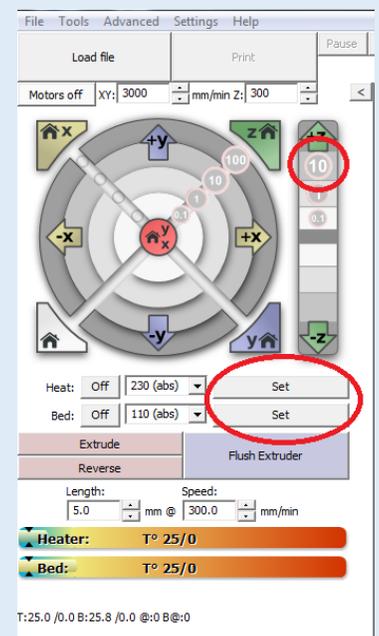
9. Click the “Save” button in the top right corner of the program. Select a location (the Thaw Space is preferred) to save your file, which will be in the .gcode format.



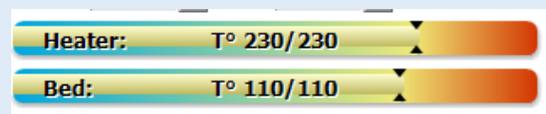
10. Open the Pronterface program and look in the top right corner of the screen. If an option to “Connect” is available, click it to connect the printer. If “Disconnect” appears, the printer is already connected.



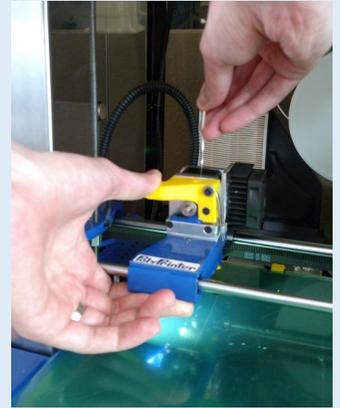
11. The menu on the left side of Pronterface is where the printer can be controlled and adjusted. If you would like to change the filament color in the printer, the buttons in this menu will allow you to prepare the printer to do so. If the printer extruder is very close (less than an inch) to the print bed, click the “10” icon below the “+Z” icon a few times. This will raise the extruder away from the bed. Then, click both “Set” buttons. This will heat the extruder and the bed to the temperatures necessary for printing.



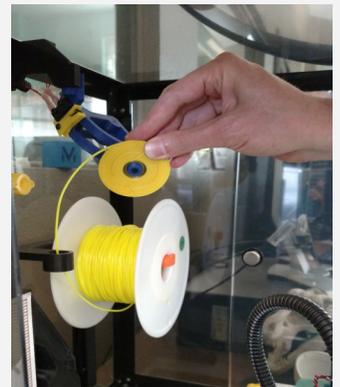
Once the temperature of the extruder (labeled “Heater”) reaches 230 degrees, the filament may be changed.



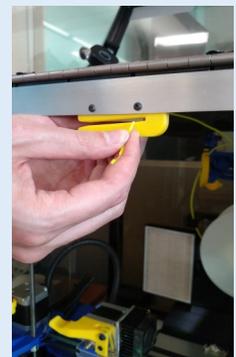
12. While pressing your thumb down on the yellow portion of the extruder and placing your fingers on the blue portion of the extruder for support, gently pull the filament out. You should feel little to no resistance. Remove the filament spool from the printer.



13. Add the new filament spool to the machine by placing it on the arm in the back right portion of the printer. Be sure to load it so that the filament comes up from underneath the spool. Feed the filament through the black loop and the yellow pulley (the pulley opens to make this easier).



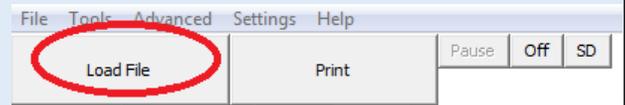
14. Use the yellow cutter located on the underside of the printer's top side to trim off about a half inch of filament. This will make the filament easier to load in the extruder.



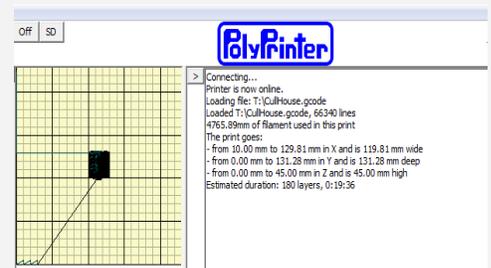
15. Position your hand on the extruder in the same fashion as you did in step 12 and push the filament through the extruder. You should see filament begin extruding from the nozzle. Continue pushing filament through until the color of the extruding filament changes to the color you have just added. Remove the extruded filament from the print bed with a tool (do not use your hand).



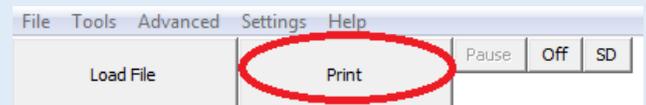
16. In Pronterface, click the “Load File” button in the top left corner. Select the .gcode file of your object that you saved in step 9.



17. After your file is loaded, on the right side of Pronterface, you will see an estimate of the amount of filament that will be used and an estimated print duration. The duration estimate is overly optimistic; prints usually take at least 50% longer than the estimate.



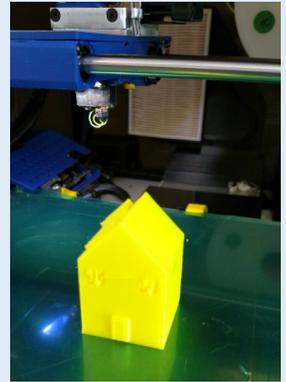
18. Click the “Print” button in the top left corner. Once the extruder and print bed reach the proper temperatures, printing will begin.



19. Depending on the size and complexity of your object, your 3D print job can take anywhere from 15 minutes to several hours to complete. You are not obligated to remain with the printer until your job is finished; you may come back and pick up your object at a later time.



20. Once your print is complete, you may remove it from the print bed after the bed temperature reaches below 70 degrees. Check the temperature gauge in Pronterface to determine when the print bed is cool enough. In most cases, objects can be removed with your hands, but the scraper tool may be used if the object sticks.



21. If your object printed with supports, you may opt to remove them at the library (there are tools available) or later at your convenience. Ask library staff to weigh your printed object and prepare a payment slip for you. The cost of 3D printing at BPL is 10 cents per gram. You may pay at the circulation desk.

