Afinia H480 Training

By: David Cardarelli
Version 2.2

Procedures

1. Note the time, and power the machine on using the rear power switch
2. On the PC, select the Afinia 3D software
3. Navigate to the "3D Print" menu, and select "Maintenance" and leave this window open
4. Back at the machine, depress and hold the front "Initialize" switch until the machine beeps.
   a. The machine will go through the initialization, and the light will change from red to green.
   b. The table will be sent out for you to load/unload the perf board if necessary.
   c. This will also report the temperatures of the machine to you
5. Ensure that the FR4 is properly affixed. It simply needs to be snug, not tight. The allen bolts are merely to hold the table in place.
   a. If removing the board, unscrew each of the 3 allen bolts (2.5mm), and gently lift the front edge and slide it away. The springs at the rear will release.
   b. To install, slide the back edge (with the notch) in, and place the board down.
   c. Ensure that the springs are not pinched under the board.
   d. There are 2 allen bolts along the right side, and a binder clip. Secure these as snug, NOT tight.
6. On the "Nozzle & Platform" section of the "Maintenance" window, click "Center"
7. Once the table is centered, enter "Nozzle Height Value" of 125mm (for a slight margin of safety) in the fillable window on the "Nozzle & Platform" section of the "Maintenance" box.
8. Click "To:"
   a. This will elevate the perf board (FR-4) to the 125mm position on the Z axis.
   b. We need to get the perf board base to be 2mm from the nozzle.
   c. Using a 2mm allen, laying on one of its flats, we can gauge the separation.
   d. Continually raise the "To:" height in small SAFE increments until you near a 2mm gap in the center of the board, and then raise it in .1mm increments, or as necessary to reach the 2mm gap.
9. Once we have the center height adjusted. We need to select each of the 4 corners to ensure that the table height at each corner is 2mm as well.
   a. Since our platform has bolt heads on it from user damage, we MUST move the nozzle in a specific order to avoid impacting the nozzle.
   b. Select "NL" and check the gap
   c. Select "NR" and check the gap
   d. Select "FL" and check the gap
   e. Select "FR" and check the gap
10. If the gap is not even across the board, there are three thumb screws in the center below the table.
    a. Tightening these will lower the perf board, and INCREASE the gap. Loosening them will of course raise the board, and DECREASE the gap.
    b. I suggest adjusting these with the kinked elbow screwdriver in 1/16th of a turn increments.
11. Once we're sure that the table is leveled correctly at 2mm, we need to manually set our nozzle height to be point two millimeters
    a. To do this, enter the appropriate value into the "To:" window, and send the table to that height
    b. Using our feeler gauge, check the gap at the center, and adjust as needed
    c. Once the center is adjusted, click "Set Nozzle Height" on the "Maintenance" window
12. Once our table is 100% level, and our nozzle height is manually set, select "Table Heat 1hr", send the table "To Bottom" and "quit" the "Maintenance" window.
13. Navigate to open your .STL file using the open icon
    a. You can "unload" a design if you want to clear the digital board
14. Navigate to "3D Print" > "Setup"
    a. Here we can make adjustments to our Z resolution, our Part, our Fill, and our Support.
    b. Adjusting the Z Resolution
       i. Increasing this can save you significant amounts of time, and provide a greater margin for error
       ii. Decreasing this will of course do the opposite, but it will also provide you with a finer print quality. Our nozzle is a .40mm nozzle diameter.

       c. Adjusting the Part settings

- i. Angle will determine the angle relative to your model that the rectilinear infill will be printed.
- ii. Surface will determine how many solid layers will make up the surface of your model before it moves into printing the infill

       d. Fill settings will determine how "solid" your models infill will be. The values here are 50%, 25%, 15%, and 12%

- i. Shell will print a hollow model
- ii. Surface will print a single layer footprint of your model (this should only be used when using Flex Board)

       e. Support determines how and where support structure will be printed for your model

- i. Dense determines how dense your support structures are. More layers = stronger, denser support.
- ii. Space will determine the spacing between support structures. It is an advanced setting, and will be material specific.
- iii. Area> is an on/off switch for support. If the area where support would be needed is less than this in square millimeters, no support will be printed. If it is greater than the specified area, then support will be printed
- iv. Angle< is also an on/off switch for support. If the angle between the model and the platform surface is less than the specified angle, then support will be printed. If it is greater than that, then it will not be printed.
- v. Stable support will print stable support for your model. This can be useful for tall supports, but it will make them more robust, and thus more difficult to remove.

