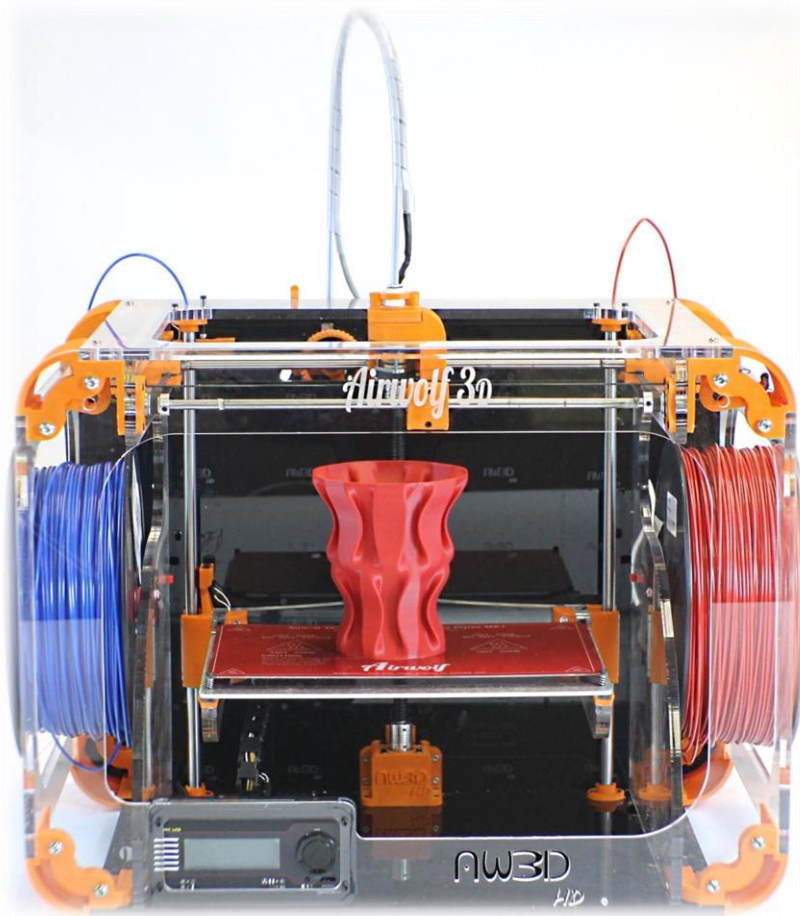


Airwolf3D™



AW3D *HD* and *HDx*

User Manual

www.Airwolf3D.com

Table of Contents

Specifications	4
Unboxing the Printer.....	5
Box Contents	5
Part Directory.....	6
How it Works.....	7
Extruder Assembly	7
Hot End and Heat Bed Motion	7
Plug In & Turn On.....	8
Pre-Printing Steps	9
Quick Start Guide	9
Quick Start Guide (contd.).....	10
1) Leveling the Heat Bed	11
First Print & Calibration.....	12
First Print & Calibration (contd.)	13
2) Loading Filament	14
3) Elmer's Glue OR GOO.....	15
4) The First Layer	15
5) LED Lights	15
VIKI LCD Screen	16
VIKI: Introduction	16
VIKI: Home Screen.....	16
VIKI: Layout.....	17
VIKI: How To	18
RAMBo Driver	19
Setup (Windows 8 Only).....	19
Installation (PC Only).....	20
5 Steps to Printing.....	23
1) Download or Design a 3D Model	24
2) Convert to STL File Type.....	25
3) Netfabb: Layout (STL Clean Up)	25
3) Netfabb: How To (STL Clean Up).....	26
4) Slicing	27

Slicing: Breakdown	28
4-5) MatterControl PRO: How To.....	29
4-5) MatterControl PRO: How To (contd.)	30
4-5) Repetier Host: Setup.....	31
4-5) Repetier Host: The Basics	32
4-5) Repetier Host: How To	33
4-5) Repetier Host: How To (contd.)	34
Material Selection	35
Materials: Comparison	35
Materials: Comparison (contd.)	36
Materials: Settings.....	37
Materials: Settings (contd.).....	38
Post Processes.....	39
Acetone Vapor.....	39
Two Part Epoxy.....	40
Firmware	41
Maintenance	42
Belt Tensioning.....	43
Rod Alignment (X-axis)	44
Rod Alignment (Y-axis)	45
Rod Lubrication	46
Inserting Filament	46
Wiring Diagram	47
Clogged Nozzle	48
Changing the Nozzle.....	49
Trouble Shooting.....	50
Glossary of Terms	51
Hardware.....	51
Software	51
Contact List	51
Notes.....	52
Notes.....	53

Specifications

General	Designed and Manufactured In:	USA	
	Build Volume (LxWxH)	12 x 8 x 12	inch
		300 x 200 x 300	mm
	Printer Size (LxWxH)	24 x 17.5 x 18	inch
		610 x 445 x 460	mm
	Weight	38	lbs
		17	kg
	Material that can be printed:	ABS, PLA, Soft PLA, PLA 4043D, BendLay, LayWOO D-3, Laybrick, HIPS, T-Glase, PVA, Nylon, TPE & more to come!	
	Filament Diameter	Nominal: 3.00mm Actual: 2.85 ± 0.1mm	mm
	Nozzle Diameter	0.5 and 0.35	mm
Specs.	Minimal Layer Thickness	0.06	mm
		0.0024	inch
	Max Speed-Perimeter	150	mm/s
		5.91	inch/s
	Max Speed-Travel	400	mm/s
		15.75	inch/s
Hard ware	Circuit Board	RAMBo	
	Bed	Temp Controlled Heated Bed	
	Frame	Partially Enclosed Acrylic	
	Display	VIKI LCD Screen	
	Power Supply	Internal auto-switching 13.5V DC, 300W	
Comp.	File Format	STL & GCODE	
	Software	Netfabb & MeshLab	
		Cura & Slic3r	
		Matter Control & Repetier-Host	
	System Compatibility	Windows, Mac, Linux	

Copyright 2013 ©

Unboxing the Printer

Box Contents

Airwolf 3D HD Printer



User Manual



Spool of ABS Filament



Glass with PET Film



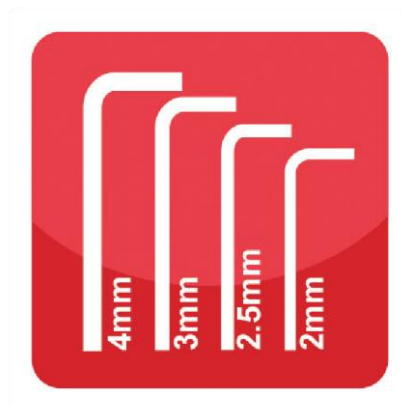
USB Drive



Micro SD Card



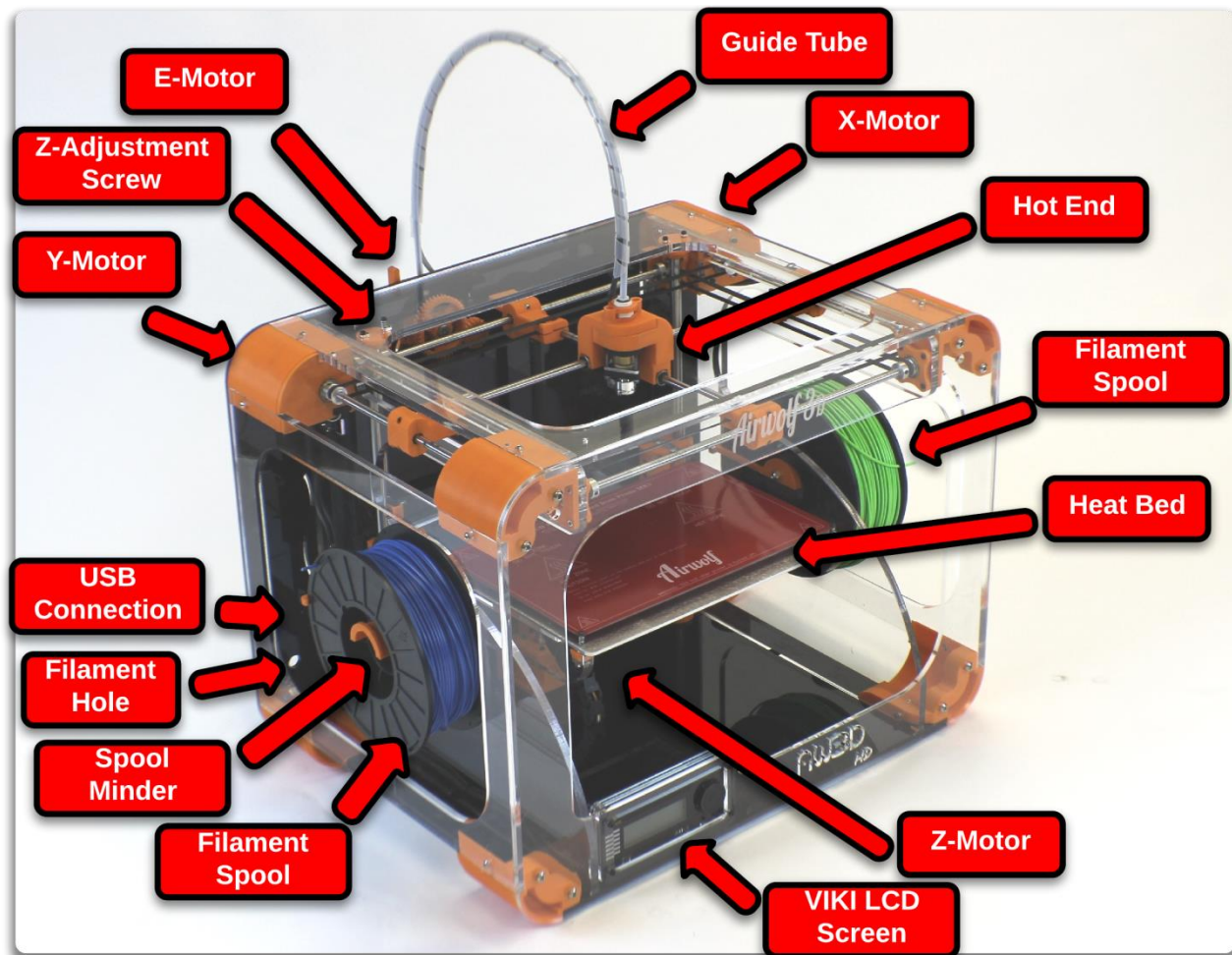
Allen Wrenches



ABS-Acetone Mixture



Part Directory



Warnings!!!



1) Don't touch the Hot End while it's hot

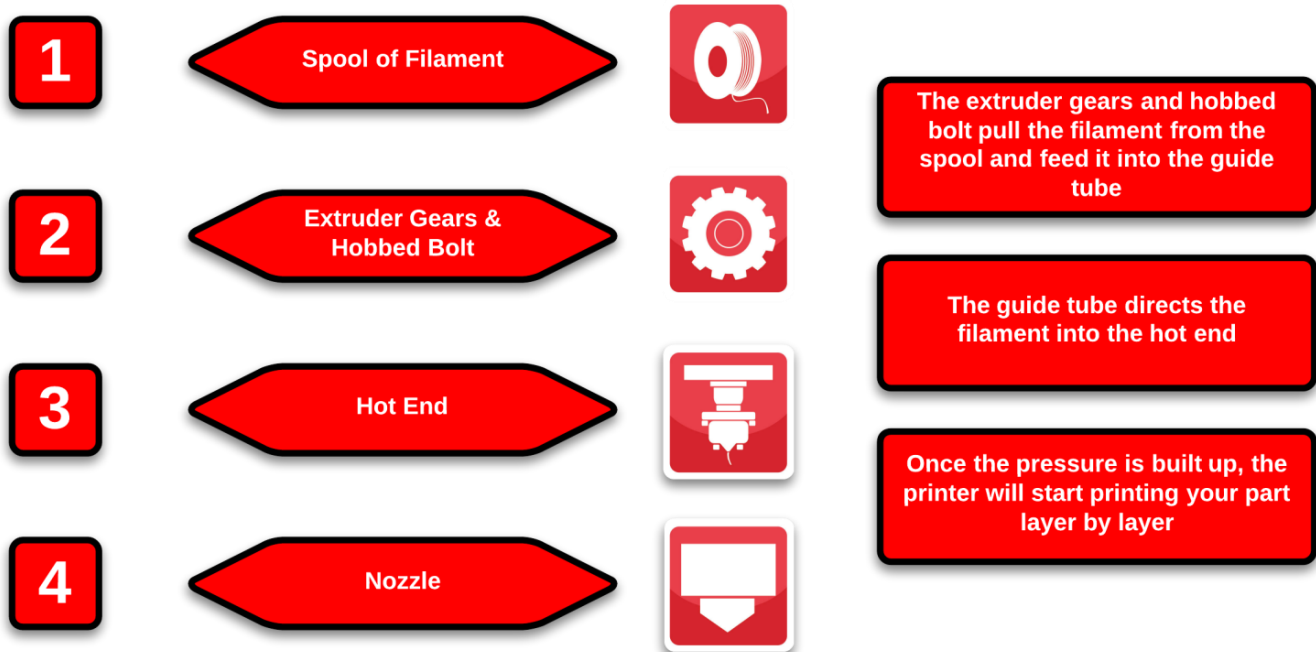
2) Don't touch Heat Bed while it's hot (glass included)



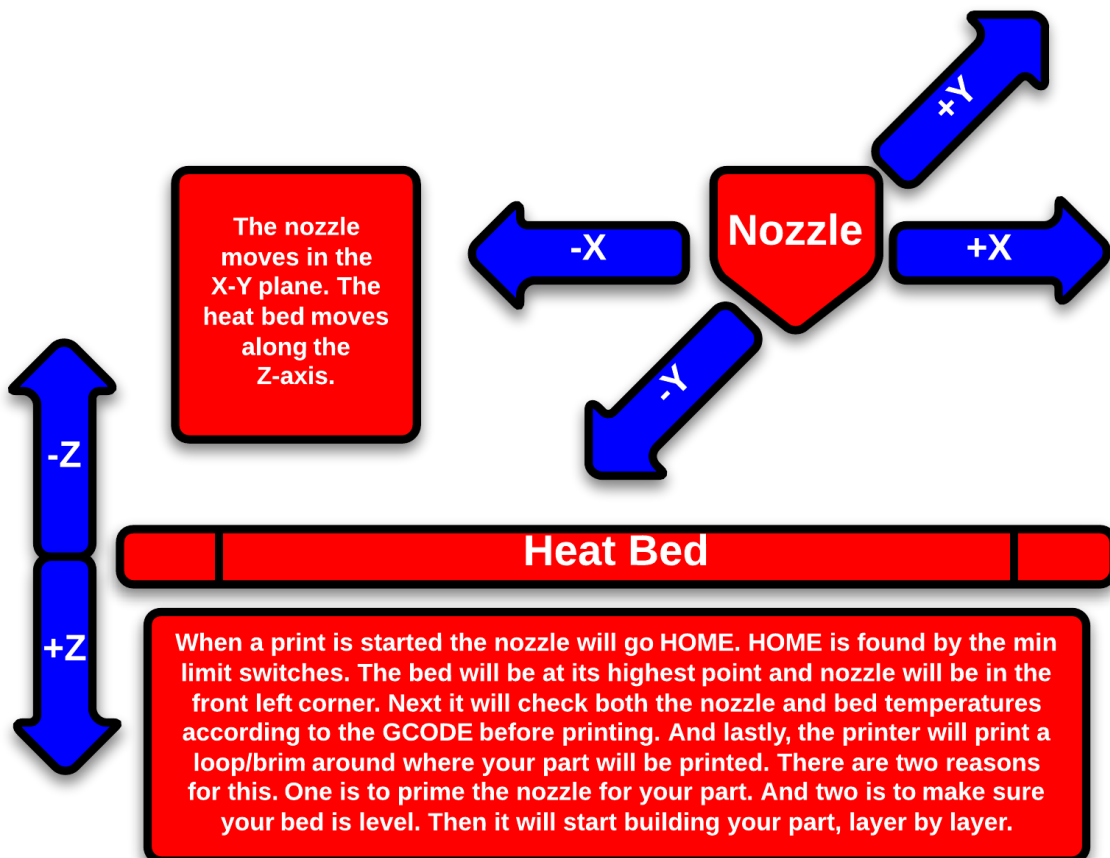
3) Pinching Hazard
(gears, pulleys, belts)

