How to operate the Afinia printers (the little red ones)

1. Turn on the printer using the switch on the back of the printer.
2. Launch the Afinia 3d version 2.12 software on the computer.
3. Go to “3d Print” in the top left of the Afinia 3d window, and click “Initialize”.
4. Watch the printer carefully, if it shudders or clunks during initialization, see trouble shooting.
5. Go into the “Maintenance” option in the “3d Print” menu.
6. **SKIP THIS STEP IF FILAMENT IS ALREADY LOADED.** If there is no filament loaded, select “Load new spool” and select the material that you are loading (ABS or PLA) then hit okay.
7. Select “Table Heat 1hr” to start heating the bed. The bed will heat to the proper temperature depending on what material is loaded in step 6.
8. While the table heats, select “Extrude” to heat the nozzle and to make sure that filament is extruding correctly, If the filament does not extrude smoothly, make sure that there is no material hanging off or melted to the extruder hot end. If there is, use a pair of needle nose pliers to remove the material while the nozzle is heated (to heat the nozzle, simply select extrude from the maintenance window and watch the temperature readout on the lower right corner of the software).
9. Go into the “3d Print” menu again and select “Platform Calibrate”
10. Set the nozzle height to 134.76 (or to the point where a sticky note folded in half will JUST fit between the nozzle and the print bed). By using the arrows in the “Move platform to nozzle” section of the “Platform Calibrate” window. Then click set nozzle height once you are happy with the height of the nozzle, then press apply/save current values.
11. Click “Open” in the top left of the Afinia software and navigate to the .STL file you wish to print and select it.
12. The Afinia software will automatically place your object in the build area **HOWEVER** the software is not smart. Make sure that the orientation makes sense for your object and that it is actually touching the build plate in the software, if it is not, you will end up with nothing but a mess of plastic. To adjust the position of the item, select move, rotate or scale from the top tool bar of the Afinia software, then enter the amount you want to adjust the thing you just selected and then select the axis you wish to adjust it on. (for scale, you select scale, then select the amount you wish to scale it by, the press the scale button again).
13. Once you are happy with the position of your piece, check the bottom right of the software at the info box that is switching between nozzle and platform. Make sure that the Platform is at least 97 degrees for ABS and 45 degrees for PLA before you print.
14. Click print in the top toolbar.
15. Make sure that the nozzle height is the same in the print window as it is in the platform calibrate window. If it is not, close the print window and re-do step 10.
16. In the print window, select preferences and set your z resolution (the lower the number, the higher the quality but the longer the print.). 0.25 is a good middle of the road for quality vs time.
17. In the same print window, select your fill options. The more lines in the option you choose, the denser the fill. **MOST PRINTS DO NOT REQUIRE MORE THAN THE BARE MINIMUM FILL.** If you want the print to be hollow with no fill, select shell, and if you are printing a vase type print, select surface.
18. In the same print window, select your support options. The wider the angle the more supports it will print The density will adjust how large those supports are while the space will define how much distance is between each support. The area will determine the minimum size an area needs to be for support to be generated, i.e. if it is set to 3mm2 it will not print support for areas under 3 square millimetres. The stable supports toggle will make sure that whatever supports it does generate are structurally stable by them selves, **WARNING - USING THIS SETTING MAKES REMOVING SUPPORT DIFFICULT AND MAY RESULT IT DAMAGING YOUR PRINT WHILE REMOVING, USE WITH CAUTION.**
19. Once you are happy with the print preferences, click OK and then double check that the bed is at the right temperature and that the nozzle height is what you had set it to, if you are satisfied with what its showing you, Press OK to start your print.
20. Watch the first few layers (about 3 minutes) of your print to make sure that the print has started successfully (no lifting off the bed, no printing into thin air, no nozzle jams) if something goes wrong, stop the print using the software or by turning off the printer and start again at step 1. If nothing goes wrong in the first few layers then sit back, relax and wait for the print to finish!
21. When your print finishes, remove the print bed from the printer by twisting the spring clamps on the front of the printer and then slide the build plate off the printer.
22. Remove your print from the build plate (You may have to use a fair bit of force to get it to separate from the build plate, ask for the scrapper if you’re having issues removing it.)
23. Clear the build plate of any debris left over from removing your print, and then replace the build plate on to the printer.
24. Done!