15. Once you have all of these settings to your liking, navigate to "3D Print" > "Print Preview"

       a. Here we see a few additional options to select

- i. UnSolid Model will examine your manifolds on your model. If this box is checked, your model has defects. You can leave this box checked, and the software will attempt to repair them for you and print regardless, but I would suggest that you repair your model yourself.
- ii. Thin Wall – This option will only be used if you're printing an item with a thickness of roughly .75mm or less. Typically, the printer will ignore these details as they are too fine to print, but with this selected, it will print those as they are.
- iii. No Raft should only be used if using the Flex Board. Even then, a raft may be beneficial
- iv. Pause at would enable you to pause your print at a specified layer height.
- v. Quality determines the SPEED of the print, which has a MINOR effect on quality
- vi. Heat platform after finish will heat the dang platform after the print finishes!

       b. Once you've selected your options, select "OK".

- i. This will send your layers to the machine for an estimate on material costs and time to completion. Select "OK"

16. Once all of the settings, and the timeframe are to your liking, navigate to "3D Print" > "Print"

       a. Ensure that your options remain as you set them, and select "OK"

17. You should now see the software sending your layers to the machine in the very lower left status bar.
18. A box will come up confirming your material use and timeframe, select "OK"
19. You will now see the nozzle, and bed heating in the lower right status bar
20. You will also notice a rough estimate on the time
21. Navigate back to the "Maintenance" window to monitor the temps and access on screen printing controls
22. When your product is done printing. The printer will automatically eject the table to the down and forward position.
23. Remove the perfboard with your object still attached using a protective glove as the surface will be over 90 degrees Celsius.
24. Take the perfboard over to the workbench, and use the scraper of your choice to gently remove your printed object.
25. Once removed, scrape the board smooth with the scraper, and reattach it to the base table.
26. After the machine has cooled for a few minutes hold the "Initialize" switch on the machine for 1 second again and the machine will retract the table, and reset itself to its starting position.
27. Turn the machine off at the rear power switch, and be sure to log your time.

Afinia H800 Training

By: David Cardarelli
version 1.1

Procedures

1. Note the time, and power the machine on using the rear power switch
   a. You should see the AFINIA logo light up orange
2. On the PC, select the Afinia 3D software
3. Navigate to the "3D Print" menu, and select "Maintenance" so that you can determine the temperature of the machine after initialization.
4. Back at the machine, carefully open the upper lid to reveal the button guide
   a. There are 3 multi-function buttons located on the right side of the machine, long press the power button which will "initialize" the machine.
   b. The machine will emit a series of beeps, go through the initialization, beep upon completion and the AFINIA will change from orange to green.
   c. The table will be positioned for you to easily load/unload the FR4 board
5. Once you have confirmed that the temperature is suitable for handling, you may "quit" the maintenance window. Do not use the red X!
6. This Afinia uses two types of FR4
   a. Perf Board – Just like our H480.
      i. Excellent for first layer adhesion, but will require you to print a raft to sacrifice
   b. Flex Board
      i. Excellent for a smooth surface print, but will require additional materials for first layer adhesion.
7. Select your FR4, and attach it to the machines aluminum heating plate
   a. There are 13 screws that must be lined up properly PRIOR to locking the FR4 into position.
   b. Once we're certain that all 13 are aligned, we can PINCH the FR4 and the aluminum plate together at the front two corners. This should cause the board to slide into place.
   c. Once it has snapped into position, we should visually verify that each of the 13 screws has seated correctly. If not, our platform will not be level, and our print will be ruined.
8. Back at the PC, select "3D Print" > "Nozzle Height Detect"
   a. This will send our base towards the nozzle to determine its height along the Z axis
   b. Once it makes contact, it will ask you to select your "Platform Type", choose whichever you're using.
   c. You should receive a number (in millimeters) from the software at this point. Make a note of this number as it will be useful moving forward, then select "OK"
9. Now select the "3D Print" > "Maintenance" window again, and under the "Nozzle & Platform" section select "Center". This will center the nozzle over the platform.
10. We'll need to ensure that our table is level at this point. To do this, we should start by ensuring we have an even 2mm gap around our platform.
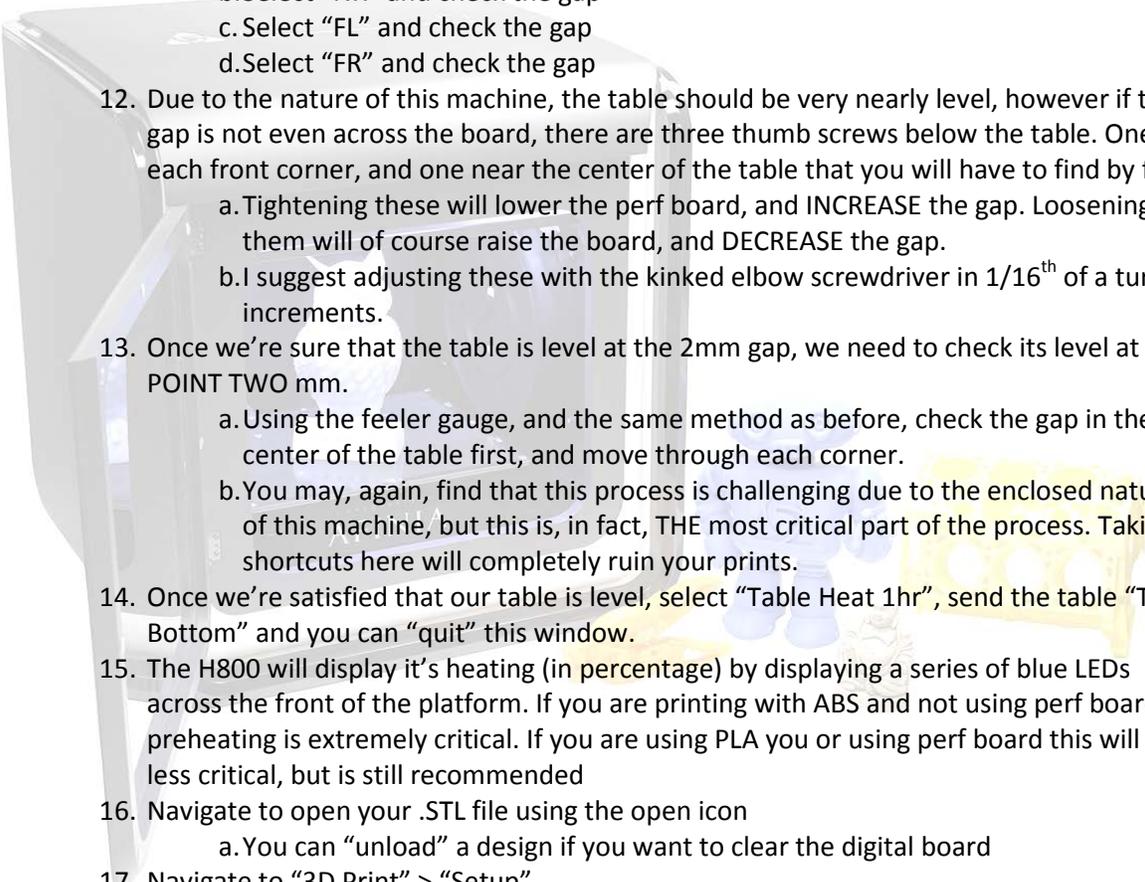
a. Using the number you received from the nozzle height detector, subtract 2, and input this into the "To:" box. Then click "To:" to send the table to this height.