How it Works

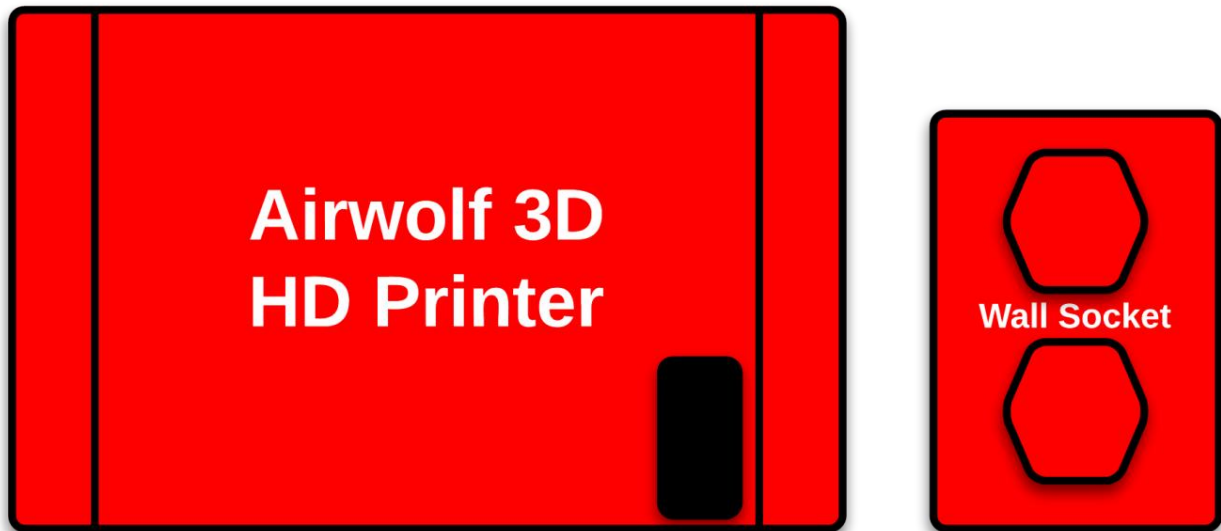
Extruder Assembly



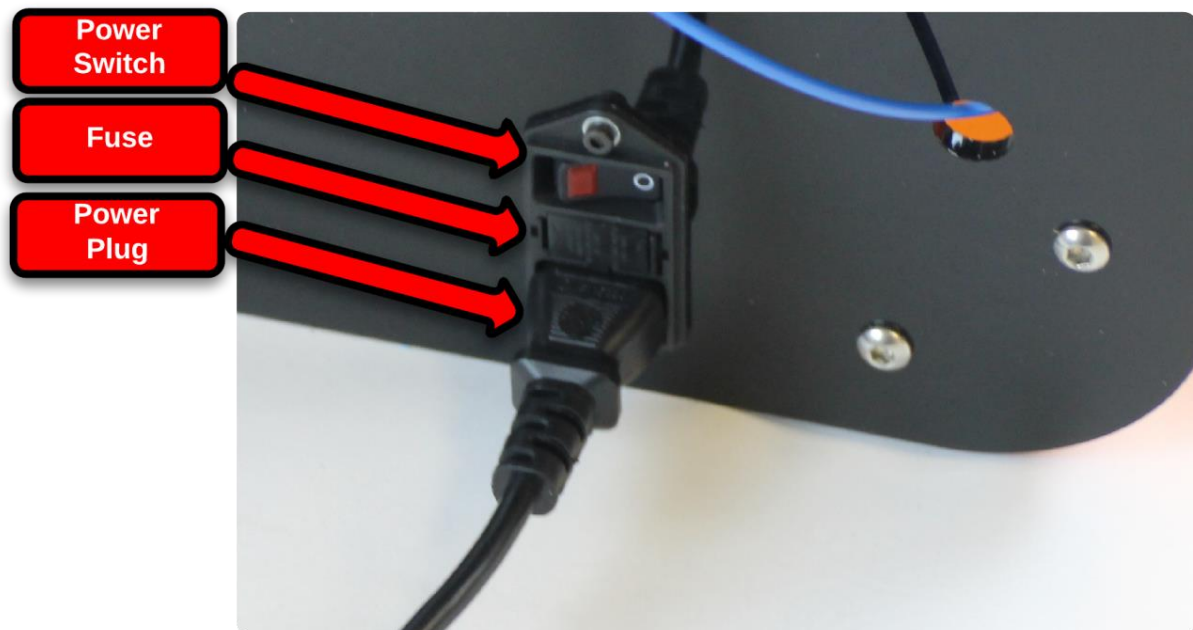
Hot End and Heat Bed Motion



Plug In & Turn On

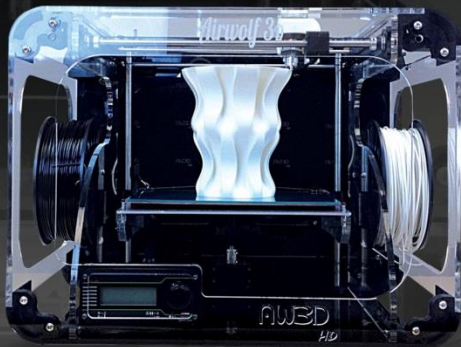


This is a view of the back of your printer. Plug your printer in with the power cable. Your printer's power supply has an internal auto switch, so regardless of which power source you supply it with, 115 or 230 volts, it will automatically switch for you. Then power on your printer. The power switch is on the back side of your printer, next to the power socket.



Pre-Printing Steps

Quick Start Guide



Airwolf 3D

HD

QUICKSTART GUIDE

1

REMOVE PRINTER FROM BOX

Face the printer away from you. Lift the printer with the side panels. **DO NOT LIFT THE TOP PANEL!!!**

2

REMOVE ALL ZIP TIES (4) AND TWIST TIES (2)

Remove 2 zip ties on the back corners of the heat bed, 1 on the bowden cable, and 1 on the filament. Also remove the 2 twist ties on the belts.

3

INSERT THE PTFE TUBE INTO THE HOT END

The PTFE tube is the semi-clear tubing that runs from the extruder motor on the back panel, to the hot end. Insert the PTFE tube into the plastic white adapter on the top of the hot end. Make sure to hold underneath the hot end and lift UP as you press the tube DOWN into the adapter. Then slip the C-clip around the adapter to secure the PTFE tube in the hot end. Refer to figure 3 for reference.

IMPORTANT!
Make sure to push the tube all the way down until it stops
Push Down Tube

Lift Up Hot End



Figure 1

4

CLAMP THE EXTRUDER ASSEMBLY AND REMOVE THE PIN

Use one hand to clamp the spring loaded extruder assembly. Use the other hand to remove the pin. Keep this pin nearby for use in a later step.



Figure 2

5

INSERT THE ADAPTER INTO THE EXTRUDER ASSEMBLY

Insert the PTFE adapter into the top of the extruder assembly. Pay close attention to the orientation of the adapter.



Figure 3

Quick Start Guide (contd.)

6 PLUG IN AND TURN ON THE PRINTER

Find the power cable and plug it into the back of the printer. Flip the switch next to the power cord input on the back of the printer. The LED lights will turn on.

7 VIKI LCD SCREEN

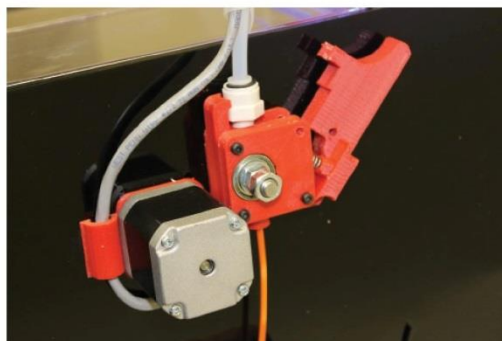
The VIKI LCD screen is located on the front panel of the printer. While looking at the Home Screen, press the center button. Then select "Prepare", "Move Axis", "1mm" and then "Z-axis". Then scroll counter-clockwise with to raise the bed (the number should be negative).

8 PREHEAT NOZZLE AND BED

Remove the foam from underneath the bed. Then select "Prepare" and then "Preheat ABS". This will heat up the nozzle and bed to general ABS printing temperatures.

9 INSERT FILAMENT

Feed the filament from the spool through the filament hole in the back panel and up into the bottom of the extruder assembly. With the nozzle temperature at 240C, you will be able to CAREFULLY push the filament through the nozzle. This way you know that the filament is completely through the hot end.



10 CLOSE THE EXTRUDER ASSEMBLY

Clamp the extruder assembly on the back panel and re-insert the pin.

11 ATTACH THE GLASS TO THE HEAT BED

Un-pack the glass with the PET film (green tint). Set the glass on the red heat bed, with the tape on the top side. Attach the 4 binder clips in the corners of the glass. Keep the front left clip about 2" away from the corner.

12 APPLY ABS-ACETONE "GOO"

Refer to the HD User Manual (in accessory box) to apply the "Goo". It's located under "Pre-Printing Steps". This will help the ABS filament to properly adhere to the heat bed.

13 START A SAMPLE PRINT

On the VIKI LCD Screen, select "Print from SD" and select a sample print GCODE file. This will help you learn the 3D printing process from start to finish.

If you have any questions about starting your first print, or have any questions about **this quickstart guide**, please call **TECHNICAL SUPPORT: 949.478.2933**

Airwolf 3D™

1) Leveling the Heat Bed

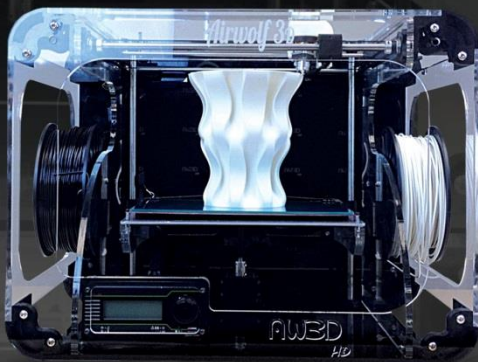
Why Level the Heat Bed- One of the most important steps in 3D printing is how the first layer prints. Like building a house, the first layer is the foundation for the remainder of your print. Without a level bed, your print will be marginal at best. Because of the configuration of the printer, the heat bed only moves slowly in the Z direction. This means that the heat bed tends to stay reasonably level over time. The better the calibrated printer, the more efficient the printing process is. Calibrating the printer should be done once a month or so and when traveling with the printer in a car. This will insure that the nozzle doesn't run into the bed and that the accuracy of the printer stays intact.

Basic Concept to Level the Heat Bed- There are two parts to leveling the heat bed. The first part is to make sure that the XY plane (the plane that the nozzle moves in) is parallel to the heat bed (glass). This is achieved by the four small M3 screws in each corner of the heat bed. The second part is the first layer height. This is achieved by the Z-adjustment screw and limit switch subassembly for the heat bed. The Z-adjustment screw is labeled on the Part Directory page. This spring loaded screw initiates the limit switch (Z-min) and sets the first layer height.

Steps to Leveling the Heat Bed- On the micro SD card, there is a Calibration Print. Run this calibration print to level the heat bed. There is also a video on our website that is a step by step guide along for this calibration. We highly recommend that you run this print before trying to print out parts. Your success rate is highly dependent on how level your heat bed is. The quick start guide will also have the steps written out as well. The next two pages have a step-by-step guide for calibrating the printer.

Once you have calibrated the printer, we highly recommend that you run a few sample prints before moving on to custom prints. This will ensure that the printer is working correctly. Also you will gain valuable knowledge from this. For one, you will understand how the printer works before having to dive into the settings. And secondly, you will have the opportunity to see some massive prints that very few printers can print. Plus you get to show off to your family, friends, coworkers and boss the incredible sample parts.

First Print & Calibration



Airwolf 3D™

HD

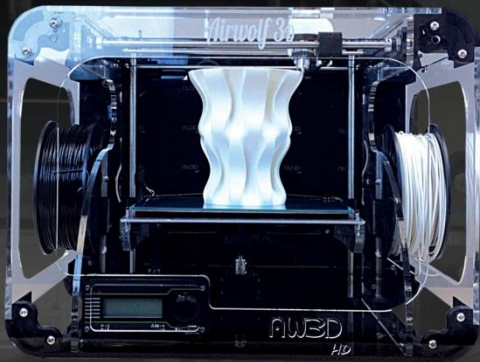
FIRST PRINT/CALIBRATION

- 1** Place glass on the heat bed with the PET film (Green) on the top side.
- 2** Place the four binder clips in each corner of the glass.
 - a. Make sure to keep the front left binder clip at least 3 inches away from the corner.
- 3** Pre-heat the printer with the VIKI LCD Screen.
 - a. Press the center button to move from the home screen to the menu.
 - b. Scroll down to "Prepare."
 - c. Press the center button to move to the next menu.
 - d. Scroll down to "Preheat ABS"
 - e. Press the center button to set the temperatures for the nozzle at 240C and the bed at 120C.
- 4** Place the spool of filament on the spool minder with the filament unspooling from the bottom towards the back.
- 5** Feed the filament through the filament hole in the back panel.
- 6** Squeeze the extruder assembly on the back panel and release the pin.
 - a. Set the pin on the top panel to keep track of it.
- 7** Feed the filament into the bottom of the extruder assembly.
 - a. Push the filament up into the extruder until the filament comes out of the nozzle.

NOTE: This will only happen if the nozzle temperature is at least 240C.

- 8** Close the filament latch and re-insert the pin
- 9** Turn the Z-adjustment screw clock-wise (on the top panel) 4 complete turns
- 10** Insert the micro SD card into the VIKI LCD Screen
 - a. Make sure it says, "Card inserted" on the VIKI LCD Screen
- 11** Start the Calibration print with the VIKI LCD Screen.
 - a. Press the center button to enter the menu.
 - b. Scroll down to "Print from SD"
 - c. Press the center button.
 - d. Scroll down to "FIRST PRINT"
 - e. Press the center button
- 12** Check to make sure the fan on the extruder is running and that the LED lights are on.
- 13** Take a small piece of printer paper and fold it in half.
- 14** The heat bed will move all the way down on the Z axis and then all the way back up until it initiates the z-limit switch.
- 15** The printer will now test the speed of the x and y axis.

First Print & Calibration (contd.)



Airwolf 3D™

HD

FIRST PRINT/CALIBRATION

- 16** The nozzle will stop at the center of the heat bed until it initiates the Z micro switch.
- 17** Place the piece of paper between the nozzle and the heat bed. You want to make sure the paper fits and is not too loose. If it is too high, or if the paper doesn't fit, adjust your Z adjustment screw. Counter-clockwise will lower your nozzle, clockwise will raise it. Adjust at about a quarter turn at a time. Adjust until the paper fits snug between the nozzle and heat bed. You have a few tries to get it right.
- 18** Place the piece of paper between the nozzle and the heat bed. You want to make sure the paper fits and is not too loose. If it is too high, or if the paper doesn't fit, adjust your Z adjustment screw. Counter-clockwise will lower your nozzle, clockwise will raise it. Adjust at about a quarter turn at a time. Adjust until the paper fits snug between the nozzle and heat bed. You have a few tries to get it right.
- 19** The nozzle will not re-locate to the front left corner. Again, take the piece of paper and slide it between the nozzle and heat-bed. If the nozzle is too low (paper won't fit), turn the heatbed screw on that corner clockwise. If it is too high, turn the screw counter-clockwise. We want to get this height the same as the middle in the previous step. You will have a few tries to get it right. Repeat this step for the next 3 corners.

- 20** The entire process will now repeat itself, just to make sure everything was done properly. It is a good idea to double-check the settings.
- 21** The printer will now pause, and at this time you will want to apply the "Goo" solution on to the print bed. The extruder will now heat to full temp (240C)
- 22** The bed will lower, and then raise back up. The calibration print will now begin. It will print a large rectangle around the edges of the bed. Pay close attention to the bead of plastic. If it is rounded and not sticking well, this means your nozzle is still too high. Make minor Z screw adjustments. If it is flat and even, then your first layer has been set correct. Please review the images below. If your first layer is sticking and is uniform, then your printer is calibrated.

This diagram show a cross-section of filament as it lays down onto the bed.



Please scan the QR code to view the Youtube video explaining this procedure in full detail.



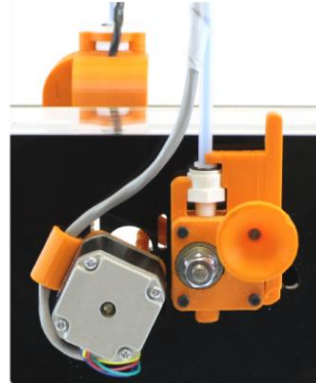
**Visit
www.airwolf3d.com
for more information.**

2) Loading Filament

Good quality filament will be $2.85 \pm 0.10\text{mm}$ in diameter. If the filament is consistently over 3.00mm, you may want to purchase the material from somewhere else.

