AW3D *HDL*

User Manual

www.Airwolf3D.com

Table of Contents

Specifications

General	Designed and Manufactured In:	USA	
	Build Volume (LxWxH)	12 x 8 x 11	inch
		300 x 200 x 280	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:	PLA, Soft PLA, PLA 4043D, BendLay, LayWOO D-3, Laybrick, HIPS, T-Glase, PVA, 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	0.1	mm
		0.0039	inch
	Max Speed-Perimeter	100	mm/s
		3.94	inch/s
	Max Speed-Travel	200	mm/s
		7.87	inch/s
Hard	Circuit Board	RAMBo	
	Frame	Partially Enclosed Acrylic	
	Power Supply	Internal auto-switching 13.5V DC, 300W	
Comp.	File Format	STL & GCODE	
	Software	Netfabb & MeshLab	
		Cura & Slic3r	
		Matter Control	
	System Compatibility	Windows, Mac, Linux	

Unboxing the Printer

Box Contents

Airwolf 3D HD Printer

User Manual

Spool of ABS Filament

Glass with PET Film

PET

USB Drive

USB

Micro SD Card

MICRO SD

Allen Wrenches

4mm 3mm 2.5mm 2mm

ABS-Acetone Mixture

Part Directory

Guide Tube

E-Motor

X-Motor

Z-Adjustment Screw

Hot End

Y-Motor

Filament Spool

USB Connection

Filament Hole

Spool Minder

Filament Spool
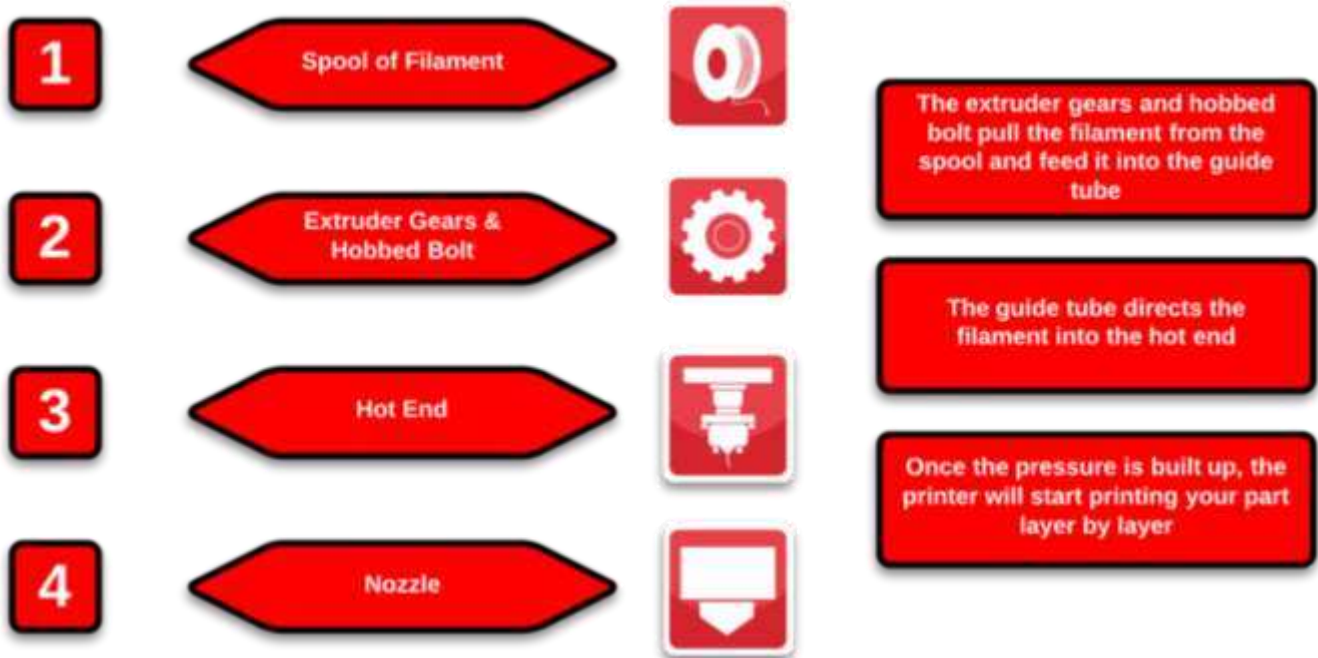
Z-Motor

Warnings!!!

DANGER

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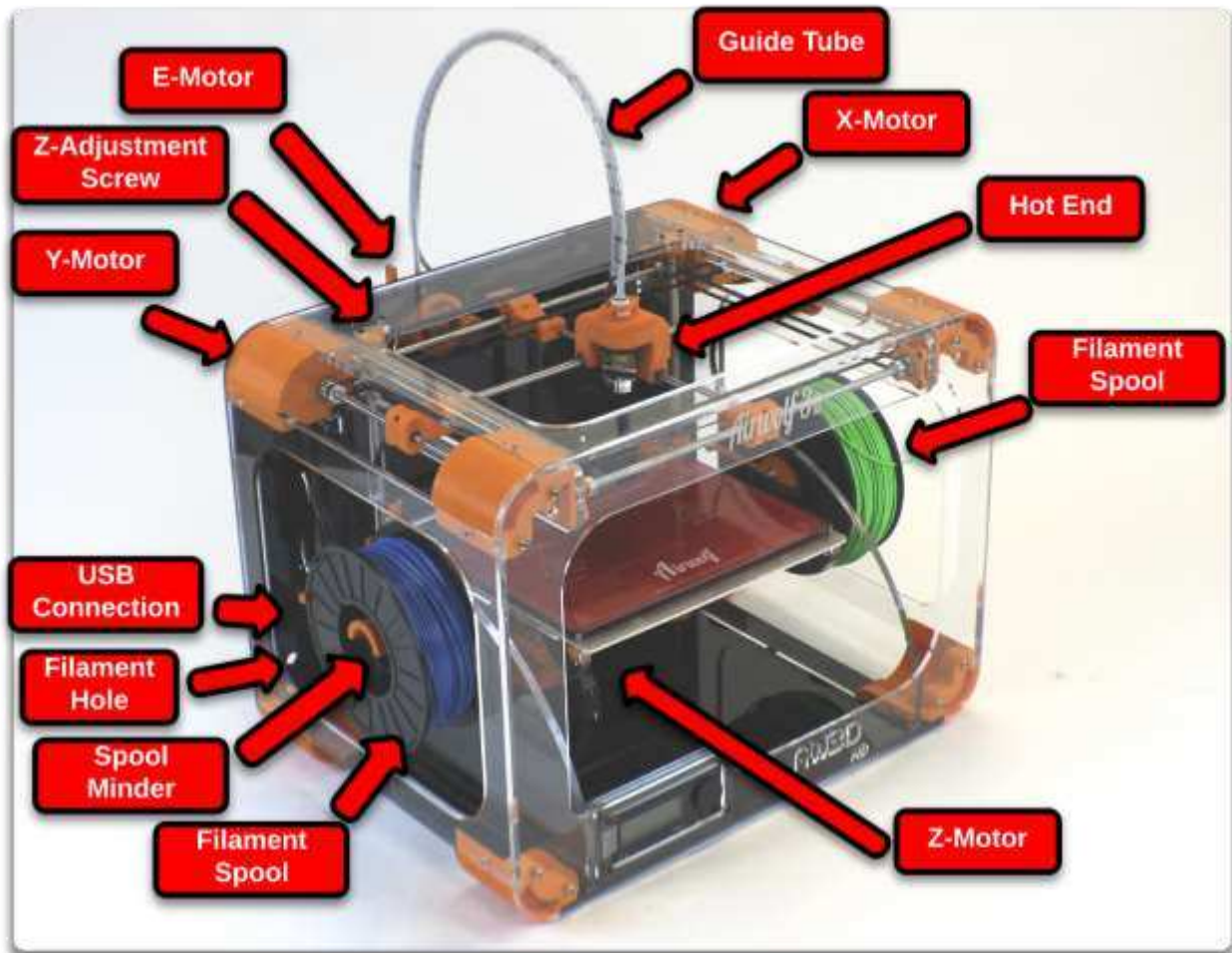
1) Don't touch the Hot End while it's hot