b. Using a 2mm allen, laying on one of its flats, we can gauge the separation.

c. Continually raise the "To:" height in small .5mm increments until you near a 2mm gap in the center of the board, and then raise it in .1mm increments, or as necessary to reach the 2mm gap.

11. Once we have the center height adjusted. We need to select each of the 4 corners to ensure that the table height at each corner is 2mm as well. Because this machine is enclosed, you may find this to be a challenging process. Take your time, and watch where you place your hands.

a. Select "NL" and check the gap

b. Select "NR" and check the gap

c. Select "FL" and check the gap

d. Select "FR" and check the gap

12. Due to the nature of this machine, the table should be very nearly level, however if the gap is not even across the board, there are three thumb screws below the table. One at each front corner, and one near the center of the table that you will have to find by feel.

a. Tightening these will lower the perf board, and INCREASE the gap. Loosening them will of course raise the board, and DECREASE the gap.

b. I suggest adjusting these with the kinked elbow screwdriver in 1/16th of a turn increments.

13. Once we're sure that the table is level at the 2mm gap, we need to check its level at POINT TWO mm.

a. Using the feeler gauge, and the same method as before, check the gap in the center of the table first, and move through each corner.

b. You may, again, find that this process is challenging due to the enclosed nature of this machine, but this is, in fact, THE most critical part of the process. Taking shortcuts here will completely ruin your prints.

14. Once we're satisfied that our table is level, select "Table Heat 1hr", send the table "To Bottom" and you can "quit" this window.

15. The H800 will display it's heating (in percentage) by displaying a series of blue LEDs across the front of the platform. If you are printing with ABS and not using perf board, preheating is extremely critical. If you are using PLA you or using perf board this will be less critical, but is still recommended

16. Navigate to open your .STL file using the open icon

a. You can "unload" a design if you want to clear the digital board

17. Navigate to "3D Print" > "Setup"

a. Here we can make adjustments to our Z resolution, our Part, our Fill, and our Support.

b. Adjusting the Z Resolution

i. Increasing this can save you significant amounts of time, and provide a greater margin for error

ii. Decreasing this will of course do the opposite, but it will also provide you with a finer print quality. Our nozzle is a .40mm nozzle diameter.

c. Adjusting the Part settings

i. Angle will determine the angle relative to your model that the rectilinear infill will be printed.