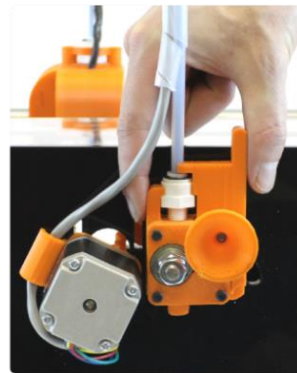
1

Closed Extruder



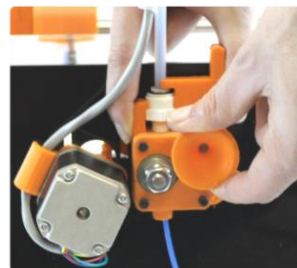
2

Press Lever against Bowden Cable



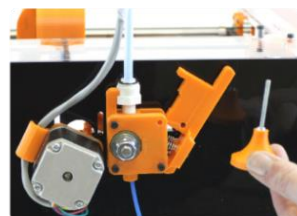
3

Pull Extruder Pin



4

Release Latch and Insert Filament into the Guide Tube



3) Elmer's Glue OR GOO

For ABS prints, a PURPLE Elmer's Glue stick can be extremely helpful in keeping your parts stuck to the bed. This is very important in keeping the part from warping or curling during the print. The PURPLE Elmer's Glue stick can be applied on top of the PET film as the bed is heating up. It only needs to be applied where you are printing your part.

GOO is a different way of helping your parts to stick. GOO is a solution of natural ABS sticks and Acetone in a glass jar. The mixture in the jar should look murky. When your bed is heating up, saturate a piece of paper towel and wipe the bed. This will be applied on top of the PET film (green) that is already on your glass. The area of the glass where the GOO is applied should have a white frost look to it.

EITHER USE THE PURPLE ELMERS GLUE STICK OR THE GOO, BUT NOT BOTH.

4) The First Layer

Why- The first layer is by far the most important layer of your entire print. If your first layer is successful, you have a very good chance of having a successful print. The first layer height is controlled by the Z-adjustment screw which is spring loaded and in the back left corner of the top panel. The bed raises and initiates the limit switch to set the first layer height. Once your bed is level, there are a couple steps to getting a good first layer.

How To- First turn the Z-height-adjustment screw clockwise a half turn with your 2.5mm allen wrench. Next, move your nozzle to the center of your bed. Then home the Z-axis. The bed will raise and initiate the limit switch. At this point, the nozzle should be off the bed. Turn the Z-height-adjustment screw counter clockwise a quarter turn. And once again home the Z-axis. Repeat this process until your nozzle is just off the bed.

Nozzle Too High- If you run a print with the nozzle too high, the filament will just lie on the glass and not stick. Turn the Z-adjustment screw counter clockwise. This means the Z limit switch to be activated later, and therefore the nozzle is closer to the glass. Home the Z again and repeat until the nozzle is just off the bed.

Nozzle Too Low- If you run a print with the nozzle too low, the filament will have no room to come out of the nozzle and seem to clog. If this does happen, raise the nozzle off the bed at least 3 inches, make sure your nozzle is heated to at least 245C, and extrude material. Then turn the screw clockwise. Home the Z again and repeat until the nozzle is just off the bed.

Nozzle Height is Just Right- The first layer should look like a pancake. The nozzle should be extruding the material and smooshing it into the glass.

***This can be done during the first layer of the print with the Z threaded rod (the threaded rod attached to the Z-motor). This assumes that the bed has already been leveled. If the nozzle is too close to the glass, turn the Z threaded rod counter clockwise to lower the bed. If the nozzle is too far from the glass, turn the Z threaded rod clockwise to raise the bed. This should be no more than 1/8 of a turn or so. NOTE that this action will only offset the first layer for the current print. If run again, the adjustment will not have effect.

5) LED Lights

The LED lights will turn on automatically when your print starts and turn off when your print ends. But you can manually turn on the lights. "M42 P6 S255" will turn them on, "M42 P6 S0" will turn them off.

VIKI LCD Screen

VIKI: Introduction

Out of the many features the VIKI LCD Screen has, there are four main ones that you will use regularly. They include **Auto Home**, **Preheat ABS**, **Move Axis** and **Print from SD**. The following page has a layout of all the features the VIKI has to offer and these four functions are underlined.

Auto Home- Auto Home (AKA the origin) will use the min limit switches to move the nozzle to the front left corner, and the bed to the top. Home is very important because that is where every print will start from.

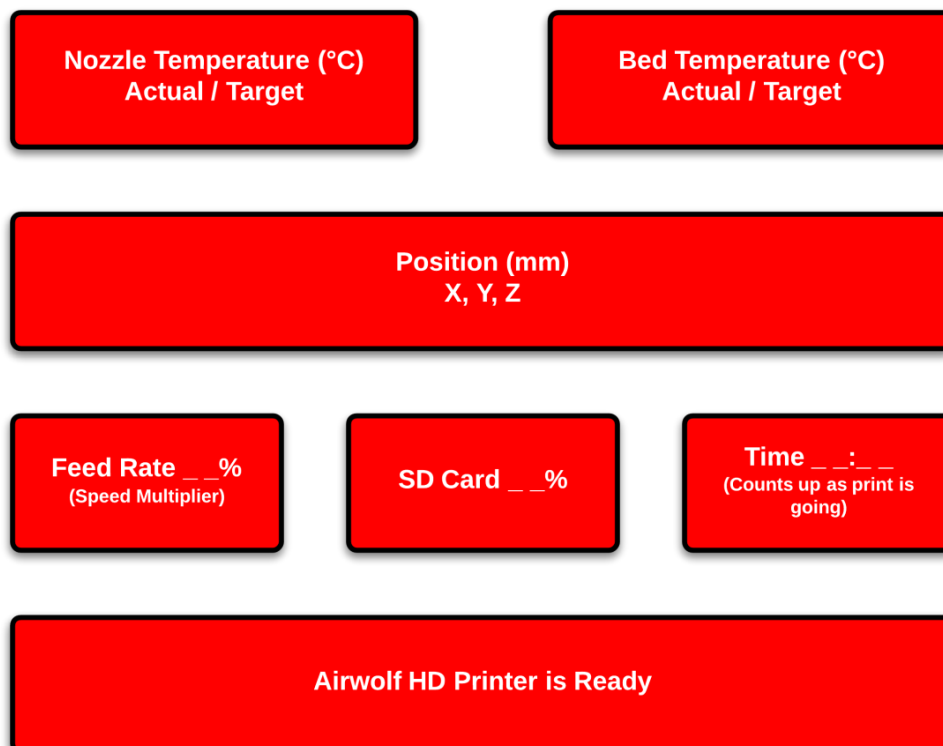
Preheat ABS- Preheat ABS is a great feature for preheating your printer before printing. When you are done designing, slicing, and loading your SD card, your printer is ready to go.

Move Axis- Move Axis allows you to manually move the nozzle and bed in each direction.

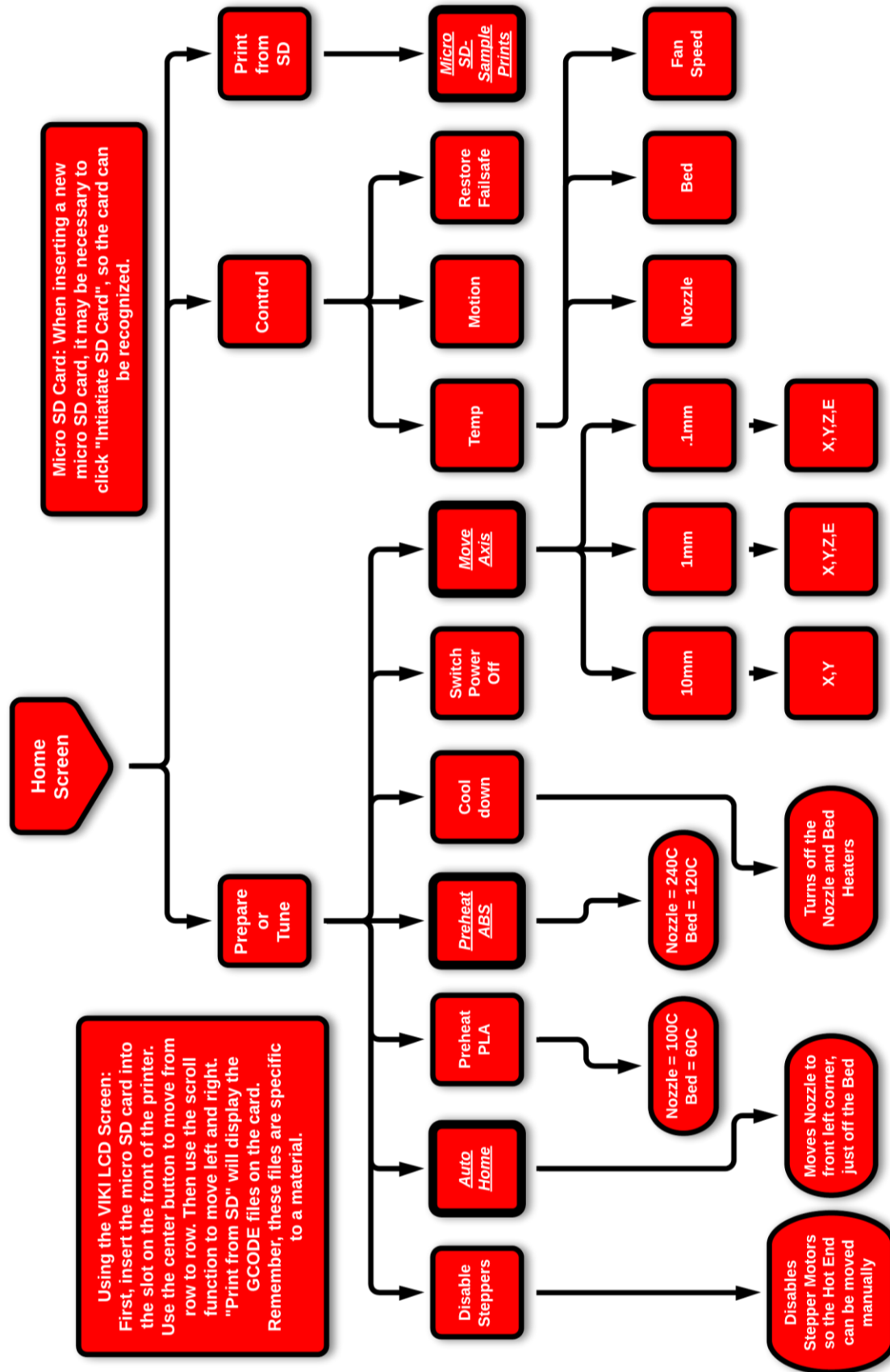
Print from SD- Print from SD is where you can access the GCODE files on your micro SD card. We have already loaded a library of sample prints for you to run right away.

VIKI: Home Screen

This is the home screen for the VIKI LCD Screen. There are two heated components on the printer, the nozzle and the bed. Both actual and target temperatures are displayed. The "POSITION" is the current position of the nozzle relative to its home position. The "FEED RATE" is a multiplier. At 100%, the printer is running exactly what the GCODE is instructing it to do. If the feed rate is increased or decreased, the printer will speed up or slow down in every direction. To change the federate, turn the dial on the VIKI home screen and watch the percent change. It is not recommended to go any faster when using the fast settings. The SD Card __% is how full your micro SD Card is. The "Time" is the amount of time that the printer has been printing its current job.



VIKI: Layout



VIKI: How To

The four main functions of the VIKI LCD screen are to prepare for a print, manual control, run a print and control the printer during a print. Use the layout on the previous page as a road map to find each function. To go from row to the row, simply click the center button. Use the scroll wheel to move left and right in the road map. If left idle for 15 seconds, the VIKI will return to the home screen.

Prepare- The main functions used under “Prepare” are “Auto Home”, “Preheat ABS”, and “Move Axis”. “Auto Home” will instruct the printer to move the extruder to home in each direction (X, Y, Z). The home for the printer is essentially the origin in the XYZ coordinate system. “Preheat ABS” will turn on both heaters for the extruder and bed to near the printing temperatures of ABS. This way, when you start your print, the temperatures may just have some minor changes if needed.

Manual Control- Under “Prepare”, there is “Move Axis”. Once selecting an increment and axis, the extruder and bed can be manually controlled by the user. This also includes the extruder. This function can be used to purge the machine of a previous material or color. When changing materials, purge between 500 and 1000mm to completely get rid of the old material. (Also see the material section for further instructions about different materials)

Print from SD- “Print from SD” utilizes GCODE files to run the printer. Once an STL file is sliced and GCODE is created, it can be saved onto the micro-SD card. Keep in mind that once a part is sliced, that GCODE file is for a specific material (temperatures, speeds, etc.)

Tune- There are a few things that can be altered during the print by using the VIKI LCD screen. One of them is speed or feed rate. This will speed up or slow down a print. The rule of thumb is not to go any faster than fast settings and not any slower than slow settings. At 100%, the printer is running the GCODE exactly, utilizing the settings. Above 100%, the printer will increase its speed; below 100%, the printer will decrease its speed. It is not recommended to increase the feed rate higher than 125% or lower than 75%. This is mainly because of the retraction in the bowden system. The other way to increase or decrease the speed during a print is to spin the wheel while looking at the home screen; this will cause the feed rate value to change accordingly. The nozzle and bed temperatures can also be altered during the print. Some times for big or tall parts, it may be useful to increase the bed temperature a little. The fan speed is only used for materials that would require fans (refer to “Materials Selection”). ABS does NOT need the fans.

RAMBo Driver

Setup (Windows 8 Only)

Windows 8 by default will not let you install unsigned drivers. This means that a Windows 8 computer will not recognize the RAMBo board unless you follow the steps shown below to start Windows 8 in "Disable Driver Signature Enforcement" mode.

1

Settings Button

Move the cursor to the bottom right corner of the screen and select the gear "Settings"

2

Change PC Settings

Select "Change PC Settings"

3a

General

If you have "General", select it and proceed to step 4

3b

Update & Recovery

If you don't have "General" select "Update & Recovery"

3b

Recovery

Select "Recovery"

4

Advanced Startup > Restart Now

Scroll down and select "Restart Now" under "Advanced Startup"

5

Troubleshoot > Advanced Options > Startup Settings

After selecting "Startup Settings", select "Restart"

6

Restart and 7

The PC will restart. Upon startup, click the "7" key on your keyboard.

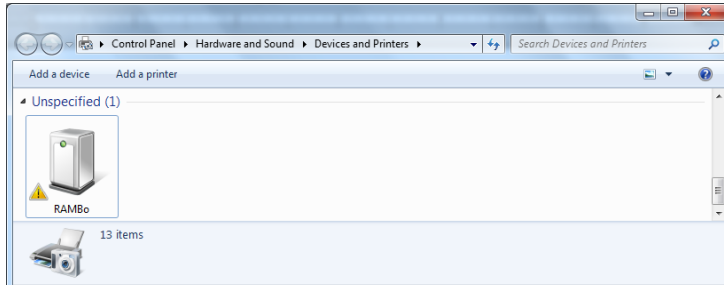
7

Install RAMBo Driver

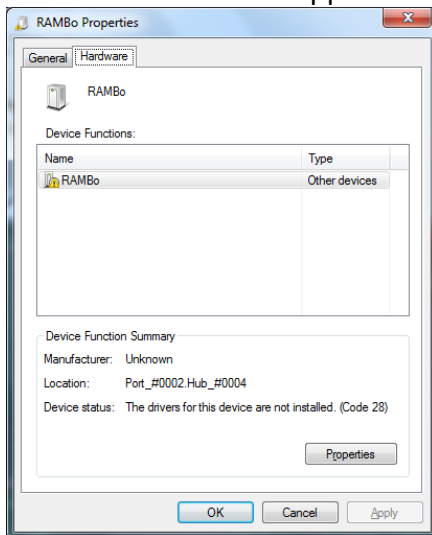
Then proceed to install the RAMBo driver

Installation (PC Only)

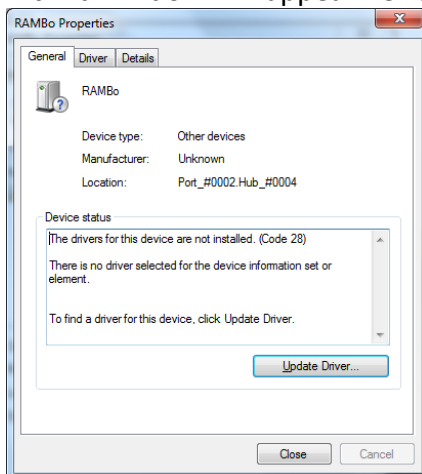
- 1) Only PC computers need the RAMBo Driver.
- 2) Plug your printer into your computer with the USB cable. Also insert the Airwolf USB stick provided with the printer.
- 3) In the “Control Panel”, open up “Device and Printers”. Under “Unspecified”, you should see a device labeled “RAMBo”. Right click the device and select “Properties”.