2) Pinching Hazard (gears, pulleys, belts)

How it Works

Extruder Assembly

1	Spool of Filament	The extruder gears and hobbed bolt pull the filament from the spool and feed it into the guide tube
2	Extruder Gears & Hobbed Bolt	The guide tube directs the filament into the hot end
3	Hot End	Once the pressure is built up, the printer will start printing your part layer by layer
4	Nozzle	

Hot End and Bed Motion

The nozzle moves in the X-Y plane. The heat bed moves along the Z-axis.

+Y

-X   **Nozzle**   +X

-Y

-Z   +Z

Bed

When a print is started the nozzle will go HOME. HOME is found by the min limit switches. The bed will be at its highest point and nozzle will be in the front left corner. Next it will check the nozzle temperature according to the GCODE before printing. And lastly, the printer will print a brim around where your part will be printed. There are two reasons for this. One is to prime the nozzle for your part. And two is to make sure your bed is level. Then it will start building your part, layer by layer.

Plug In & Turn On

Airwolf 3D HD Printer

Wall Socket

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.

Power Switch

Fuse

Power Plug
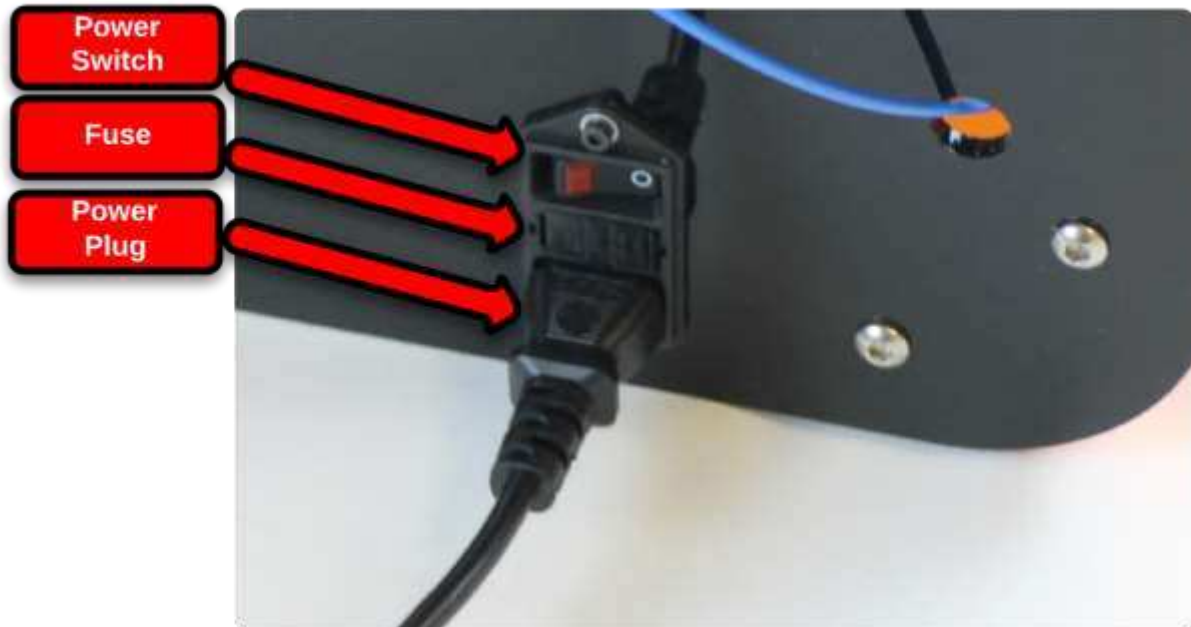
Pre-Printing Steps

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
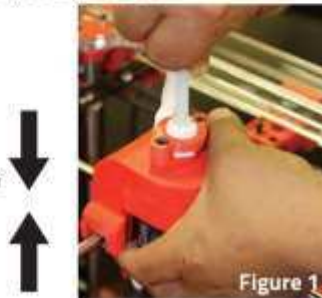
4 CLAMP THE EXTRUDER ASSEMBLY AND REMOVE THE PIN
Use one hand to clamp the spring loaded extruder assembly. Use the other hand to remove the pin. Keep this pin nearby for use in a later step.

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**
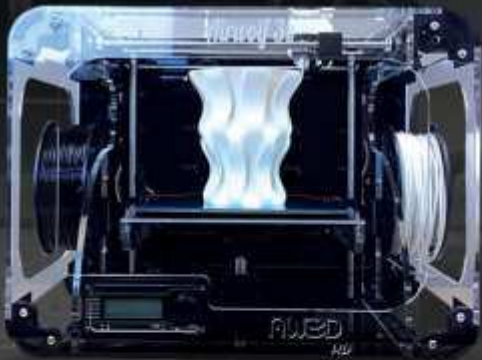
Airwolf 3D

1) Leveling the Bed

Why Level the 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 bed only moves slowly in the Z direction. This means that the 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 Bed- There are two parts to leveling the bed. The first part is to make sure that the XY plane (the plane that the nozzle moves in) is parallel to the bed (glass). This is achieved by the four small M3 screws in each corner of the bed. The second part is the first layer height. This is achieved by the Z-adjustment screw and limit switch subassembly for the 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.

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
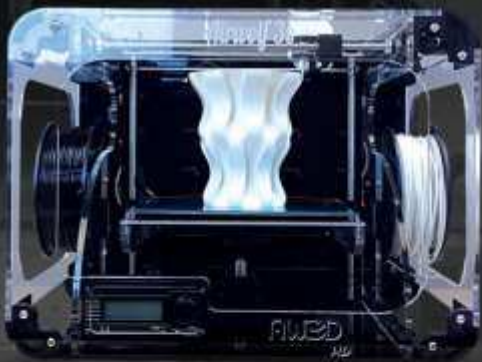

Airwolf 3D
HD
FIRST PRINT/CALIBRATION

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 deosn'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 deosn'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 heatbed. 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.