ii. Surface will determine how many solid layers will make up the surface of your model before it moves into printing the infill

d. Fill settings will determine how "solid" your models infill will be. The values here are 50%, 25%, 15%, and 12%

    i. Shell will print a hollow model

    ii. Surface will print a single layer footprint of your model (this should only be used when using Flex Board)

e. Support determines how and where support structure will be printed for your model

    i. Dense determines how dense your support structures are. More layers = stronger, denser support.

    ii. Space will determine the spacing between support structures. It is an advanced setting, and will be material specific.

    iii. Area> is an on/off switch for support. If the area where support would be needed is less than this in square millimeters, no support will be printed. If it is greater than the specified area, then support will be printed

    iv. Angle< is also an on/off switch for support. If the angle between the model and the platform surface is less than the specified angle, then support will be printed. If it is greater than that, then it will not be printed.

    v. Stable support will print stable support for your model. This can be useful for tall supports, but it will make them more robust, and thus more difficult to remove.

18. Once you have all of these settings to your liking, navigate to "3D Print" > "Print Preview"

a. Here we see a few additional options to select

    i. UnSolid Model will examine your manifolds on your model. If this box is checked, your model has defects. You can leave this box checked, and the software will attempt to repair them for you and print regardless, but I would suggest that you repair your model yourself.

    ii. Thin Wall – This option will only be used if you're printing an item with a thickness of roughly .75mm or less. Typically, the printer will ignore these details as they are too fine to print, but with this selected, it will print those as they are.

    iii. No Raft should only be used if using the Flex Board. Even then, a raft may be beneficial

    iv. Pause at would enable you to pause your print at a specified layer height.

    v. Quality determines the SPEED of the print, which has a MINOR effect on quality

    vi. Heat platform after finish will heat the dang platform after the print finishes!

b. Once you've selected your options, select "OK".

    i. This will send your layers to the machine for an estimate on material costs and time to completion. Select "OK"

19. Once all of the settings, and the timeframe are to your liking, navigate to "3D Print" > "Print"

a. Ensure that your options remain as you set them, and select "OK"
20. You should now see the software sending your layers to the machine in the very lower left status bar.
21. A box will come up confirming your material use and timeframe, select "OK"
22. You will now see the nozzle, and bed heating in the lower right status bar
23. You will also notice a rough estimate on the time
24. Navigate back to the "Maintenance" window to monitor the temps and access on screen printing controls
25. When your product is done printing. The printer will automatically lower the table for you to easily access the FR4
26. **REMOVE** the FR4 from the machine with your object still attached using a protective glove as the surface will be over 90 degrees Celsius.
    a. To do this, pinch the FR4 and the aluminum platform together again and draw back with your thumbs. Remember, our objective here is to remove the FR4 only, and move the platform as little as possible since it is extremely delicate.
27. Take the perfboard over to the workbench, and use the scraper of your choice to gently remove your printed object.
28. Once removed, scrape the board smooth with the scraper, and place it in the appropriate colored bin
29. This will have allowed the machine to cool sufficiently so long press the "power" switch which will "de-initialize" the machine.
30. Turn the machine off at the rear power switch, tidy up, and be sure to log your time.
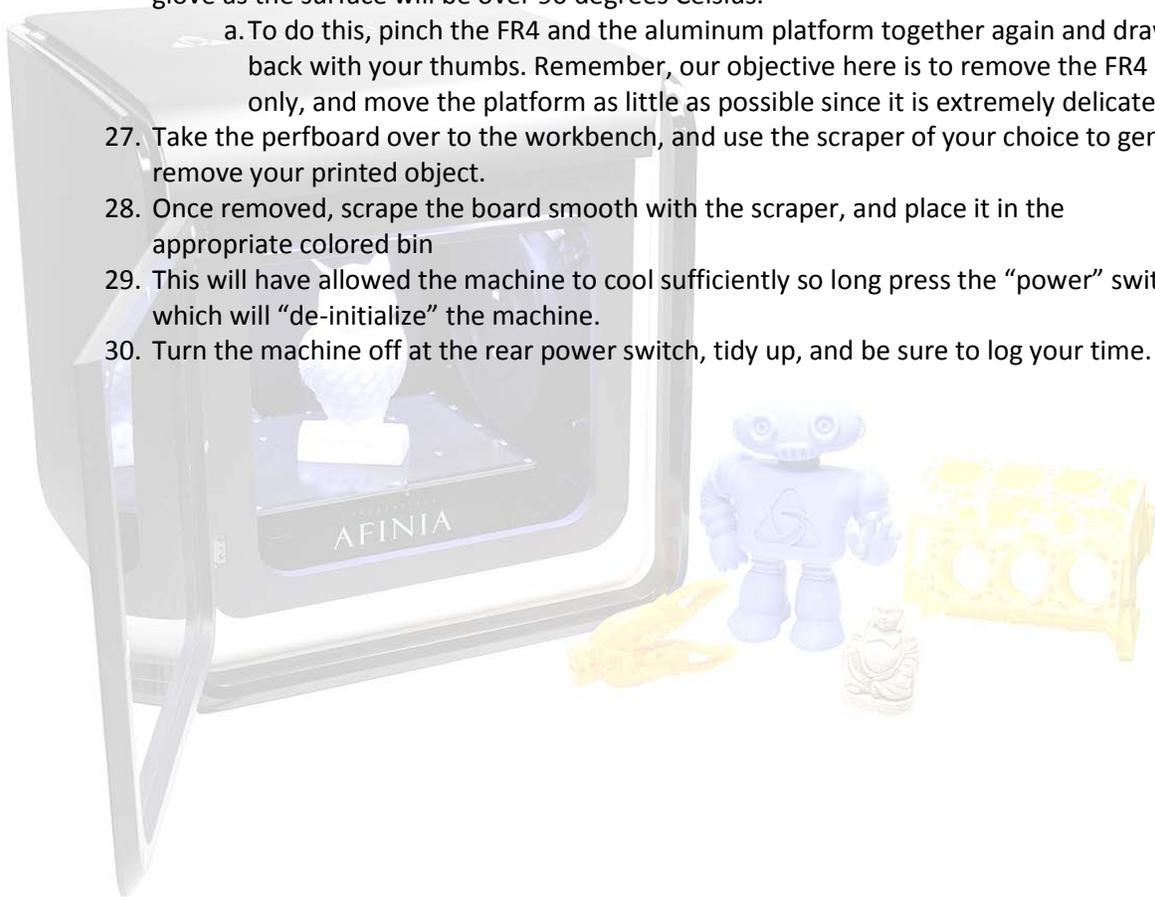
Fusion F306 Advanced Training
Operations Guide

