

*Airwolf 3D*TM

**3D PRINTER
USER MANUAL**

model AW3D

(model AW3D v.4, v.5, v.5.5)

www.airwolf3d.com

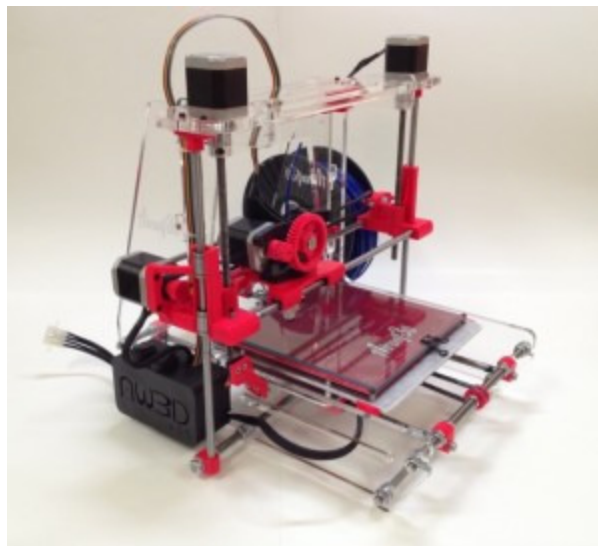
**INSTRUCTIONS FOR OPERATING YOUR 3D PRINTER -
UPDATED 02/12/2013**

THIS USER MANUAL IS MANDATORY READING BEFORE OPERATING YOUR
AIRWOLF 3D PRINTER



THIS 3D PRINTER IS NOT A TOY

KEEP OUT OF THE REACH OF CHILDREN AT ALL TIMES



**KEEP HANDS AWAY FROM THE HOT END, HEATED BED, BELTS, GEARS,
AND POWER SUPPLY AT ALL TIMES UNLESS THE POWER SUPPLY IS
UNPLUGGED AND THE MACHINE HAS SUFFICIENTLY COOLED DOWN**

IF ANYTHING MALFUNCTIONS, UNPLUG IMMEDIATELY!

THEN CALL OR EMAIL US. WE WILL HELP!

1. SOFTWARE FOR YOUR 3D PRINTER

- Familiarize yourself with all the appropriate licenses for using the STLs, software, and firmware included with the printer. Our website has a list of sources for this information. Much of the material is "open source." While this generally allows you to access the underlying file or programming, it does not necessarily grant you the right to reproduce or sell the item (read the license carefully). In most cases you are also responsible for crediting the author for the work when you reproduce and/or distribute the work. For the AW3D v.4, v.5, and v.5.5, designs by Prusa and Mecano were the primary inspirations for the machines.
- Install the **software included with your Airwolf AW3D v.4, v.5, or v.5.5 onto** your computer as follows (for MAC, the software will need to be downloaded from the respective websites).
 - Install **Slic3r** (v.7.2b) on your desktop or in your C directory. It is a simple install because it does not require any dependencies. For printing out the included STL files that make up AW3D v.4, v.5, and v.5.5, we recommend 7.2b because the parts were designed to be sliced on this version. The later versions however (such as 9.8) are very sophisticated and allow you to alter a number of parameters which can greatly improve the quality of your prints (MAC users refer to www.slic3r.org).



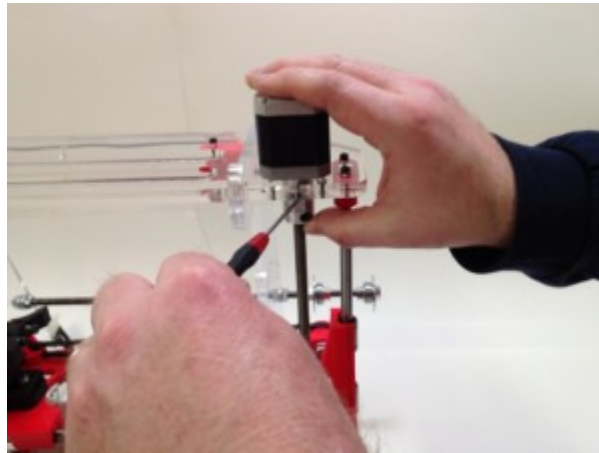
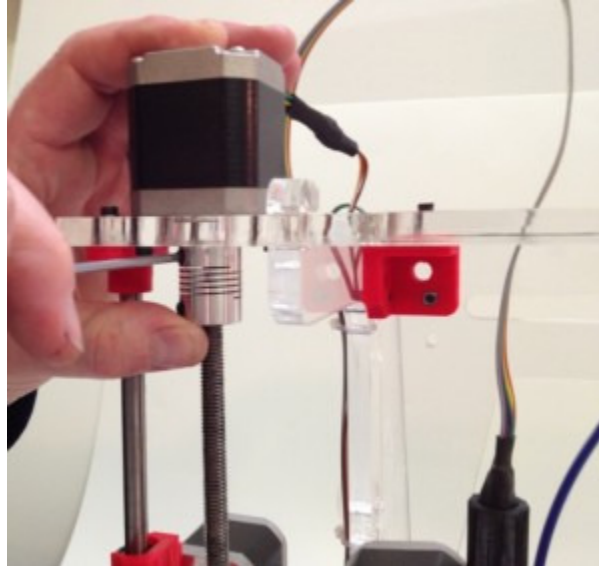
- Install Python 27 directly into your C: directory (MAC users refer to <http://www.python.org/getit/>).
- Install Pronterface (can also be downloaded as ZIP from <https://github.com/kliment/Printrun>) similarly and put a shortcut on your desktop. On older versions of windows, you may need to download Python 2.7.3 from <http://www.python.org/getit/>. If you go to open up Pronterface and the user interface comes up, everything is working correctly (MAC users refer to <http://koti.kapsi.fi/~kliment/printrun/> and <https://github.com/kliment/Printrun>).