Correct

Incorrect

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 ± 0.10mm 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) Bed Preparation

For PLA prints, it will be important to use the masking tape on the glass. This will help your parts stick to the glass during the duration of the print.

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 there for 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.

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".
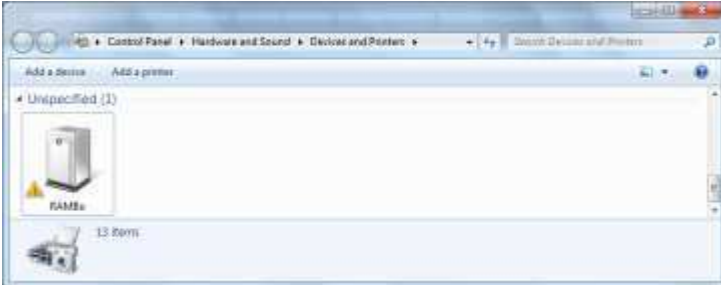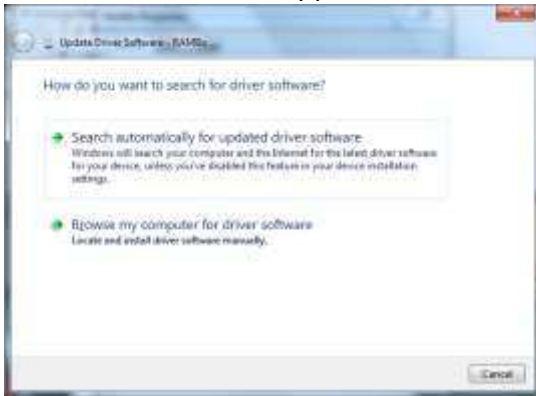

4) A second window will appear. Under the second tab labeled "Hardware", select "Properties".
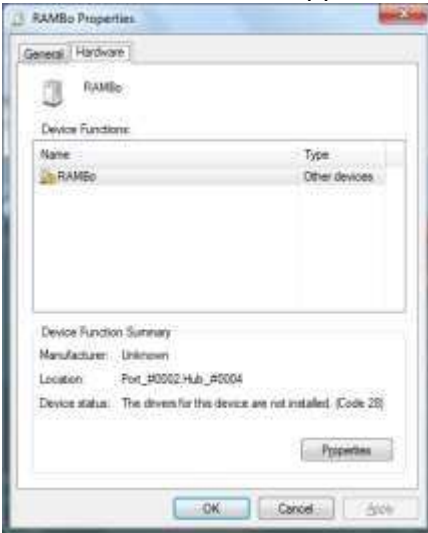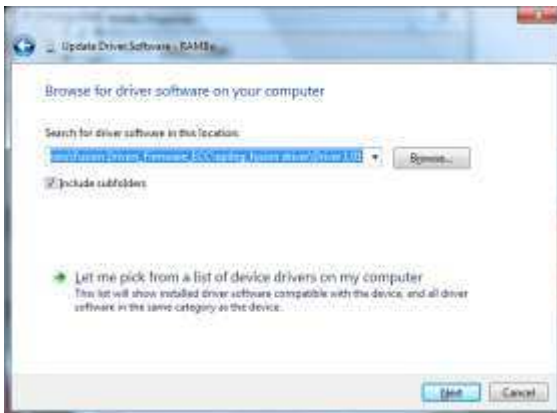

5) A third window will appear. Under the first tab labeled "General", select "Update Driver".
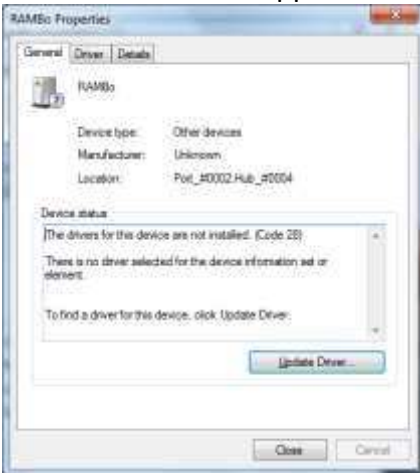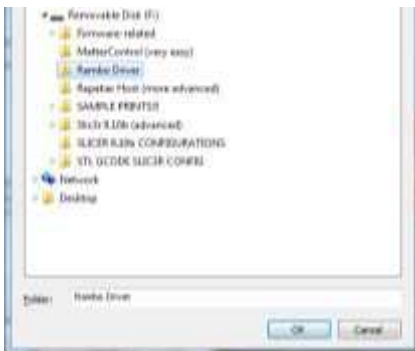
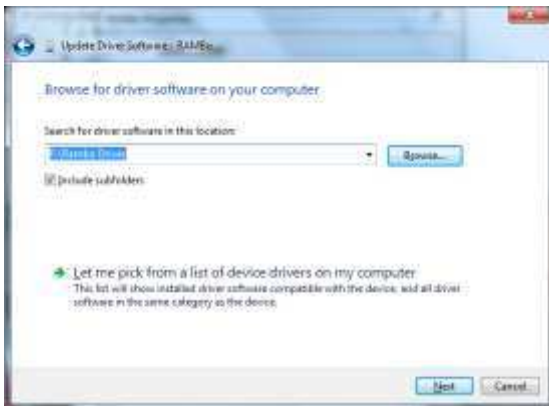6) A fourth window will appear. Select "Browse my computer for driver software".

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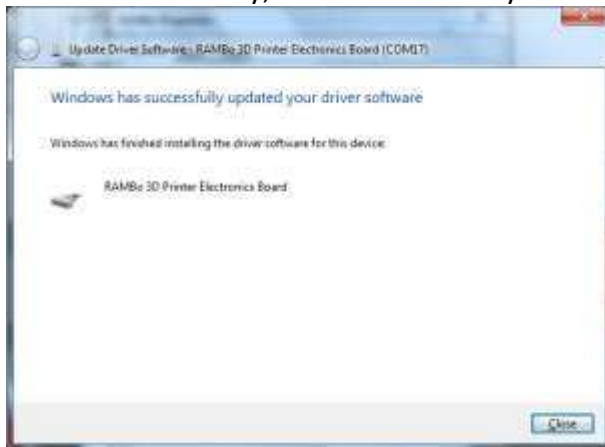
12) Now in your "Devices and Printers", under "Unspecified", this is the screen you should see. The "(COM17)" is dependent on what USB port you plugged the printer into. So yours may display a different COM number.