by: David Cardarelli
version 1.0

1. Note the time for logging purposes
2. This machine is secured using a physical locking mechanism. The combination is:
3. Ensuring that the machine is switched OFF, plug the YC-12 end of the supplied cord into the machine on the left side, and plug the other end into an available power outlet.
4. Power the machine on using the power switch located above the YC-12 input
5. Ensure that the bed is clean and free of debris
   a. If necessary, sprits the bed with the water bottle, and wipe it clean with the heavy towels.
   b. Using the machines control box, depress the orange button to bring up the menu
   c. Select "Prepare"
   d. Select "Disable Motors"
      i. This will enable you to freely move the car, x-axis arm, and the z-platform (via the control belt) without damaging the machines sensitive electronics.
   e. The primary adhesion technique for this machine is purple glue stick. It will dry clear, and show up as purple when spritzed with water.
6. This machine is designed so that the table will remain level for many uses. Do NOT attempt to level the table for any reason. If you feel it needs to be leveled, you should contact the Garage Manager.
7. With the z-platform completely clean and dry, it's time to deploy the purple glue stick.
   a. You don't need anything more than a thin layer exactly where your object will print
   b. To apply, hold the stick vertically, and apply all in one direction typically window to door
   c. Less is more!
8. Once complete, manually raise the z-platform to be approximately two inches from the nozzle using the belt, and then select:
   a. "Prepare" > "Home all axes" which will send the car to its FRONT home position, and raise the z-platform to its correct height.
9. With your file sliced to the SD card, insert the card (label down/away) into the left side of the control box
10. Depress the orange button, and navigate to "Card Menu" and select your file. THIS WILL BEGIN YOUR PRINT!

Simplify3D Beginner Setup Guide

by: David Cardarelli
version 1.0

1. Importing and manipulating a model
   a. Select "Import" or simply drag your .stl file onto the virtual build platform
   b. Select "Center and Arrange"
      i. If you have multiple models, this will shuffle them into position
   c. Double click on your model to bring up the "Model Settings" window
      i. This allows us to change the position, the scale, and the rotation exactly like on the Afinia, but with much greater accuracy.
      ii. Be sure to click "save" prior to exiting this window
2. Setting up a process for printing
   a. Ensure that "EIGM Stock Fusion" is showing as an "FFF" process. Double click this to open the "FFF Settings"
   b. To start off, lets "Hide Advanced" if it's showing and work with the basic settings
      i. The profile should be for the Fusion3 F306 Single (modified)
      ii. The "Auto-Configure for Material" option should indicate either PLA or ABS, but if you'd like to print with a different material, we welcome that… but please speak with me prior. Abrasives are not allowed (metal/wood/fiber type filaments)
      iii. Select your option for "Auto-Configure for Print Quality" taking note that these settings *do* have a significant effect on print quality with this machine
      iv. Select your "Infill Percentage" based on how solid you'd like your model to be. For reference, the Afinia's infill settings were as follows:
         1. Solid = 50%
         2. Loose = 25%
         3. Hollow = 17%
         4. Big Hole = 12%
      v. Please consider that this machine uses differing types of actual infill structure. The Afinia only used "rectilinear". For most models on the Fusion, 10% - 20% infill seems to work well.
      vi. A "Raft" is not needed for this machine unless you have a very small footprint, and would like additional bed adhesion (see intermediate section) or if you require a strong foundation to support your part. A raft is still a sacrificial layer on this machine.
      vii. Be sure that "Generate Support" is selected and click "OK"
3. Printing a file
   a. Select "Prepare to Print"
   b. This will bring up "Preview Mode" where you can see your material costs and build time (multiply this number by 1.5x).
      i. Additionally, you can use the colorway, along with the sliders on the bottom of the window to scale through your design and ensure it is setup in a manner you'd like.
   c. Once everything is set, using the USB card reader (which plugs nicely into the left side of the monitor) slide the SD card in (from the bottom, label towards you), select "Save Toolpaths to Disk" and save to the SD card.
   d. From here, transfer the card to the control box on the front of the machine, and insert it facing down