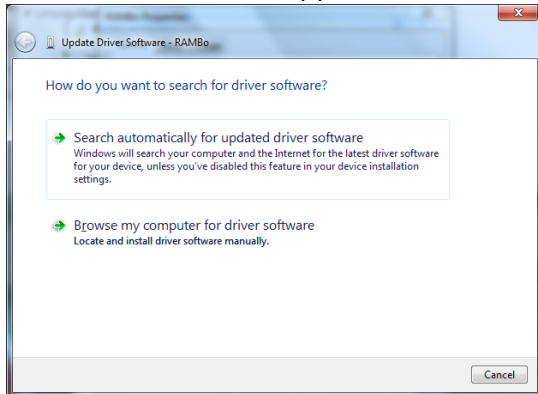
- 4) A second window will appear. Under the second tab labeled “Hardware”, select “Properties”.



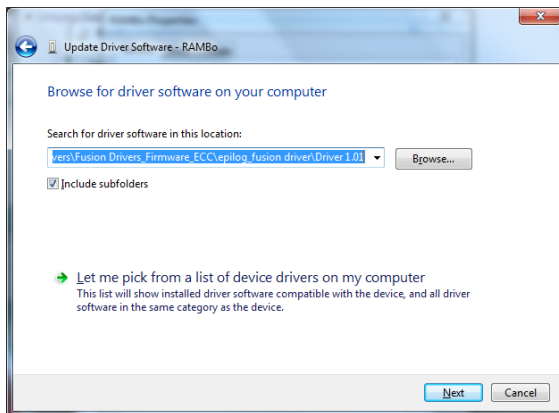
- 5) A third window will appear. Under the first tab labeled “General”, select “Update Driver”.



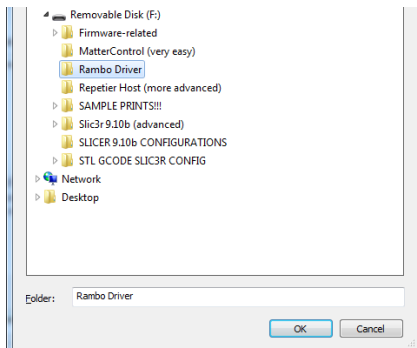
6) A fourth window will appear. Select “Browse my computer for driver software”.



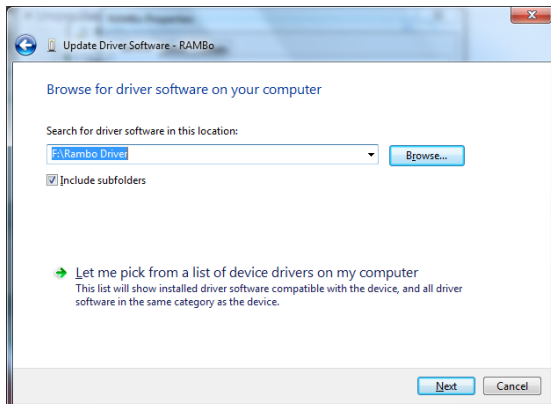
7) Select the “Browse” button.



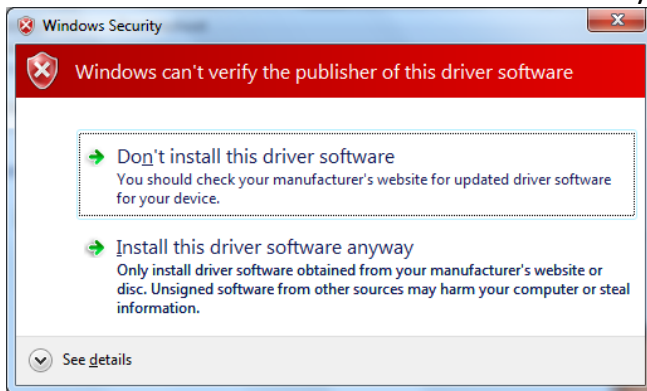
8) Browse for the Airwolf USB stick. Select the folder “Rambo Driver” and select “OK”.



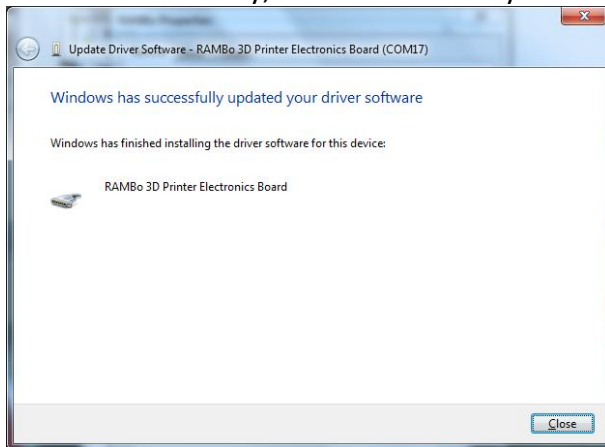
9) That window will close and return to this screen. Select “Next”.



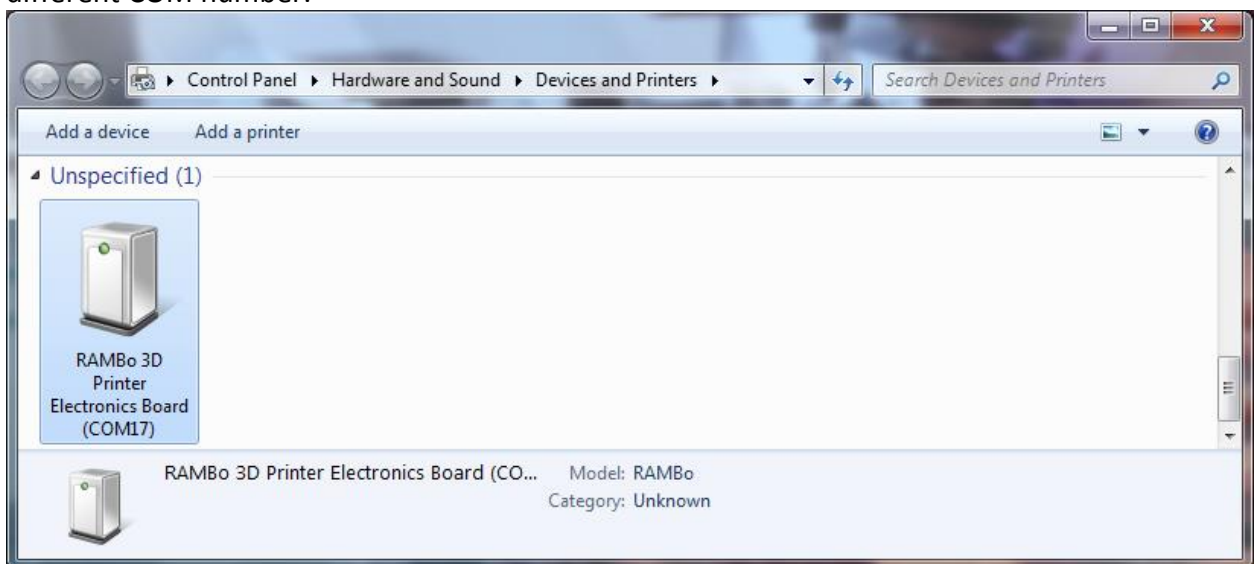
- 10) Another screen may appear and say “Windows can’t verify the publisher of this driver software”. Select “Install this driver software anyway”



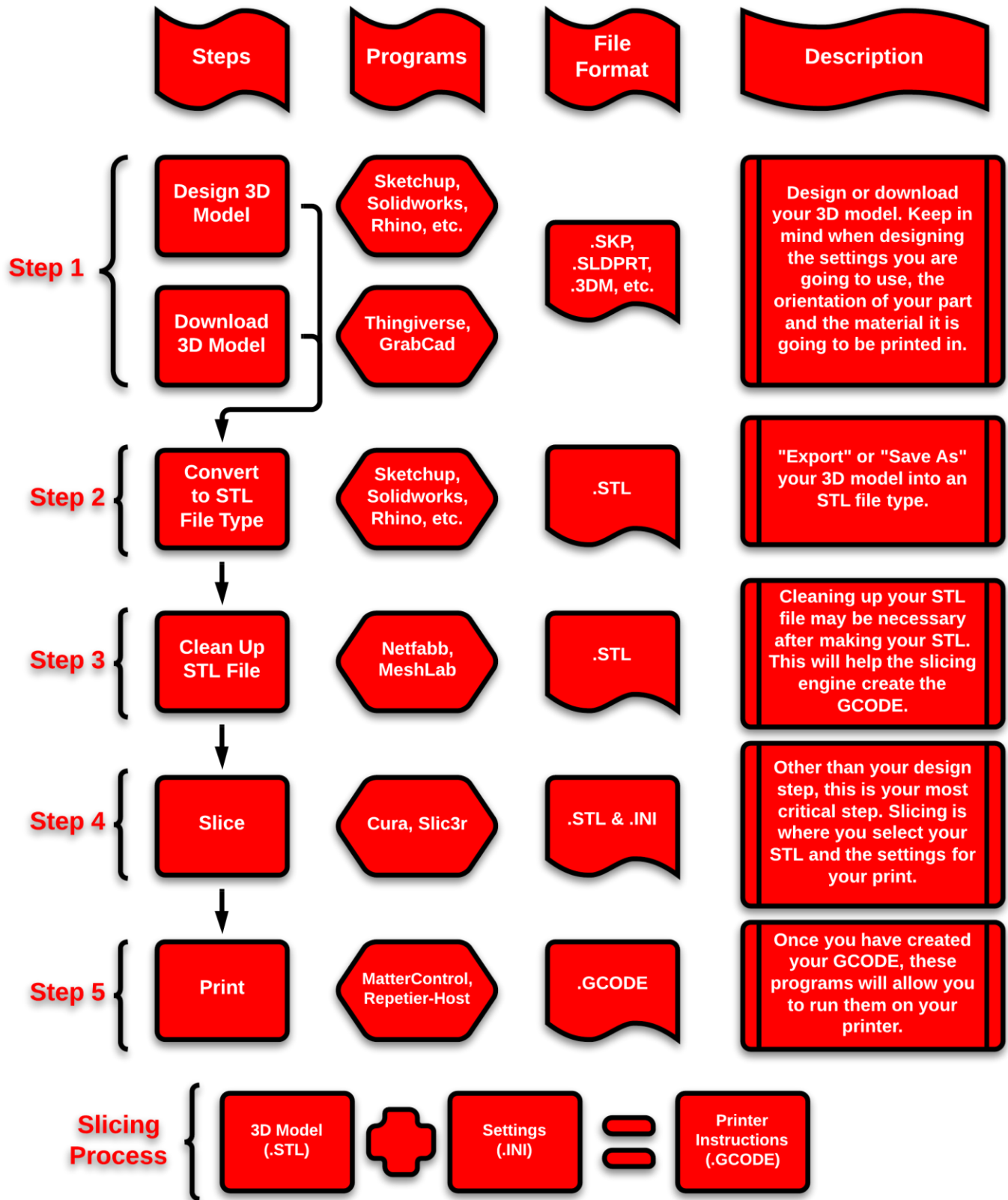
- 11) If installed correctly, this is the screen you should see. Select “Close”, “Close”, and “OK”.



- 12) Now in your “Devices and Printers”, under “Unspecified”, this is the screen you should see. The “(COM17)” is dependent on what USB port you plugged the printer into. So yours may display a different COM number.



5 Steps to Printing



1) Download or Design a 3D Model

There are a couple of websites where you can download 3D models. Thingiverse is a great place to find STL files to print. Another place to find 3D models is GrabCad. GrabCad has many models for many different CAD programs. There are a couple things to consider when selecting or designing a 3D model.

Orientation- The orientation of your part is critical to a successful print. Different orientations will produce a difference in strength, print time, appearance and success rate. Definitely envision the orientation of your part as you are designing the part, feature by feature. This will greatly enhance your printing experience.

Support Material- There are three ways to approach support material. The best way is to design and orient your part so that it does not need any support. The next best option is to design your own supports into the part. This way there is only support where you want it and it is easy to remove. Also you can control how thick the supports are as well. The last way is to have the slicing program “generate support material” where needed. This is helpful when options one and two do not work sufficiently. This is the most extensive method of printing. It takes time and patience.

Settings- As far as settings go, we recommend that you import the settings from the USB stick that was provided with the printer. These are already preconfigured settings for each of the materials. Quite regularly we update the material settings, so send us an email for the latest version of settings. Another thing to consider while designing is what settings you will ultimately use to slice the part (fast/slow, solid/hollow, etc). One way that you will select settings is by the application of your part. For parts under stress or heat, you may want to select slow print settings and make it almost completely solid. For parts like sculptures or statues where there is almost no physical function of the part, you may select fast and hollow settings.

Material Selection- Material plays a big part in the application of your part. ABS is by far the easiest and cheapest printing material. It can take physical and thermal stresses quite well and can be post processed easily. Turn to the “Materials Selection” section for a more in depth introduction to the different materials.

2) Convert to STL File Type

STL is a file type created specifically for 3D printing. All modern 3D Printers, desktop and industrial, use the STL file format and it is therefore transferable between various 3D Printing operating programs. It stands for Standard Tessellation Language but is commonly mislabeled as Stereolithography. STL files are composed of data points in three dimensional space and triangles linking those data points, therefore creating a mesh. It is important that this mesh is clean and watertight so the slicing software can properly interpret the data, the next section covers how to ensure a clean watertight mesh.

When exporting an STL from your CAD program it is important to make sure your settings are not unnecessarily high resolution. Creating very high resolution STLs can often times cause processing delays and issues when slicing or printing. It is good practice to generate STL files with parameters tailored to the capabilities of your machine.

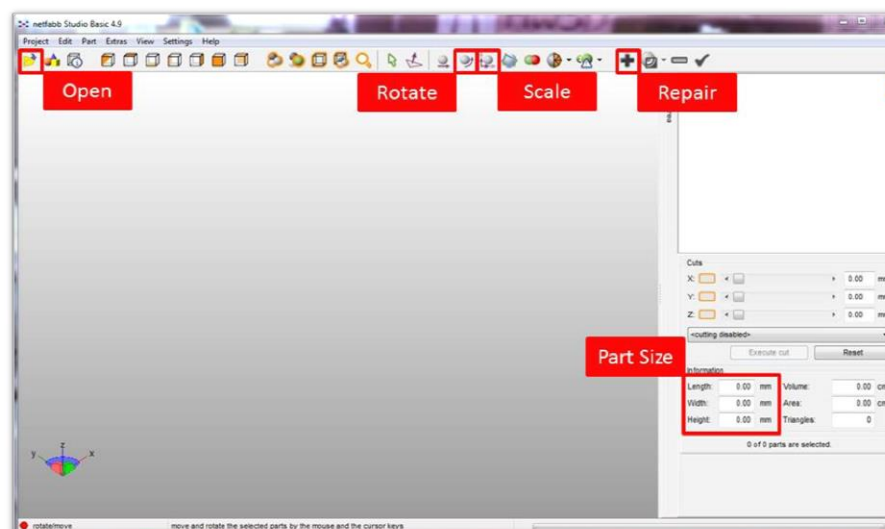
Recommended settings for the HD are as follow:

Deviation Tolerance: 0.05mm or greater

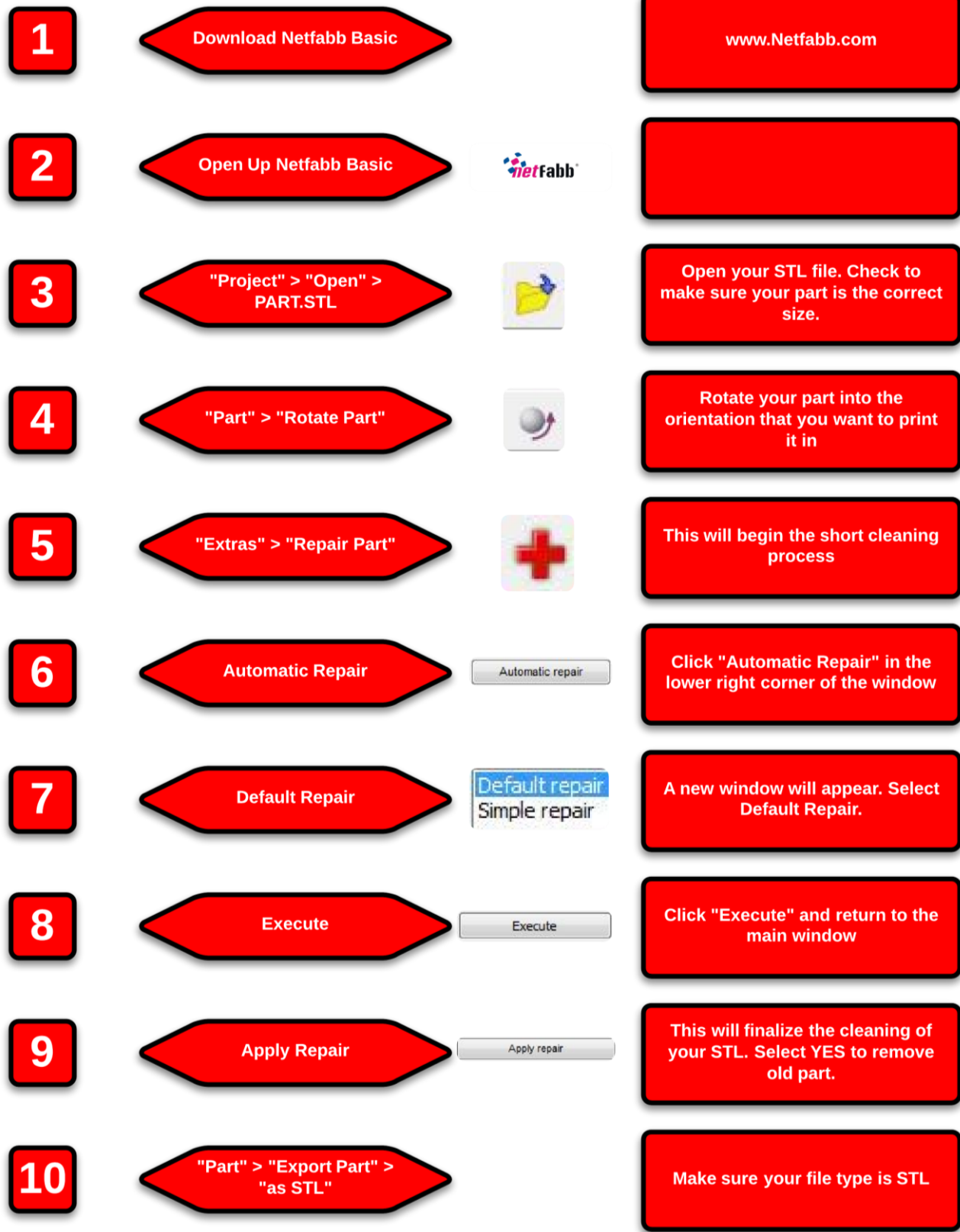
Angle Tolerance: 5 degrees or greater

3) Netfabb: Layout (STL Clean Up)

Having a clean STL file is crucial to a successful slice and eventually a successful print. Depending on the CAD program used to design the 3D model, and also the complexity of the 3D model, some create the STL file cleaner than others. But just to be safe, it is a good rule of thumb to clean up your STL file. Also, this is a great place to check to make sure the part is in the printing orientation. You can also check to make sure your CAD program exported it correctly size wise. Also this is a program to make sure your STL file is “watertight” for printing correctly. This is especially important when designing in CAD programs like SketchUp. If you do change the units in Netfabb from millimeters to inches, make sure to switch it back to millimeters before exporting the STL file.



3) Netfabb: How To (STL Clean Up)



4) Slicing

Slicing is what 3D printing is all about. Slicing is a process in where the STL file is “sliced” into thin layers for the printer. There are three parts to this process; the STL file, the settings, and the GCODE. The slicing engines that are utilized are Slic3r or Cura. They create GCODE and are incorporated in MatterControl and Repetier-Host. These two programs take the GCODE and run the printer with it. Only one program at a time is needed to run the printer.

The time taken to slice an STL file is dependent on the complexity of the geometry, the settings selected and the computing power of your computer. If Slic3r is having trouble slicing your part, try using Cura. Cura can usually slice just about anything.

As you begin to learn about the slicing engines and how they work, the best way is to load preset settings from the USB stick. As you start getting more experienced, you can start adjusting certain settings to increase the quality of your specific parts.

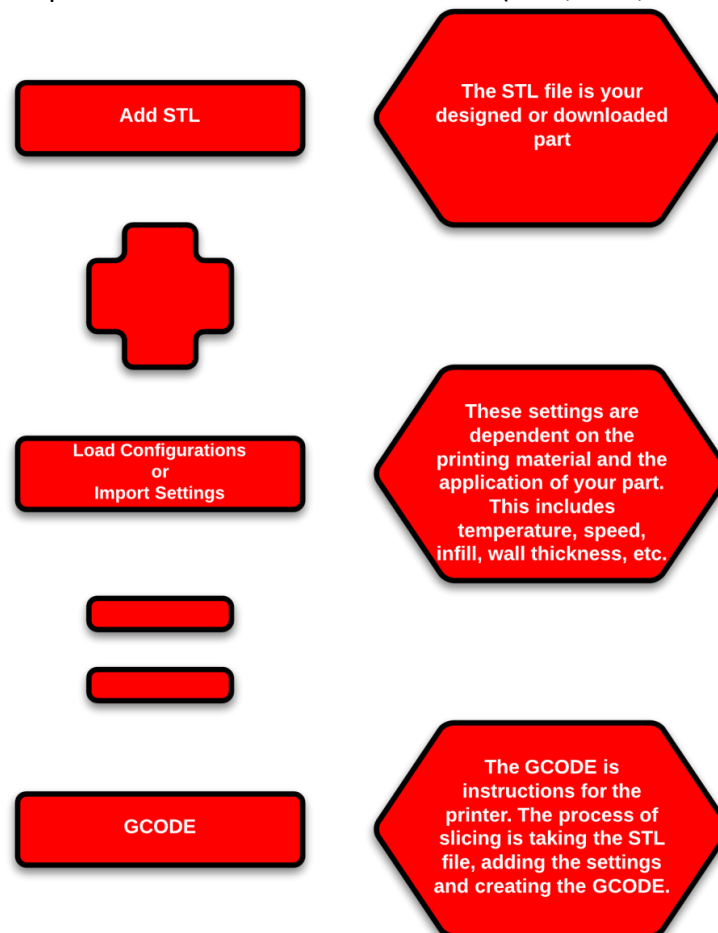
Some General Settings for the HD printer (these are already included in our preset settings):

Retraction Length = 7mm

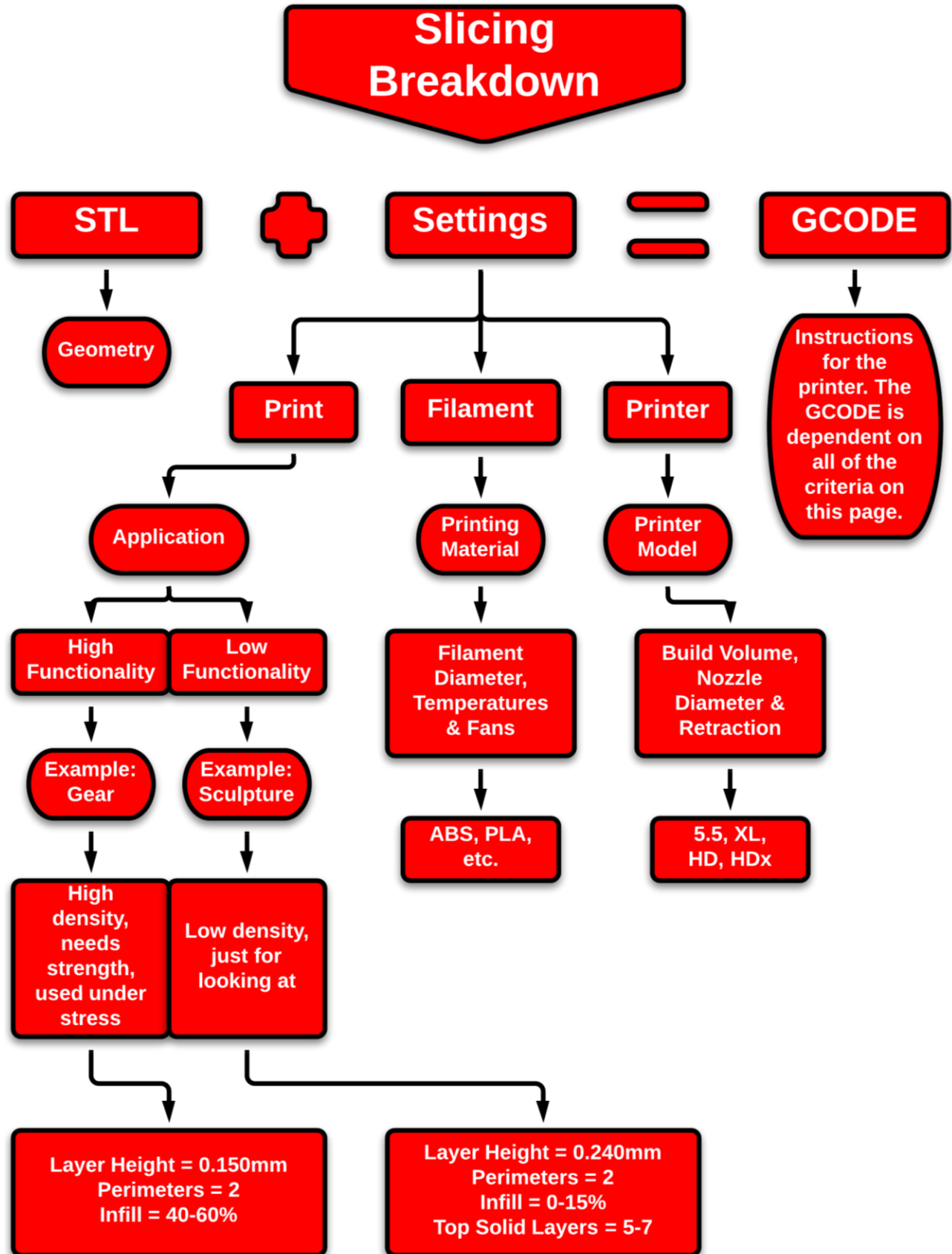
Retraction Speed = 50mm/s

HD Layer Height = a multiple of 0.03mm- ACME lead screw (0.06, 0.09, 0.12... up to 0.39mm)

HDx Layer Height = a multiple of 0.02mm – ACME lead screw (0.04, 0.06, 0.08... up to 0.40mm)

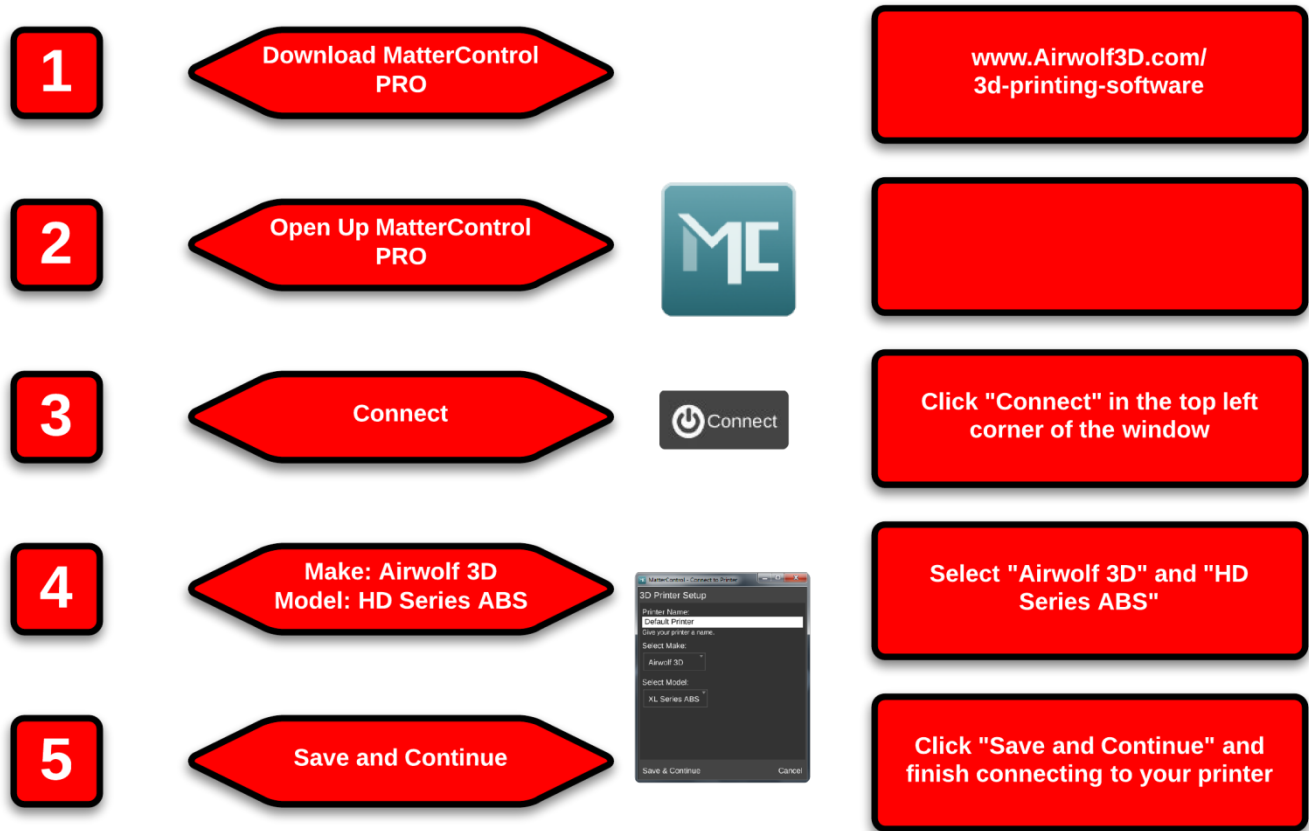


Slicing: Breakdown



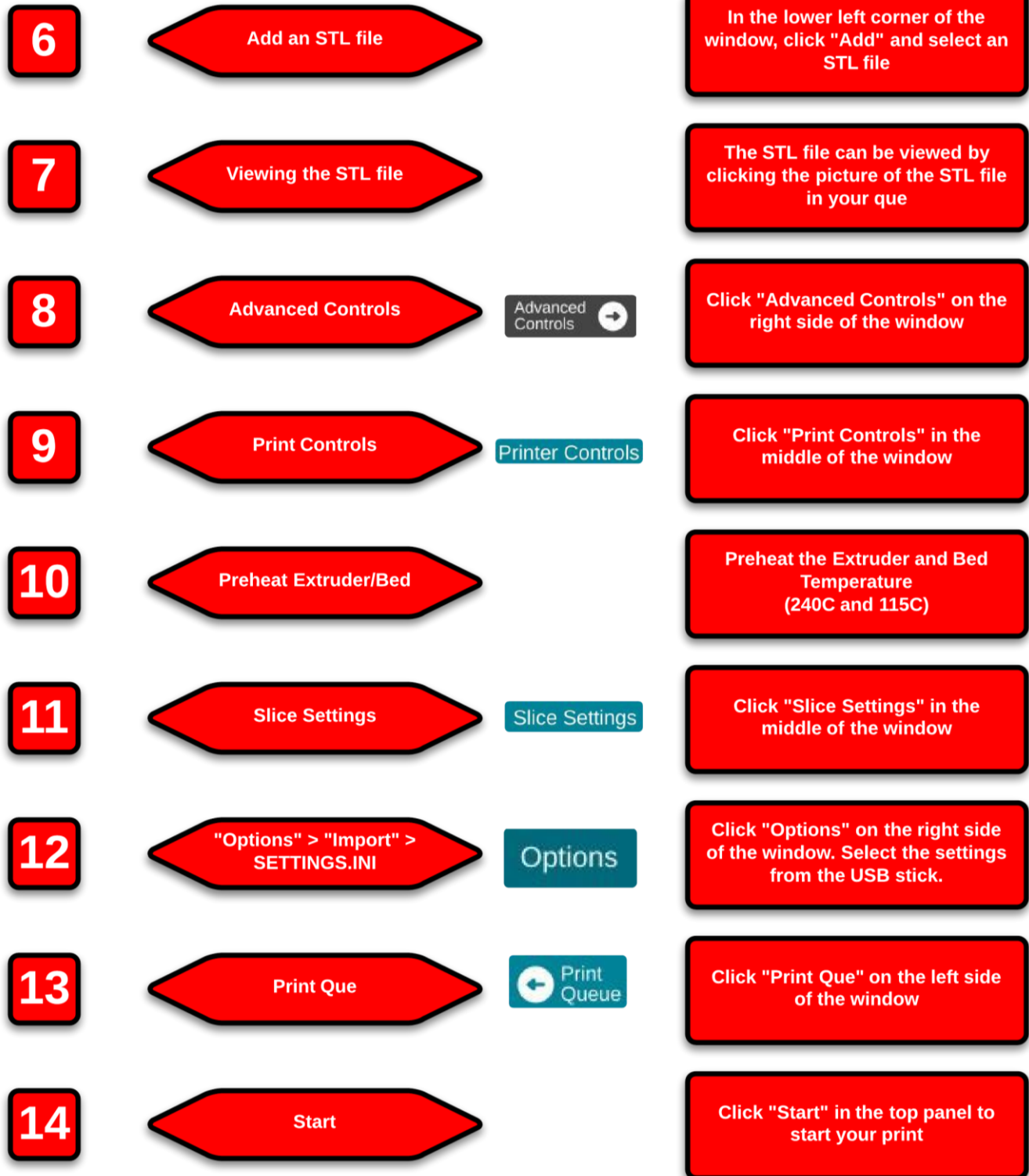
4-5) MatterControl PRO: How To

MatterControl Pro is an all-inclusive program used to “slice” your STL file and convert it into GCODE to run the printer. This includes steps 4 and 5 from the “5 Steps to Printing”. MatterControl Pro includes Slic3r and Cura as its slicing engines. From time to time, check to see if there is an updated version of MatterControl Pro. This can be done in the “Printer Que” slide, under the “About” tab. Make sure you are connected to the internet to have access to the new updated version.