5 Steps to Printing

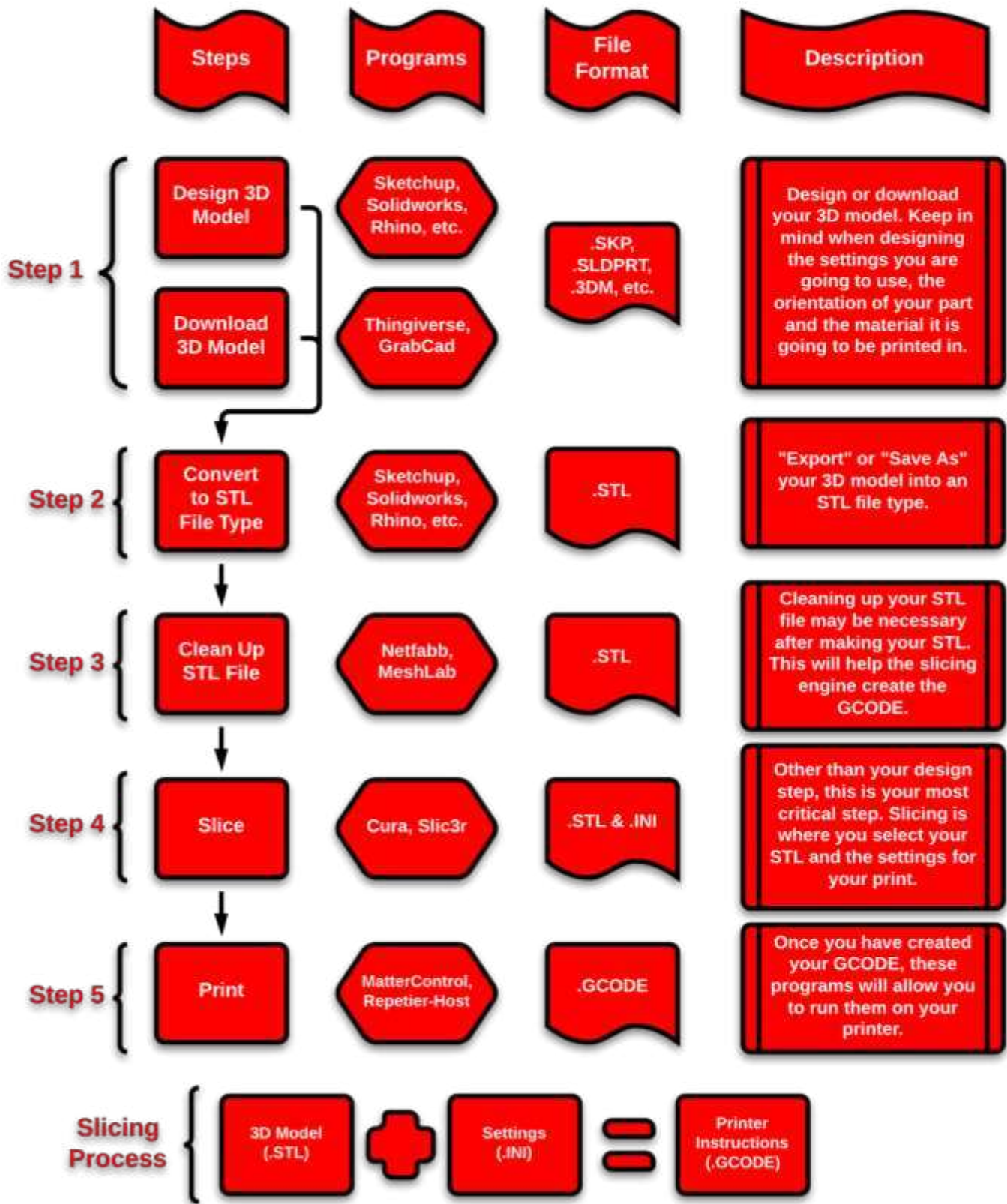| Steps	Programs	File Format	Description
Step 1 — Design 3D Model / Download 3D Model	Sketchup, Solidworks, Rhino, etc. / Thingiverse, GrabCad	.SKP, .SLDPRT, .3DM, etc.	Design or download your 3D model. Keep in mind when designing the settings you are going to use, the orientation of your part and the material it is going to be printed in.
Step 2 — Convert to STL File Type	Sketchup, Solidworks, Rhino, etc.	.STL	"Export" or "Save As" your 3D model into an STL file type.
Step 3 — Clean Up STL File	Netfabb, MeshLab	.STL	Cleaning up your STL file may be necessary after making your STL. This will help the slicing engine create the GCODE.
Step 4 — Slice	Cura, Slic3r	.STL & .INI	Other than your design step, this is your most critical step. Slicing is where you select your STL and the settings for your print.
Step 5 — Print	MatterControl, Repetier-Host	.GCODE	Once you have created your GCODE, these programs will allow you to run them on your printer.

Slicing Process: 3D Model (.STL) + Settings (.INI) = Printer Instructions (.GCODE)

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. PLA is a great material to start with as it has a large printing range.

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
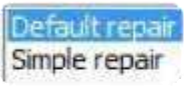
3) Netfabb: Layout (STL Clean Up)

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.
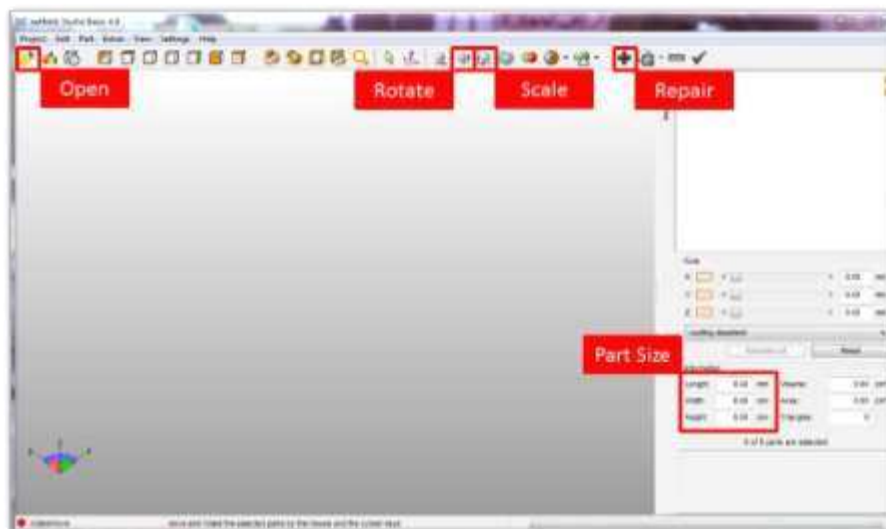
3) Netfabb: How To (STL Clean Up)

1 — Download Netfabb Basic — www.Netfabb.com

2 — Open Up Netfabb Basic — netfabb

3 — "Project" > "Open" > PART.STL — Open your STL file. Check to make sure your part is the correct size.

4 — "Part" > "Rotate Part" — Rotate your part into the orientation that you want to print it in

5 — "Extras" > "Repair Part" — This will begin the short cleaning process

6 — Automatic Repair — Automatic repair — Click "Automatic Repair" in the lower right corner of the window

7 — Default Repair — Default repair / Simple repair — A new window will appear. Select Default Repair.

8 — Execute — Execute — Click "Execute" and return to the main window

9 — Apply Repair — Apply repair — This will finalize the cleaning of your STL. Select YES to remove old part.

10 — "Part" > "Export Part" > "as STL" — Make sure your file type is STL

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. It takes the GCODE and runs the print with it.

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.