Proceed to the Operations Guide (Document/Page 1)

Fusion F306 Advanced Control Box Operation Supplement

by: David Cardarelli

version 1.0

1. Current temperature of the primary extruder
2. Temperature the primary extruder is set to
3. Current and set temperature of the heated z-platform
4. The current XY and Z position of the extruder nozzle and z-platform
5. The feed rate of the system. Default value is 100%
6. The percentage that the machine seems to think it has completed of your design
7. Elapsed time. We typically find this to be a 1.5 multiplier from what Simplify has indicated
8. The message line

Menu's within the control box

1. "Watch" will send you to the primary home screen. Think of this as the "Maintenance" window from the Afinia
2. "Prepare" is the menu where you'll find:
    a. Motor controllers
        i. Homing
        ii. Stepping
        iii. Disabling
    b. Filament Controllers
        i. Unloading
        ii. Loading
            1. To properly load filament, first snip the tip at a 45°
            2. Using the pull tab, apply slight backpressure and slide the clamp down
            3. With the motors disabled, spin the gear wheel by hand, and ensure that the toothed drum is <u>COMPLETELY</u> free of debris by using the toothbrush
            4. Pass the filament through the lower plastic hole, and insert it into the PTFE tubing until the tip reaches the first zip tie (about two inches)
            5. Lift the pull tab back into place select the "Load Filament" option and follow the screen prompts
            6. Ensure that your file is set to the appropriate type of filament or it will extrude at the wrong temperature
    c. Heater Controllers
        i. Disabling
3. "Control" is the menu where you'll find:
    a. Temperature controllers
   b. **Motion Controllers (Please do not alter these for any reason at all. Ever.)**

Simplify3D Advanced Setup Guide

by: David Cardarelli

version 1.0

1. You can be sure you're starting with a fresh profile by selecting the "Edit Process Settings" button and selecting "EIGM Stock Fusion"
   a. If this profile states in parenthesis that it is "(modified)" you'll want to select the "Fusion3 F306 Single" profile. The software will then ask you if you'd like to save the changes you made to the EIGM profile. Select "NO"
   b. Once you're back on the Fusion3 F306 Single profile, you should see that the stock EIGM profile no longer says modified, so let's switch back to that profile to ensure we don't corrupt our primary process.
2. With the FFF Settings window now open, and your profile set to the EIGM Stock Fusion, select "Show Advanced" to ensure we're viewing the advanced menu
3. ==Please note! Simplify3D is a universal type slicing software. It has many settings for many different machines, and even different *types* of machines. There are settings in here that if mis-managed can cause irreversible damage to our printer. If you are unsure, please ask!==
4. Viewing the "Extruder" menu you will see options for changing the extrusion settings
   a. Overview
      i. Our nozzle diameter is set to a physical limit of .40mm.
      ii. Extrusion Multiplier would only be used in conjunction with a filament diameter check/test. You can find more information in the manual on this process, and if you feel you need to modify it, speak with me.
      iii. Extrusion Width should be set to .45mm. This would get modified elsewhere.
   b. Ooze Control
      i. Retraction - Typically selected will retract filament in-between extrusion passes to help reduce stringing. You can lower the retraction distance (default is 6mm) to help decrease printing time.
      ii. Coast at End - is off by default. This feature essentially relieves the pressure in the nozzle as the extruder finishes a pass. It is designed to prevent blobbing. Typically not necessary on our machine, but may be used at your discretion.
      iii. Wipe Nozzle – Off by default, and almost completely unnecessary. It is used to keep the nozzle primed and properly pressurized between layers. This will add a SIGNIFICANT amount of time to your print. Note: The nozzle WILL wipe at the beginning. That's controlled by a different setting.
5. The "Layer" menu will offer you options for your printable layers
   a. Layer Settings
      i. We can only set layers for one extruder here, because we only have one extruder
      ii. Primary Layer Height – This will default to .20mm (50%). This is the setting to alter your Z resolution
      iii. Top/Bottom Solid Layers – Will determine how many solid layers appear on the top and bottom before using your infill options. More will increase the strength, but add significant time
      iv. Outline/Perimeter Shells – Works the same as top and bottom layers. More layers means thicker skin
      v. Outline Direction – Determines how your part is born

      vi.   Printing Islands Sequentially Without Optimization – Used to control overheating of small parts. Left off for best printing speeds

      vii.   "Vase Mode" will help eliminate any seams in your work by ramping the extrusion rather than stepping it

  b. First Layer Settings

      i.   First Layer Height – Defaults to 40% which works nicely to aid adhesion. This setting can be altered as needed. A good zone to work in is 25%-75% of the nozzle diameter.