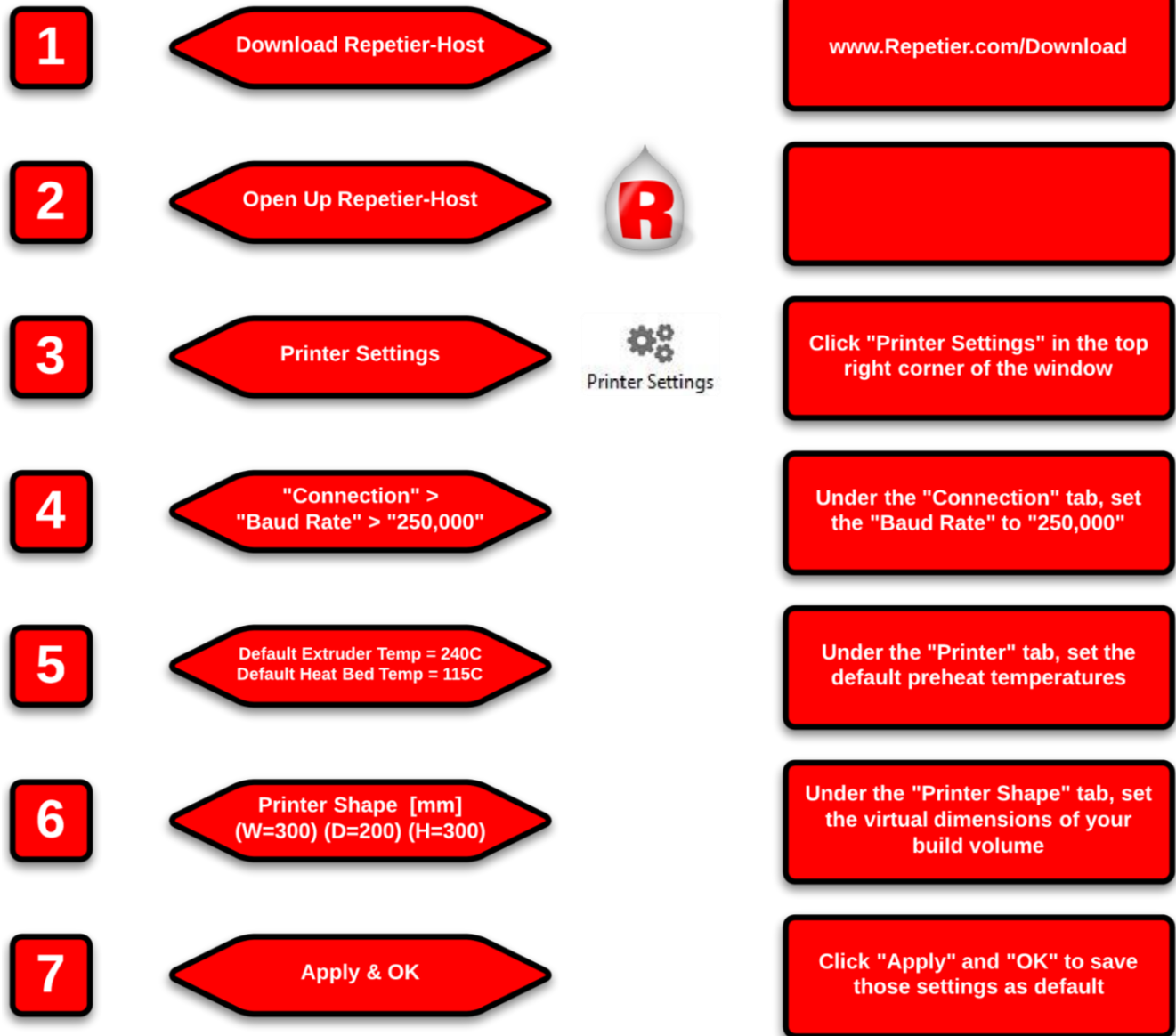
4-5) MatterControl PRO: How To (contd.)

Once you have connected to your printer, the next steps are to add an STL file, preheat the extruder and bed, and import slice settings. Export the GCODE from MC to an SD card to print from the VIKI.



4-5) Repetier Host: Setup

Repetier Host is a similar program to MatterControl PRO. It utilizes Slic3r as its slicing engine and is used to run the printer.



4-5) Repetier Host: The Basics

1

Printer Settings



Click "Printer Settings" in the top right corner of the window

2

Port: COM ##

Under the "Connection" tab, select the COM Port # for your printer

3

Apply & OK

Click "Apply" and "OK"

4

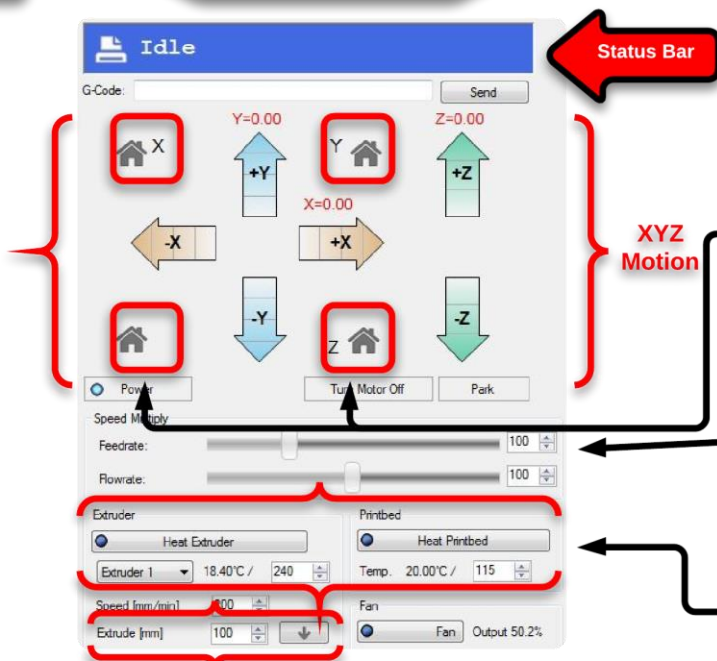
Connect



Click "Connect" in the top left corner of the window

5

Manual Control



Status Bar

The status bar will show what your printer is doing (preheating, printing, ETA)

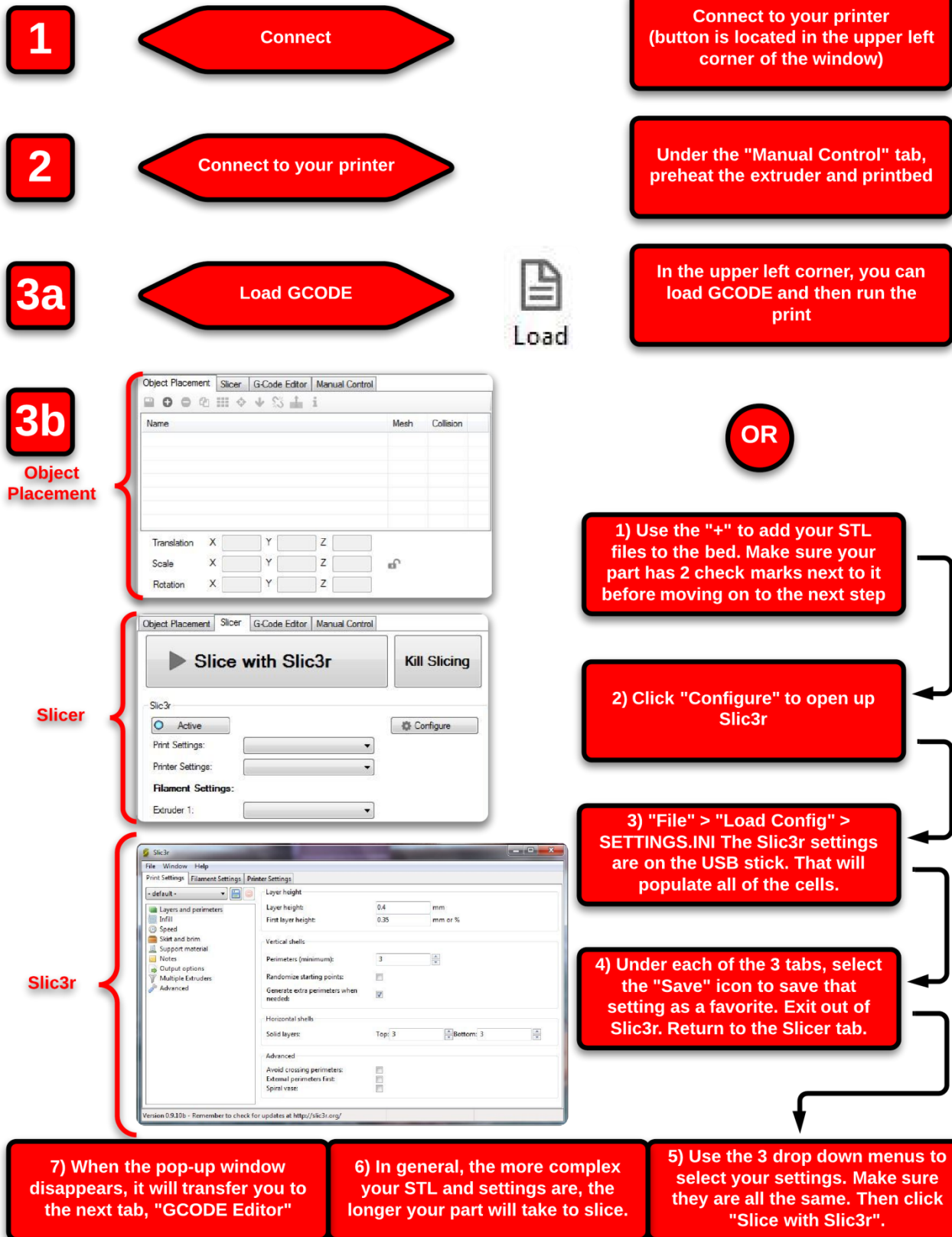
X, Y, Z, All Home. By using the min. limit switches

Feedrate is a Speed Multiplier. During your print, you can speed up or slow down your print

Preheat the Extruder and Printbed [Actual Temp / Target Temp]

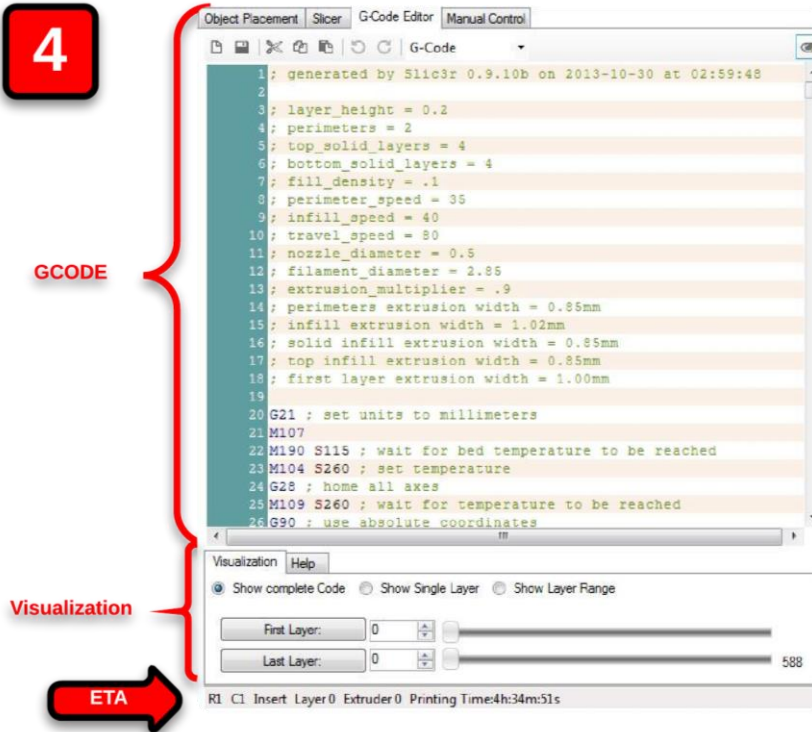
PERGE: Extruding 100mm of material will give you a full color or material change

4-5) Repetier Host: How To



4-5) Repetier Host: How To (contd.)

4



This is your GCODE. It displays a couple of the basic settings that were used to slice your STL.

As you gain experience, you will understand how each setting affects your print differently.

The visualization tab allows you to view all the layers, a single layer, or a range of layers

At the bottom, the GCODE will display an ETA. Certain settings will change the build time more than others

5

Run Job



Once you are satisfied with your slice, in the upper left corner of the window, click "Run Job"

Material Selection

Materials: Comparison

Here at Airwolf 3D, we do not manufacture 3D printing materials. Some companies we recommend would be Ultimachine and IC3D for ABS, MatterHackers for specialty material, Ninja Flex or Fenner Drives for TPE, and Taulman 3D for Nylon. When we construct new settings for new materials, they will be available to you, regardless of the time of your purchase. Send us an email and we will reply with the latest settings.

	Material	Positives	Negatives
1	ABS	Cheap, easy to use, very functional under stress and temps, and your printer is made out of ABS!	For extremely tall and long builds, the part may tend to crack in thin areas
2	PLA	Great surface finish; Great for tall vases or statues; Biodegradable	Harder to print with than ABS. Low strength; Degrades overtime when exposed to the elements
3	Soft-PLA	Flexible like rubber, relatively easy to print; Biodegradable	Degrades; great in compression, not so much in tension
4	PLA 4043D	Prints like PLA; Looks like ABS; Biodegradable	Degrades, low strength
5	Laywood	Prints like PLA, can have a smooth or rough finish	Degrades, low strength
6	Laybrick	Smooth or stone like texture; paintable; grindable	Brittle, low strength
7	HIPS	High Strength; Food Safe; Dissolves in Limonene; Great for support material	
8	PVA	Water Soluble; Great for support material; Easy to print	Expensive, harder to print than

Materials: Comparison (contd.)

	Material	Positives	Negatives
9	ColorFabb		
10	BendLay	Very Translucent and Elastic; Food Safe; Easy to Print;	
11	T-Glase	Food Safe; FDA Approved; Clear Crystal-like appearance; Easy to Print	
12	TPE	Very Strong & Flexible Elastomer; Rubber Like	Slow printing
13	Nylon-618*	Very High Strength	Some Difficulty Printing
14	Nylon-645*	Very High Strength	Some Difficulty Printing
15	Nylon-Bridge*	Very High Strength	Some Difficulty Printing
16	Polycarbonate*	Very High Strength and High Rigidity	Some Difficulty Printing

*For filaments 13-16, these are our advanced materials. We do not recommend using these filaments for our standard HD printer. The HDx comes standard with our JRx hot end. With this, you can now print with these higher temperature materials. The JRx can be added to our standard HD printer.

Materials: Settings

	Material	General Temps (c)	Heat Bed
1	ABS	Nozzle: 240-260 Bed: 110-130	Glass + PET Film; Purple Elmer's Glue Stick; NO Fans
2	PLA	Nozzle: 210-220 Bed: 60-70	Glass + Blue Painter's Tape; NO Goo; Fans (100%)
3	Soft-PLA	Nozzle: 210-220 Bed: 60	Glass + Blue Painter's Tape; NO Goo; Fans (100%)
4	PLA 4043D	Nozzle: 200-210 Bed: 60	Glass + Blue Painter's Tape; NO Goo; Fans (100%)
5	Laywood	Nozzle: 200-210 Bed: 60	Glass + Blue Painter's Tape; NO Goo; Fans (100%)
6	Laybrick	Nozzle: 200-210 Bed: 60	Glass + Blue Painter's Tape; NO Goo; Fans (100%)
7	HIPS	Nozzle: 220-240 Bed: 105-110	Glass + PET Film; Purple Elmer's Glue Stick; NO Fans
8	PVA	Nozzle: 195-210 Bed: 105-110	Glass + PET Film; Purple Elmer's Glue Stick; NO Fans

Switching Material: When switching material, it is very important that you purge the extruder of the old material. The reason is because there is still some of the old material left in the nozzle. First, heat up your nozzle to at least 240C. Then take out your old material. Next, insert your new material and use the "Extrude" feature to extrude at least 500mm for a full color or material change.