HDL Layer Height = ranges from 0.10mm to 0.40mm

Slicing: Breakdown

Slicing Breakdown

STL ✚ **Settings** ═ **GCODE**

STL → Geometry

Settings → Print, Filament, Printer

GCODE → Instructions for the printer. The GCODE is dependent on all of the criteria on this page.

Print → Application

Application → High Functionality, Low Functionality

High Functionality → Example: Gear → High density, needs strength, used under stress

Low Functionality → Example: Sculpture → Low density, just for looking at

Filament → Printing Material → Filament Diameter, Temperatures & Fans → ABS, PLA, etc.

Printer → Printer Model → Build Volume, Nozzle Diameter & Retraction → 5.5, XL, HD, HDx

Layer Height = 0.150mm
Perimeters = 2
Infill = 40-60%

Layer Height = 0.240mm
Perimeters = 2
Infill = 0-15%
Top Solid Layers = 5-7

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.

1	Download MatterControl PRO		www.Airwolf3D.com/ 3d-printing-software
2	Open Up MatterControl PRO	MC	
3	Connect	Connect	Click "Connect" in the top left corner of the window
4	Make: Airwolf 3D Model: HD Series ABS		Select "Airwolf 3D" and "HD Series ABS"
5	Save and Continue		Click "Save and Continue" and finish connecting to your printer
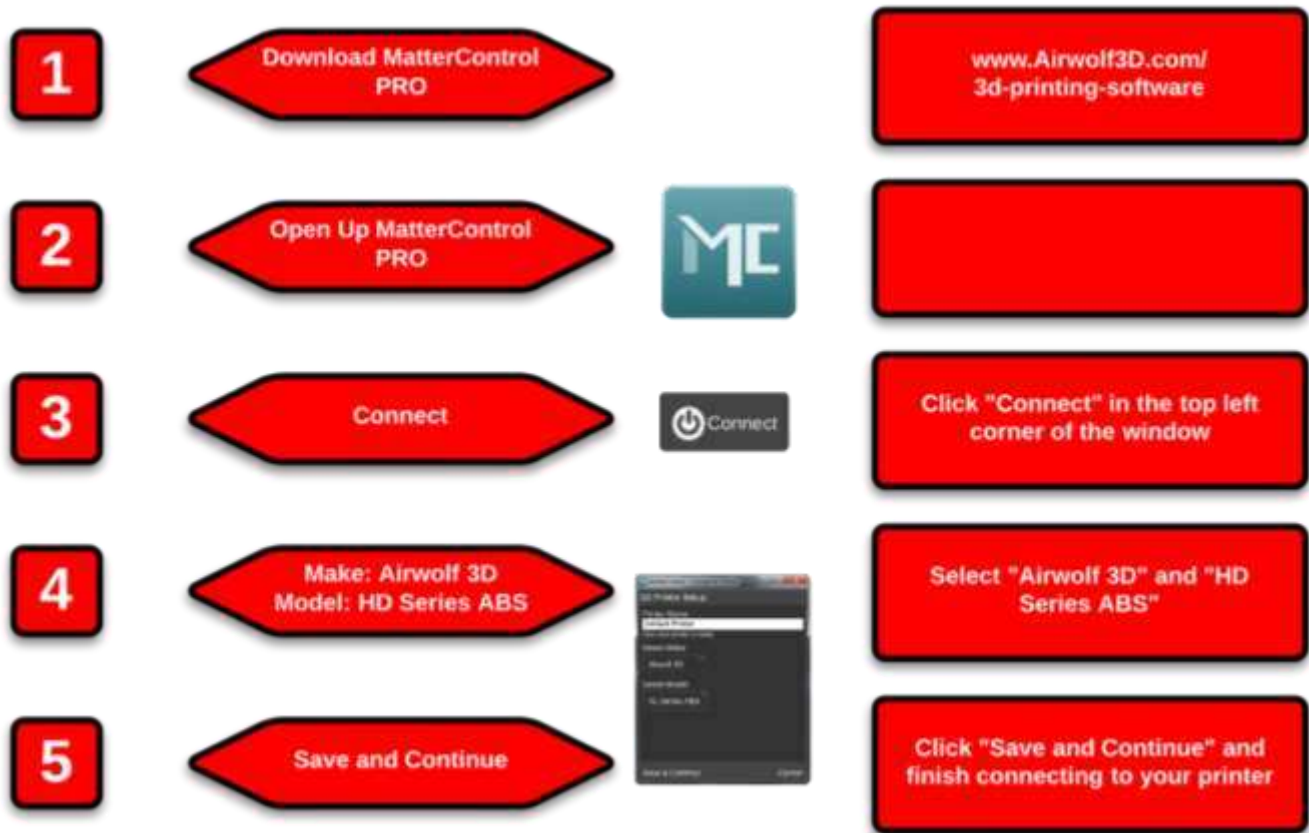
4-5) MatterControl PRO: How To (contd.)

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.

6	Add an STL file		In the lower left corner of the window, click "Add" and select an STL file
7	Viewing the STL file		The STL file can be viewed by clicking the picture of the STL file in your que
8	Advanced Controls	Advanced Controls →	Click "Advanced Controls" on the right side of the window
9	Print Controls	Printer Controls	Click "Print Controls" in the middle of the window
10	Preheat Extruder/Bed		Preheat the Extruder and Bed Temperature (240C and 115C)
11	Slice Settings	Slice Settings	Click "Slice Settings" in the middle of the window
12	"Options" > "Import" > SETTINGS.INI	Options	Click "Options" on the right side of the window. Select the settings from the USB stick.
13	Print Que	← Print Queue	Click "Print Que" on the left side of the window
14	Start		Click "Start" in the top panel to start your print

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.

1	In Each Cup Measure A & B (5:1) [Weight]	Plastic Cup (2); Gloves; Weight Scale; Two Part Epoxy
2	Pour Each Cup into Large Plastic Bag; Seal and Mix Bag	Large Plastic Bag
3	Open Bag and Submerge Printed Part in the Epoxy	Printed Part
4	Release as Much Air as Possible and Reseal	
5	Let Sit for 20-25 Minutes	
6	Take Part Out of Bag Drain Part of Excess Epoxy	
7	Hang Dry Part with Cup Under Part to Catch Excess Epoxy	Metal Hanger Cup
8	For Best Results, Hang in Oven @ (37-49)C or (100-120)F	Oven