      ii.   First Layer Width – Defaults to 100% which is probably the best to use.

      iii. First Layer Speed – Defaults to 40%. Works well. Typically settings of 20% - 50% are used.

  c. Start Points

      i.   Random/Optimize/Specific – Each of these settings will do exactly what you think they'll do. Default is Specific at X0, Y300

6. The "Additions" menu discusses Skirts/Brims, Rafts, Pillars, and Ooze Shield settings. Prior to altering these settings, I would encourage you to read the "Additions" supplement so you're clear about when/which to use.

  a. Skirt/Brim – A skirt and brim are technically the same thing, and differentiated based on how many layers/outlines are selected. This tool is used to prime the extruder prior to extrusion to ensure a smooth flow of plastic. Default is selected on with the following settings:

      i.   Skirt Layers = 1

      ii.   Skirt Offset from Part = 5mm

      iii. Skirt Outlines = 1

  b. Rafts – A raft is a sacrificial layer that can help with minute leveling issues, primary layer adhesion (typically with ABS) or a strong foundation for a model with minimal surface contact. The default settings are:

      i.   Raft Layers = 2

      ii.   Raft Offset from Part = 4mm

      iii. Separation Distance = .10mm

      iv. Raft Infill = 100%

      v.   Disable Raft Baselayers – Selected

  c. Prime Pillar – This option is typically only used on dual extruder, or dual material prints. I do not recommend its use, but see me if you have questions about its use.

  d. Ooze Shield – An option designed to prevent a dual extruder from oozing post extrude, onto the primary object. Typically unnecessary for single extrusion.

7. The "Infill" menu will allow you to adjust the unseen interior structure of your model

  a. General

      i.   Our Extruder is again the Primary (only) Extruder

      ii.   Internal Fill Pattern – Will offer you a number of options to choose the design of your models internal structure. The Honeycomb options provide a good blend of speed and structural integrity. The Afinia would typically produce a rectilinear infill. You're free to select whichever is best for your design.

      iii. External Fill Pattern – Will only affect your top and bottom layers. You can choose to have a rectilinear (think Raster) pattern, or a concentric (think cinnamon roll of death) pattern.

      iv. Infill Fill Percentage – How much your model will be filled in. Typically 10% - 20% works very well on this machine

      v.   Outline Overlap – This setting controls how much of the extruder nozzle will overlap the infill with the primary structure. 0% would mean they never touch, and your internal structure would be independent of the primary structure. 100% would mean it is being over printed. 20% is default.

      vi.   Infill Extrusion Width – If you would like to have a "finer" infill, you can set this to be less than 100%. I would advise leaving it at the default setting of 100%

      vii.   Minimum Infill Length – Anything below this value will not be printed. 5mm is default.

      viii.   Print Sparse Infill Every – Is asking you how often you would like your infill printed. 1 means it is printed every layer (strongly recommend this). 2 would mean it is printed every other layer. This would be useful if you were messing with primary layer heights.

      ix.   Include Solid Diaphragm Every – Will print a solid floor every X layers to help with structure assembly. Default is 100, but you're free to adjust this as you deem necessary.

b.   Infill Angle Offsets is a setting that will determine the angle your infill gets printed at. While there can be some structural benefits, it is largely used for aesthetics.

      i.   You could determine the angle for every single layer if you wanted, and build an extremely strong (but messy) structure.

8.   The "Support" menu lets you control the GENERAL support settings. We can fine tune this with the "Customize Support Structures" icon on the right hand tool bar.

a.   Support Material Generation

      i.   Generate Support – Default is active.

      ii.   Support Infill Percentage – Determines the internal fill structure of your support. Default is 40%

      iii.   Extra Inflation Distance – A setting that can extend the support structure beyond the perimeters of your part. .20mm is default.

      iv.   Dense Support Layers – Determines how many solid layers print where the support structure interfaces with your part. Default is 3, but this may be best implemented if you adjust it based on your external layer settings.

      v.   Dense Infill Percentage – Sets the infill for that "Dense Support". 75% is default

      vi.   Print Support Every – The setting to determine how often to print support. Typically every 1 layers.

b.   Separation From Part

      i.   Horizontal Offset – Defaulted to .50mm is the setting that determines how much your support is offset from the primary structure.