Materials: Settings (contd.)

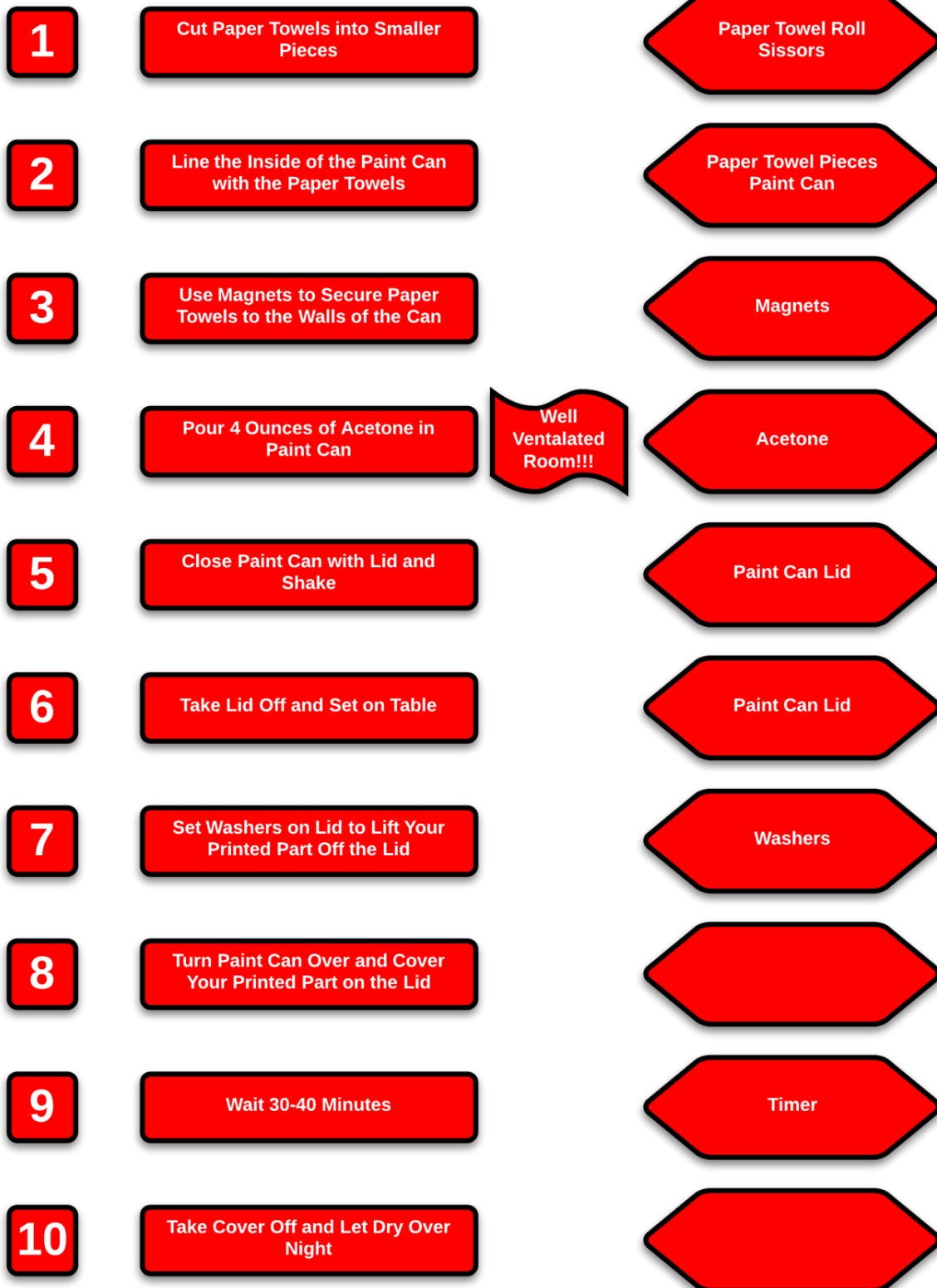
	Material	General Temps (c)	Heat Bed
9	ColorFabb	Nozzle: 240-250 Bed: 80	Glass + PET Film; Purple Elmer's Glue Stick; NO Fans
10	BendLay	Nozzle: 240-250 Bed: 70	Glass + PET Film; Purple Elmer's Glue Stick; NO Fans
11	T-Glase	Nozzle: 240-250 Bed: 60	Glass + PET Film; Purple Elmer's Glue Stick; NO Fans
12	TPE*	Nozzle: 235 Bed: 40	Glass + Blue Painter's Tape; NO Goo
13	Nylon-618**	Nozzle: 240-250 Bed: 60-70	Glass + PET Film; Purple Elmer's Glue Stick; NO Fans
14	Nylon-645**	Nozzle: 240-270 Bed: 60-70	Glass + PET Film; Purple Elmer's Glue Stick; NO Fans
15	Nylon-Bridge**	Nozzle: 240-270 Bed: 70-80	Glass + PET Film; Purple Elmer's Glue Stick; NO Fans
16	Polycarbonate**	Nozzle: 310-315 Bed: 135	Glass + PET Film; Purple Elmer's Glue Stick; NO Fans

*For TPE, use PLA to purge the printer of the old material. Once you have purged the printer with at least 500mm of material, then pull the PLA out and print with the TPE. See our website for a full video for printing with TPE. **For filaments 13-16, these are our advanced materials. We do not recommend using these filaments for our standard HD printer. The HDx comes standard with our JRx hot end. With this, you can now print with these higher temperature materials.

Post Processes

Acetone Vapor

(PROCEED WITH CAUTION)



Two Part Epoxy

A company called BJB makes an epoxy called TC-1614 A/B. This is what is referred to in the following instructions. More detailed instructions are included with the purchase of their epoxy.

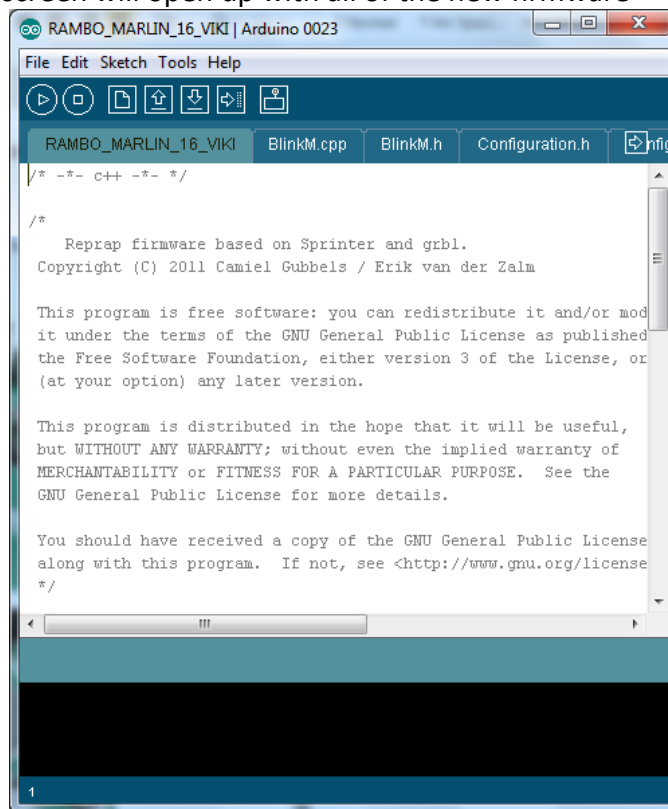


Firmware

The HD firmware is called “Marlin_HD_23”. (If there is something newer, use it instead) The firmware is the communication link between the software and hardware. The firmware is already uploaded to your printer. Only alter the firmware if you have past experience or are directed to by an Airwolf technical assistant.

Before Starting- make sure your power cable is plugged into the wall and the printer is plugged into your computer. Also you must have the printer turned on and that your RAMBo driver is installed correctly.

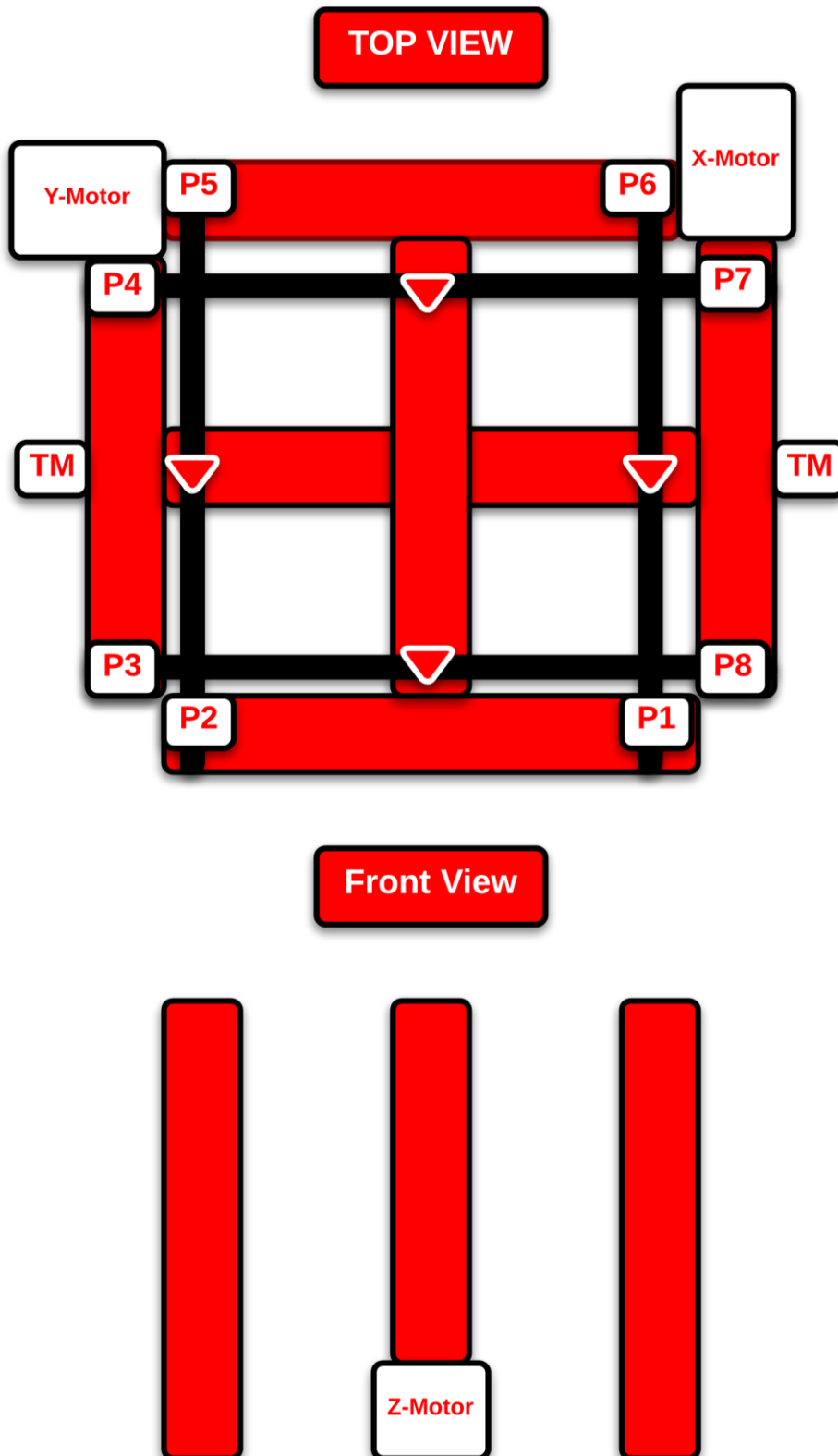
- 1) Download everything from the firmware folder either on the USB stick or the Dropbox link.
- 2) Install Arduino 0023
 - a. If you already have it, make sure you have the “LiquidTWI2” folder in “Libraries”
- 3) Open up Arduino 0023
 - a. If it asks you to install the latest version, select “NO”
- 4) Select File > Open > Marlin_HD_23 and select the “Configuration.h” file
- 5) A new screen will open up with all of the new firmware



- a.
- 6) Select Tools > Board > Arduino Mega 2560
- 7) Select Tools > Serial Port > COM##
 - a. Select the COM Port for your HD printer. If you are not sure which COM## it is, it can be found in the “Devices and Printers” section of your computer.
- 8) There are 7 buttons at the top, select the 6th one labeled “Upload”. If done correctly, it should say “Done Uploading”.

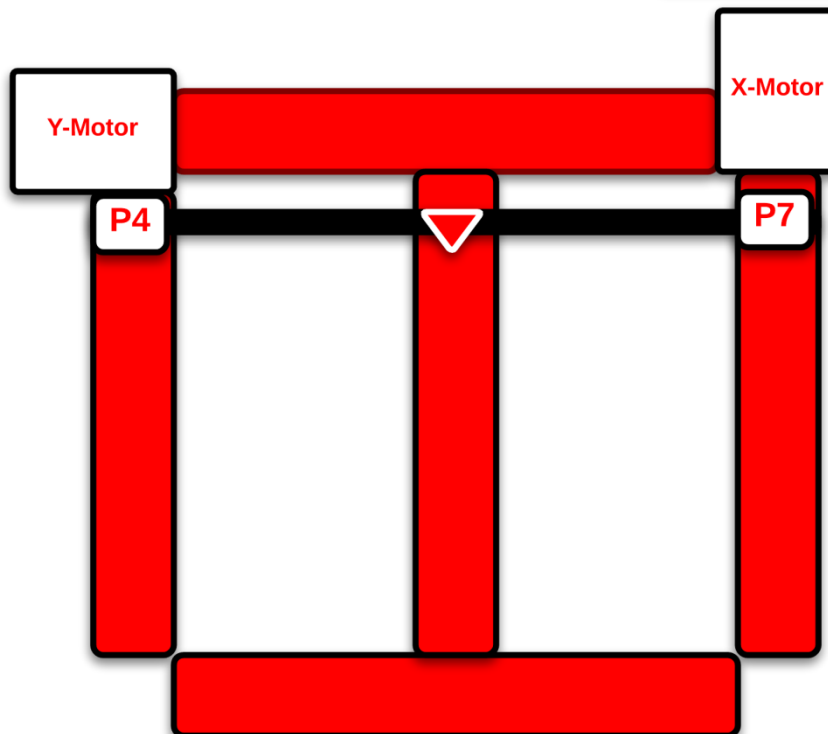
Maintenance

Here at Airwolf 3D, we have already done these maintenance steps. These should be done periodically (about every 100 printing hours) to ensure a working printer.