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.
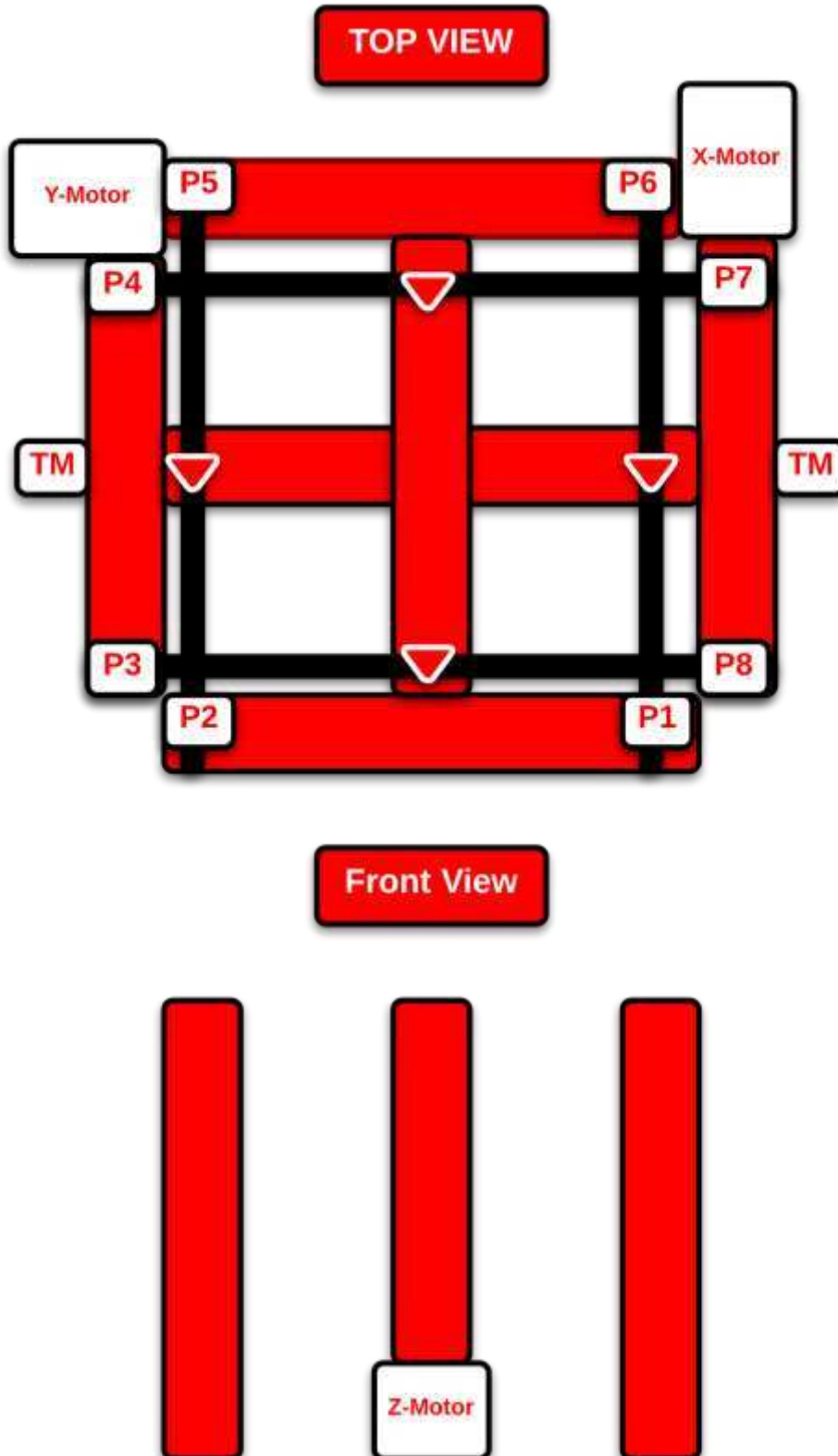
Belt Tensioning

| 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)

Y-Motor

X-Motor

P4

P7
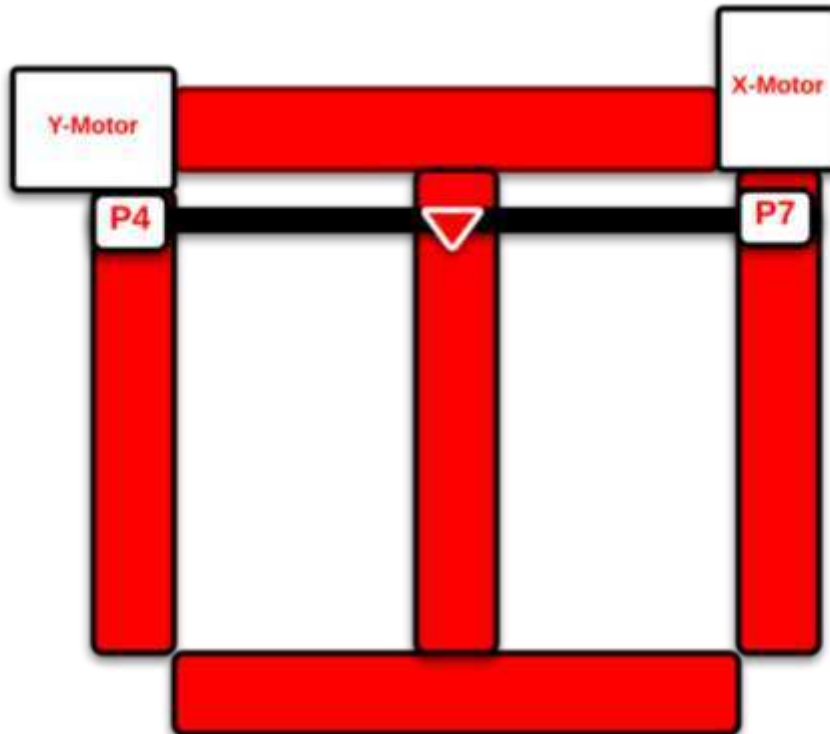
Rod Alignment (X-axis)

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]

Y-Motor

X-Motor

P6

TM

X-axis

TM

The timing marks are white and are on the inside of the inner acrylic panels

P1
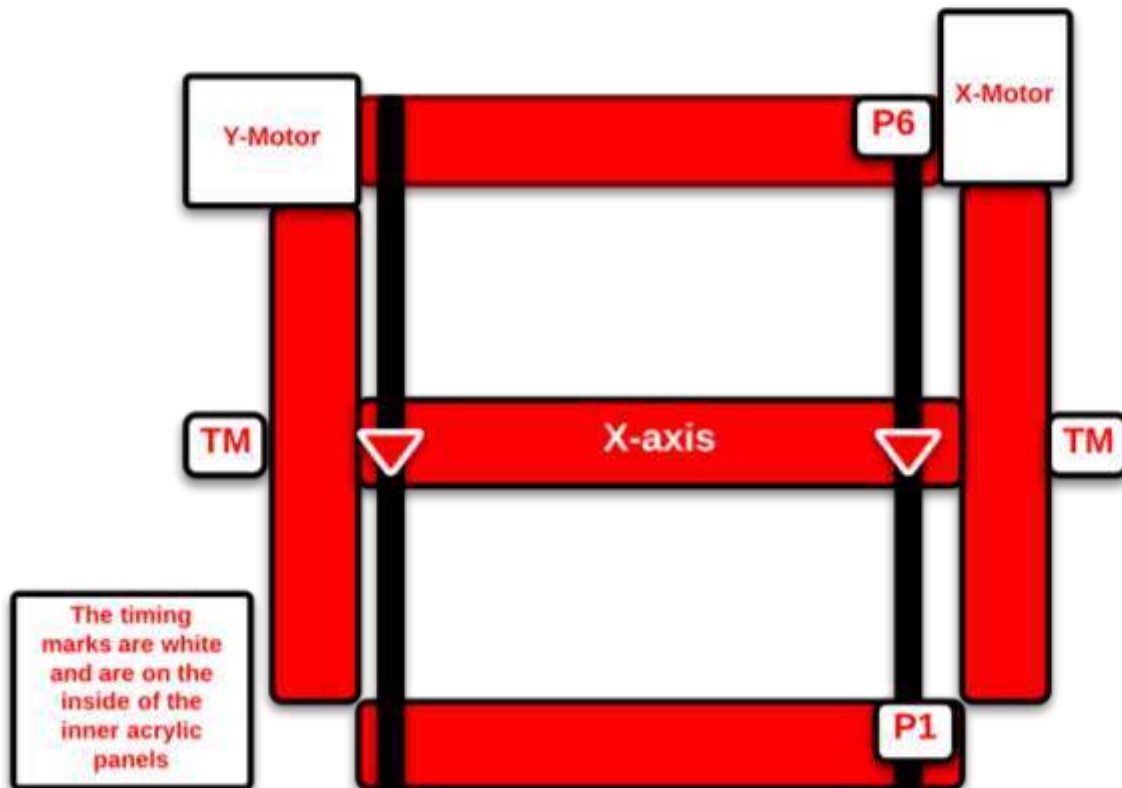
Rod Alignment (Y-axis)