      ii.   Upper/Lower Vertical Separation Layers – Defaults to 1. Recommend leaving it at 1.

c.   Automatic Placement – Simplify3D is able to auto generate all of your support, as well as allowing you to manually place it.

      i.   Support Type – "Normal" is default. You can also set it to print from the build platform only if you would like a clean model.

      ii.   Support Pillar Resolution – Adjust this to any diameter you'd like. The lower the number, the finer the resolution of the support pillars, but the weaker they'll be. 3mm is a good general setting.

      iii.   Max Overhang Angle – Works the same way that it does on the Afinia. 0 will generate support for everything and 90 will skip all support.

d.   Support Infill Angles – Works the same as the other infill angle settings.
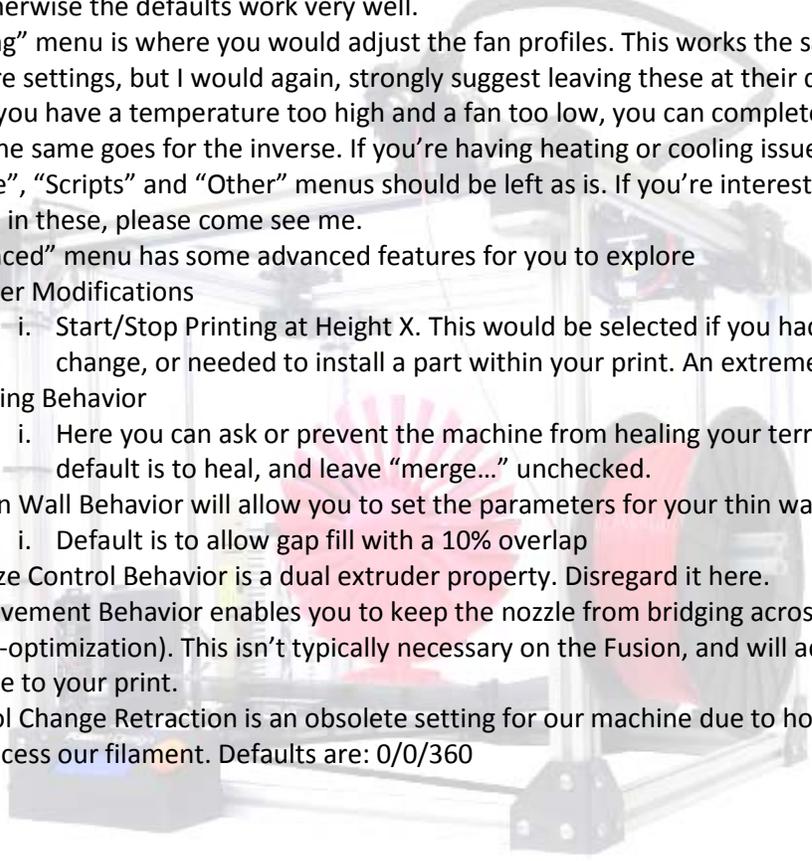
9. The "Temperature" menu avails you all of the temperature settings.
   a. First you will need to select (highlight) the proper controller from the "Temperature Controller List"
   b. The "Overview" menu will allow you to assign a temperature controller to an identifier. **You should never change these**
      i. Our Primary Extruder is T0
      ii. Out Heated Bed is T3
      iii. You can select to wait for the temperature to stabilize before beginning the build if you'd like. Default is on.
   c. Per-Layer Temperature Setpoints. These will default based on the material you select, and I would strongly suggest leaving them as is. A good use of this feature would be if you needed to print with multiple materials you could adjust the temperature per layer. Otherwise the defaults work very well.
10. The "Cooling" menu is where you would adjust the fan profiles. This works the same as the temperature settings, but I would again, strongly suggest leaving these at their defaulted settings. If you have a temperature too high and a fan too low, you can completely brick the machine. The same goes for the inverse. If you're having heating or cooling issues, come see me.
11. The "GCode", "Scripts" and "Other" menus should be left as is. If you're interested in exploring the options in these, please come see me.
12. The "Advanced" menu has some advanced features for you to explore
    a. Layer Modifications
       i. Start/Stop Printing at Height X. This would be selected if you had a material change, or needed to install a part within your print. An extremely useful tool.
    b. Slicing Behavior
       i. Here you can ask or prevent the machine from healing your terrible designs. The default is to heal, and leave "merge…" unchecked.
    c. Thin Wall Behavior will allow you to set the parameters for your thin walled settings
       i. Default is to allow gap fill with a 10% overlap
    d. Ooze Control Behavior is a dual extruder property. Disregard it here.
    e. Movement Behavior enables you to keep the nozzle from bridging across your model (de-optimization). This isn't typically necessary on the Fusion, and will add significant time to your print.
    f. Tool Change Retraction is an obsolete setting for our machine due to how we load and process our filament. Defaults are: 0/0/360