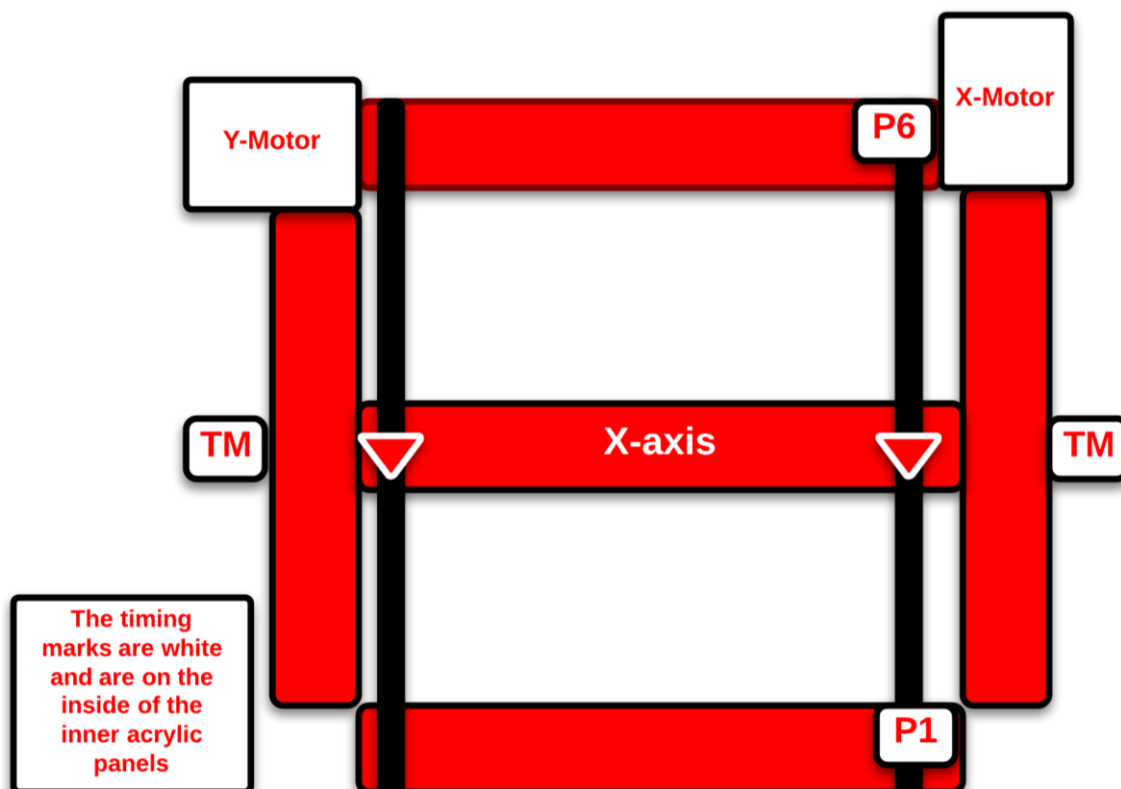
Belt Tensioning

1	Check Belt Tension	Flick belt; it should sound like a cello or low deep tone; if it does, do not adjust belt tension
2	Loosen Pulley	Use 2mm allen wrench to loosen ONE pulley [P7] (2 set screws)
3	Equalize Belt Tension	Move hot end left and right to equalize the tension in the top and bottom of the belt
4	Tighten Belt	Tighten the belt with a belt adjuster screw [triangle] (DO NOT OVER TIGHTEN)
5	Check Belt Tension	Flick belt; it should sound like a cello or low deep tone
6	Tighten Pulley	Use 2mm allen wrench to tighten the set screws on pulley [P7] (2 set screws); repeat on other belts (x3)



Rod Alignment (X-axis)

- 1** Check X-axis Alignment
Align the left end of the rod with the timing mark [TM] and check the right side in relation to its timing mark
- 2** Loosen Pulleys
On the right end, use the 2mm allen wrench to loosen both pulleys [P1 & P6] (4 set screws)
- 3** Align the Rod
Adjust the right end of the rod to align with its timing mark
- 4** Tighten Pulleys
Use the 2mm allen wrench to tighten both pulleys [P1 & P6] (4 set screws)
- 5** Re-Check the X-axis Alignment
Move the hot end forward and backward; check to see that the x-axis is aligned with the timing marks [TM]



Rod Alignment (Y-axis)

1

Check Y-axis
Alignment

Move the Y-axis rod to the right
so the front end of the rod is 1
inch from the pulley [P1]

2

Loosen Pulleys

On the back end, use the 2mm
allen wrench to loosen both
pulleys
[P4 & P7] (4 set screws)

3

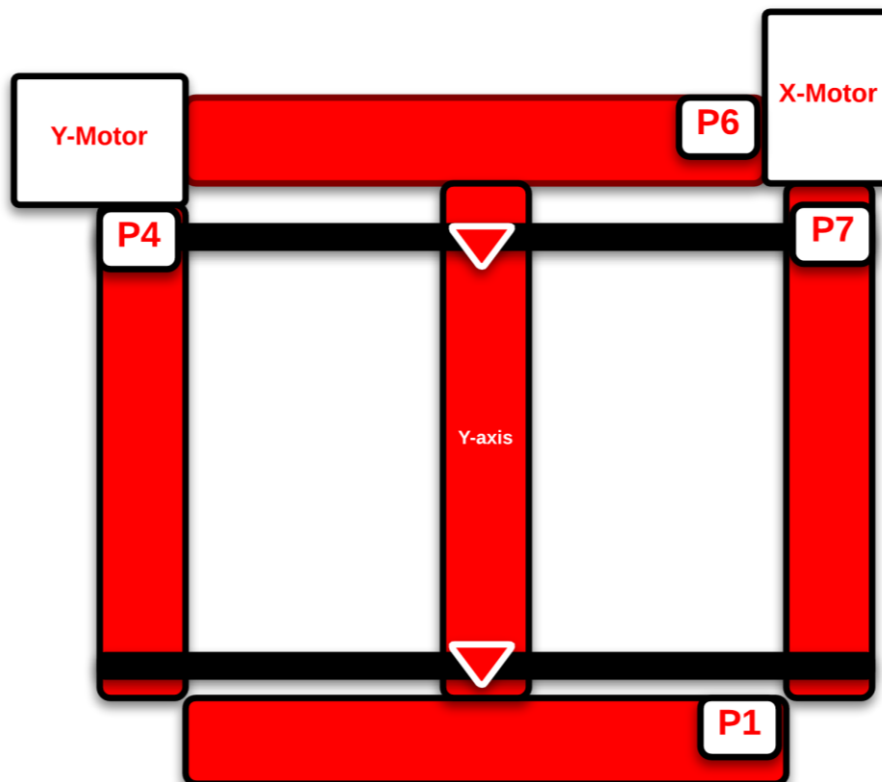
Align the Rod

Adjust the back end of the rod to
align so that it is 1 inch from its
pulley
[P6]

4

Tighten Pulleys

Use the 2mm allen wrench to
tighten both pulleys
[P4 & P7] (4 set screws)



Rod Lubrication

Use Lithium Grease on all the metal rods (labeled in red) on the first maintenance page.



Inserting Filament

At first, you may have trouble inserting the filament. There are two usual places where this may occur; the extruder motor/hobbed bolt or the top of the hot end.

If you are having trouble inserting the filament into the extruder motor, pull the pin, and release the spring loaded latch. Next, slide the white plastic adapter out to the right. Then fish the filament through the extruder motor from the bottom and manually feed it into the white adapter. Reinsert the white adapter into the extruder assembly and close the spring loaded latch. Finally insert the pin.

If you are having trouble inserting the filament into the top of the hot end, you will need to remove the retaining ring. Use a pair of pliers to remove the retaining ring and carefully pull out the PFT tube. Manually insert the filament into the white adapter and reinsert the PFT tube. Finally reattach the retaining ring back onto the white adapter.

Wiring Diagram

Motor- X, Y, Z1, and E0

Limit Switches- All 6 limit switches are used

Heaters- Heat 0 is for the extruder (heating element), Heat2Bed is for the heat bed

Thermistors- T0 is for reading the extruder temperature, T2 is for reading the heat bed temperature

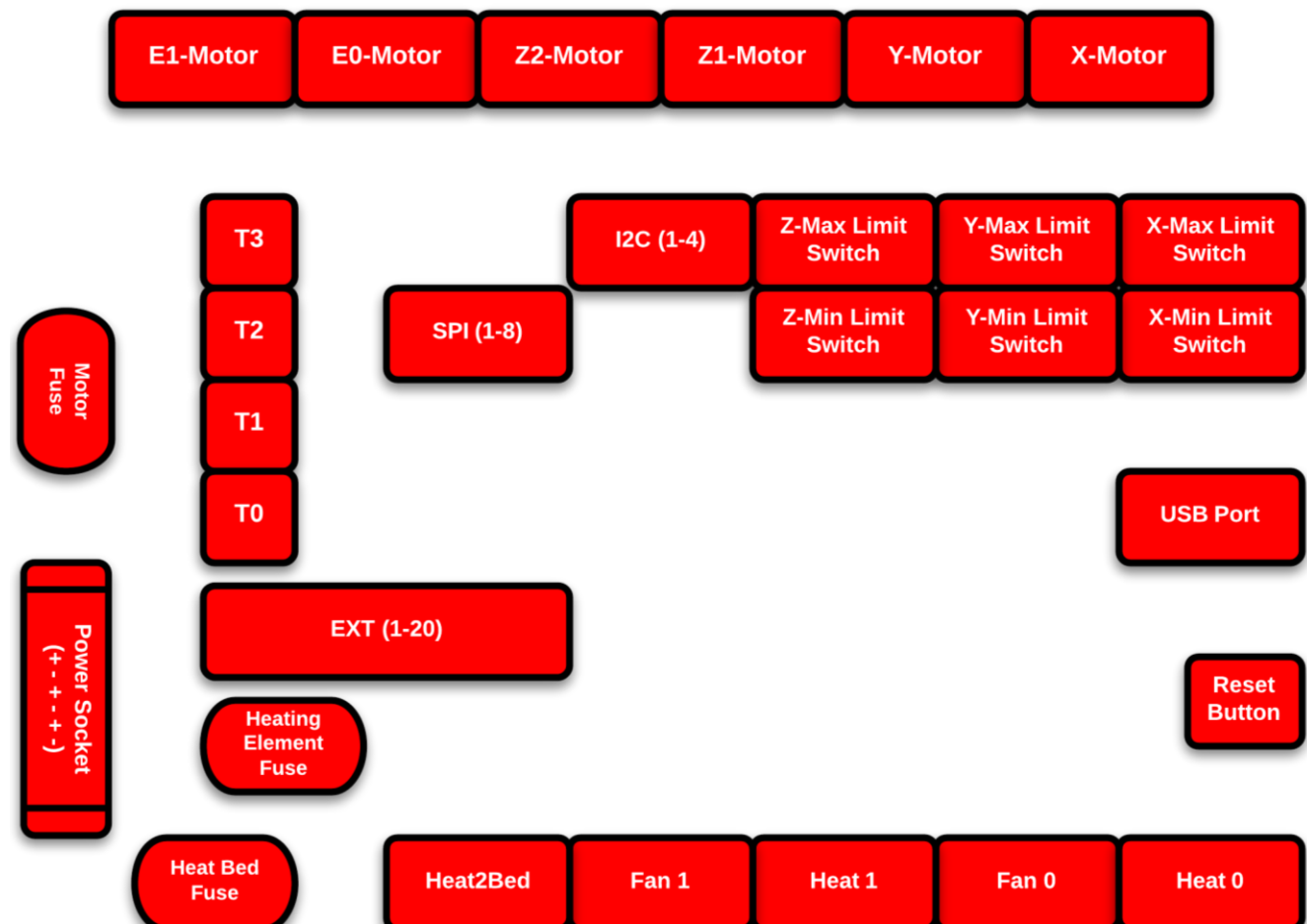
Fans- Fan 0 is for the extruder fan, Fan 1 is for the LED lights

Fuses- Motor Fuse, Heating Element Fuse (Nozzle), and Heat Bed Fuse

RAMBo has 3 replaceable fuses. The small white fuse holders are Little Fuse OMNI-BLOCK fuse holders. They are compatible with NANO2 Fuses. Fast or very fast acting are recommended. An example part number for replacement fuses is 0448005.MR

VIKI LCD Screen- I2C(1-4), SPI(3-6), EXT(13, 15, 17, 19)

VIKI WIRING		
Wire Color	Cable A	Cable B
RED	I2C 4	SPI 6
BLACK	I2C 3	EXT 13
BLUE	I2C 2	SPI 4
GREEN	I2C 1	SPI 5
YELLOW	EXT 17	EXT 15
WHITE	EXT 19	SPI 3



Clogged Nozzle

There are 3 main reasons that your nozzle may seem clogged. 1) The nozzle was not heated enough to melt the filament. 2) The nozzle was too close to the glass on the first layer. 3) You may have run some low quality filament through the hot end. This would include inconsistent filament diameters and/or debris in the material.

1

Heat the nozzle to 260C

2

Extrude at least 100mm of material

3

If nothing is coming out you may have a mouse bite

4

Release the extruder latch and remove the filament

5

Examine the material, if there is a mouse bite, cut the end off and re-insert the material

6

Close the extruder latch and re-heat the nozzle to 260C

7

Extrude at least 100mm of material

8

If material still doesn't extrude, proceed to the next section "Changing the Nozzle"

Changing the Nozzle

1

Heat the nozzle to 260C

2

Release the extruder latch and remove the material from the hot end

3

Move the heat bed at least 6 inches down, away from the nozzle

4

Put on a pair of gloves

5

Use a pair of pliers to hold the heating element steady

6

Use a 2mm allen wrench to unscrew the two screws on the face of the nozzle

7

Set the nozzle, 2 screws and 2 lock washers aside *** Remember, these parts are HOT!

8

Setting the nozzle in a cup of acetone can help clear any material stuck in the nozzle (over night)

9

Use a razor blade to scrape any extra material away from the bottom of the heating element

10

Heat the nozzle to 100C and reassemble the nozzle with the 2 screws and 2 lock washers

Trouble Shooting

- 1) How do I convert my Sketchup file into an STL file?
 - a. Use the plug-in and instructions from our website. <http://airwolf3d.com/wiki/how-to-convert-and-export-google-sketchup-skp-files-to-stl-for-3d-printing/>
- 2) What settings do I use for the printer?
 - a. Use the configurations that are on the USB stick. Configurations are a group of settings (speeds, temperatures, resolution, etc.)
- 3) What are some general settings for all prints?
 - a. First Layer Height = .4mm
 - b. ABS Temperatures: (Nozzle \geq 240C) (Bed \geq 115C)
- 4) How do I adjust the Z height?
 - a. The M3 screw on the back left corner of the top panel. This screw sets the first layer height.
- 5) What do I do if my part isn't sticking?
 - a. Make sure to use the purple elmer's glue stick on your glass. The rule of thumb is to print until your part does not stick; then apply the mixture. Also, increasing the bed temperature slightly should help with the stick.
- 6) What filament diameter can be used on the printer?
 - a. ONLY 3mm diameter filament (2.85mm \pm 0.10mm measured) can be used on the printer.
- 7) There are fans next to my nozzle. When do I use them?
 - a. Fans are NOT needed for ABS. PLA is a material that needs fans. If you use the configurations that we give you, fans (ON/OFF) are already accounted for when you slice your STL file. The fans are only on at 100% (255), anything lower than this will cause the fans to turn off.
- 8) I think my nozzle is clogged. How do I know and what do I do if it is clogged?
 - a. To see if your nozzle is clogged, raise your nozzle at least 3 inches vertically off the bed. Make sure the nozzle is up to temperature (ex. ABS \geq 240C). Use the extrude feature to extrude 100mm of material.
 - b. If that doesn't work, you may have a "mouse-bite". A mouse-bit usually occurs when the nozzle is too close to the glass on the first layer. To see, release the extruder latch on the back panel and pull the filament out. If there is, cut it off below the mouse-bite and reinsert the filament into the extruder.
- 9) I can't seem to connect to my printer. What should I do?
 - a. One quick way to solve electrical connecting problems is to unplug the printer from your computer, turn off the printer and hit the reset button. This clears the circuit board of any errors. With further software issues, please refer to our website, contact us or MatterHackers.
- 10) Once I have run out of my first roll of material, where can I buy more?
 - a. Congrats! That's a lot of material! We do NOT sell material. For ABS, our best recommendation is Ultimachine or IC3D. They make incredible ABS that is tested, roll by roll. MatterHackers is a great source for PLA and specialty filament. Also refer to the materials section of this manual.

Glossary of Terms

Hardware

Filament- Spool of printing material

Spool Minder- Printed part that holds the spool of material

Heat Bed- The red plate that heats the glass

PET Film- The green film on top of the glass for ABS

Z-Adjustment Screw- Used to adjust the first layer height; located on the top panel in the back left (spring loaded)

Extruder- The sub assembly that feeds the material from the spool to the heated nozzle

Guide Tube- The clear tube that feeds the material from the extruder motor to the hot end

Hot End- The “print head”

Heating Element- The metal block in the extruder that houses a resistor and heats the nozzle

VIKI LCD Screen- On the front panel. Used to control the motion, temps and run prints

Home- Uses the minimum limit switches to find a point of reference (the origin)

X-axis- Left and right

Y-axis- Front and back

Z-axis- Up and down

Software

STL- The geometry file of the 3D model

Netfabb- A program used to clean up STL files

Configurations/Profiles/Settings- Settings used to slice the STL file and create GCODE

Slicing- Taking an STL file, adding settings and creating GCODE

Slic3r- A program that slices STL files and creates GCODE, uses “configurations”

Cura- A program that slices STL files and creates GCODE, uses “profiles”

MatterControl- Developed by MatterHackers. Utilizes Slic3r and Cura to create GCODE. Also used to run GCODE.

Repetier-Host- Utilizes Slic3r to create GCODE. Also used to run GCODE.

Contact List

Airwolf 3D

(949) 478-2933

(949) 478-AW3D

130 McCormick (Suite 112)

Airwolf3D.com

Costa Mesa, CA 92626

General Information: info@airwolf3d.com

Transactional Support: eva@airwolf3d.com

Technical Support: tech@airwolf3d.com

Notes

[illegible]

Notes

[illegible]