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)

Y-Motor   P6   X-Motor

P4   P7

Y-axis

P1
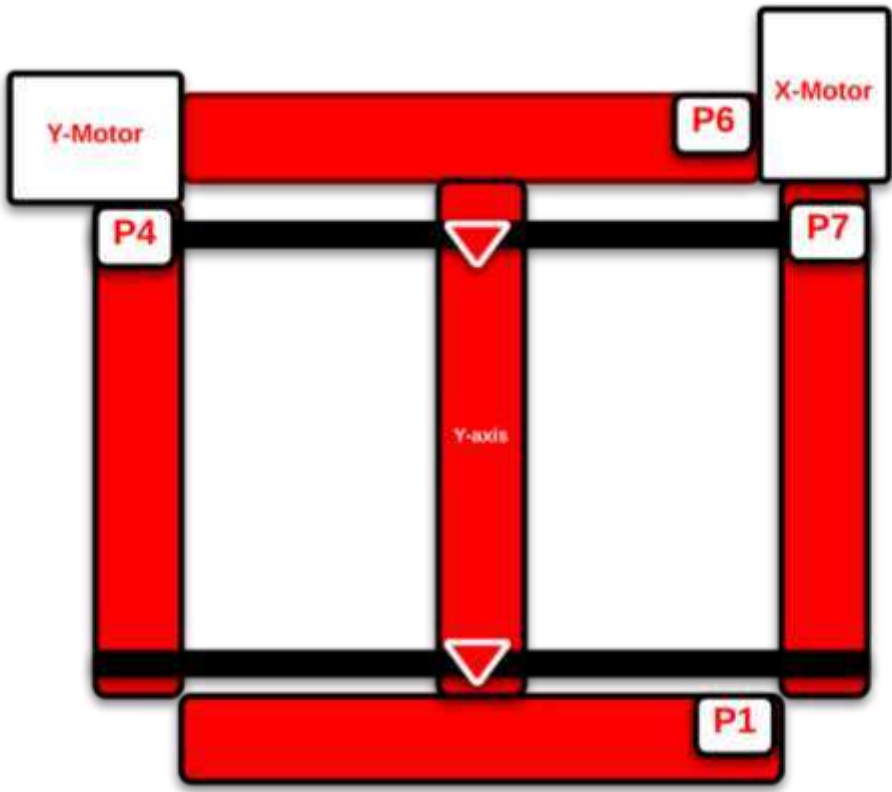
Rod Lubrication

Use Lithium Grease on all the metal rods (labeled in red) on the first maintenance page. The only exception is the threaded Z-rod. Use the "Lock-ease" for the threaded Z-rod.

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
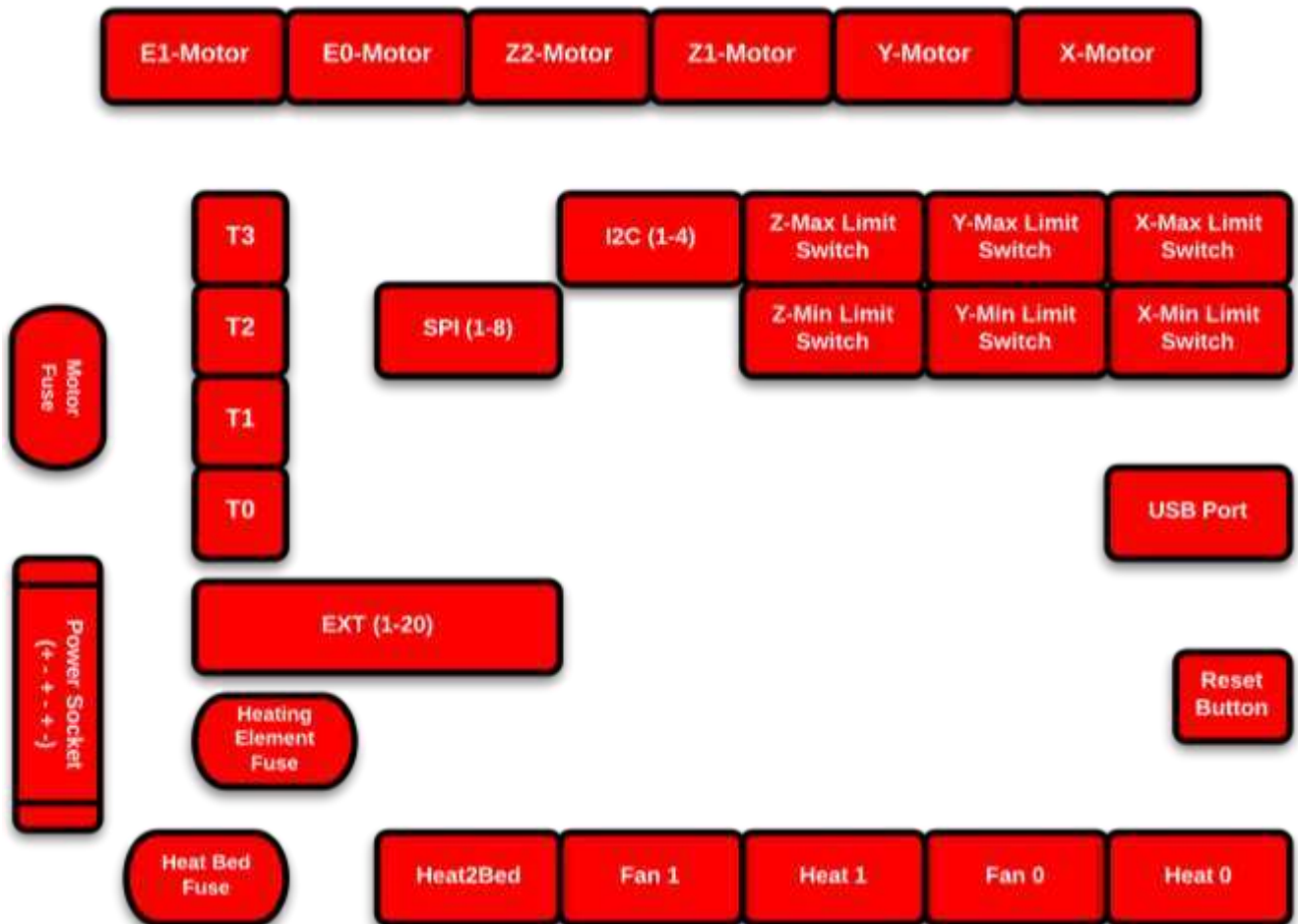**Limit Switches-** All 6 limit switches are used
Heaters- Heat 0 is for the extruder (heating element)
Thermistors- T0 is for reading the extruder temperature
Fans- Fan 0 is for the extruder fan
Fuses- Motor Fuse, Heating Element Fuse (Nozzle), and Heat Bed Fuse

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

E1-Motor	E0-Motor	Z2-Motor	Z1-Motor	Y-Motor	X-Motor

T3		I2C (1-4)	Z-Max Limit Switch	Y-Max Limit Switch	X-Max Limit Switch
T2	SPI (1-8)		Z-Min Limit Switch	Y-Min Limit Switch	X-Min Limit Switch

Motor Fuse

T1

T0

USB Port

Power Socket (+ - + - + -)

EXT (1-20)

Reset Button

Heating Element Fuse

Heat Bed Fuse	Heat2Bed	Fan 1	Heat 1	Fan 0	Heat 0

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 ≥ 240C) (Bed ≥ 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 +- 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 ≥ 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 of 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

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

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 and 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.

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

Notes

TERMS OF USE

FAIR USE NOTICE: This directory may contain copyrighted material. Such material is made available for informational and educational purposes, to advance understanding of 3D printers, 3D printing, and computer-aided modeling for 3D printing. This constitutes a 'fair use' of any such copyrighted material as provided for in Title 17 U.S.C. section 107 of the US Copyright Law. In accordance with Title 17 U.S.C. Section 107, the material on this site is distributed without profit to those who have expressed a prior interest in receiving the included information for research and educational purposes. If you wish to use copyrighted material from this directory for purposes of your own that go beyond 'fair use', you must obtain permission from the copyright owner.

Most of the models contained in this directory are subject to licenses. The "Thing" No. of the model corresponds to the thing as referenced on www.thingiverse.com. Thingiverse.com idenitifies the particular license associated with the specific model. The licenses can be found at http://creativecommons.org/.

THESE MODELS ARE PROVIDED "AS-IS" AND "AS AVAILABLE" AND AIRWOLF 3D (AND OUR SUPPLIERS) EXPRESSLY DISCLAIM ANY WARRANTIES AND CONDITIONS OF ANY KIND, WHETHER EXPRESS OR IMPLIED, INCLUDING THE WARRANTIES OR CONDITIONS OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE, TITLE, QUIET ENJOYMENT, ACCURACY, OR NON-INFRINGEMENT. WE (AND OUR SUPPLIERS) MAKE NO WARRANTY THAT THE MODELS: (A) WILL MEET YOUR REQUIREMENTS; (B) WILL BE AVAILABLE ON AN UNINTERRUPTED, TIMELY, SECURE, OR ERROR-FREE BASIS; OR (C) WILL BE ACCURATE, RELIABLE, FREE OF VIRUSES OR OTHER HARMFUL CODE, COMPLETE, LEGAL, OR SAFE.

TO THE FULLEST EXTENT PERMISSIBLE BY THE APPLICABLE LAW, WE HEREBY DISCLAIM ALL LIABILITY FOR PRODUCT DEFECT OR FAILURE, CLAIMS THAT ARE DUE TO NORMAL WEAR, PRODUCT MISUSE, ABUSE, PRODUCT MODIFICATION, IMPROPER PRODUCT SELECTION, NON-COMPLIANCE WITH ANY CODES, OR MISAPPROPRIATION.

TO THE FULLEST EXTENT PERMISSIBLE BY THE APPLICABLE LAW, WE HEREBY DISCLAIM ANY AND ALL RESPONSIBILITY, RISK, LIABILITY AND DAMAGES ARISING OUT OF DEATH OR PERSONAL INJURY RESULTING FROM ASSEMBLY OR OPERATION OF OUR PRODUCTS.

AIRWOLF 3D ASSUMES NO RESPONSIBILITY, NOR WILL BE LIABLE, FOR ANY DAMAGES TO, OR ANY VIRUSES OR MALWARE THAT MAY INFECT YOUR COMPUTER, TELECOMMUNICATION EQUIPMENT, OR OTHER PROPERTY CAUSED BY OR ARISING FROM YOUR ACCESS TO, USE OF, OR BROWSING THIS WEBSITE, OR YOUR DOWNLOADING OF ANY INFORMATION OR MATERIALS FROM THIS WEBSITE.

Limitation on Liability. IN NO EVENT SHALL AIRWOLF 3D (AND OUR SUPPLIERS) BE LIABLE TO YOU OR ANY THIRD PARTY FOR ANY LOST PROFIT OR ANY INDIRECT, CONSEQUENTIAL, EXEMPLARY, INCIDENTAL, SPECIAL OR PUNITIVE DAMAGES ARISING FROM OR RELATING TO THESE TERMS OF USE, EVEN IF WE HAVE BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES. ACCESS TO, AND USE OF, THE MODELS ARE AT YOUR OWN DISCRETION AND RISK, AND YOU WILL BE SOLELY RESPONSIBLE FOR ANY DAMAGE TO YOUR COMPUTER SYSTEM OR LOSS OF DATA RESULTING THEREFROM.

NOTWITHSTANDING ANYTHING TO THE CONTRARY CONTAINED HEREIN, OUR LIABILITY TO YOU FOR ANY DAMAGES ARISING FROM OR RELATED TO THESE TERMS OF USE (FOR ANY CAUSE WHATSOEVER AND REGARDLESS OF THE FORM OF THE ACTION), WILL AT ALL TIMES BE LIMITED TO THE GREATER OF (A) FIFTY US DOLLARS ($50); OR (B) AMOUNTS YOU'VE PAID COMPANY IN THE PRIOR 12 MONTHS (IF ANY). THE EXISTENCE OF MORE THAN ONE CLAIM WILL NOT ENLARGE THIS LIMIT. YOU AGREE THAT OUR SUPPLIERS WILL HAVE NO LIABILITY OF ANY KIND ARISING FROM OR RELATING TO THESE TERMS OF USE.

SOME JURISDICTIONS DO NOT ALLOW THE LIMITATION OR EXCLUSION OF LIABILITY FOR INCIDENTAL OF CONSEQUENTIAL DAMAGES, SO THE ABOVE LIMITATION OR EXCLUSION MAY NOT APPLY TO YOU.