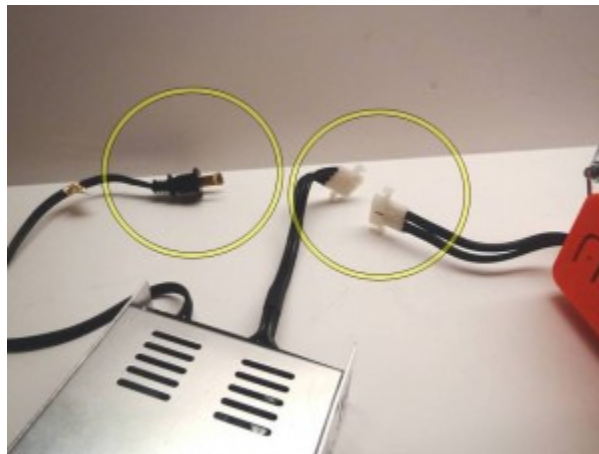
- Once you have become a pro with Pronterface, download Repetier Host from <http://www.repetier.com/>. This software will allow you to control the speed of the print and provide you with a visual before printing (both very, very important for long 10-15 hour prints). You will use this program essentially to replace Pronterface.
- These are the only programs you will need to run the printer. Later, you may wish to install Arduino 23 to modify the firmware, Google Sketchup (or an alternative CAD program) to create your 3 dimensional objects, and Meshlab (or a similar program such as Netfabb) to make sure your objects are watertight.
- To get the printer running, you may also need to download the FTDI driver for the Gen6 circuit board. With most versions of windows, it will load the driver automatically once the printer is turned on, connected to your computer, and your computer is **connected to the Internet**. If this does not happen, you can download the appropriate driver from here: <http://www.ftdichip.com/Drivers/VCP.htm>. For Windows 7 64 bit, download the "[setup executable](#)" under the Comments column.

2. INSPECTION/ASSEMBLY OF YOUR 3D PRINTER

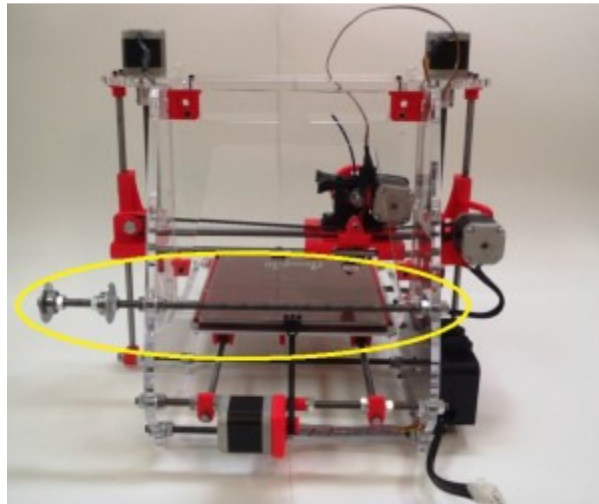
- If your 3D printer was shipped, inspect the machine carefully to remove all packing material. Also remove all fasteners such as zip ties that were used to steady the X, Y, and Z axis' during shipping.
- **Connect** the Z axis flexible couplers. They were disconnected to protect the integrity of the couplers during shipping. Line the top of each coupler up with the acrylic frame. Also, make sure there is about 1/8" separating the threaded rods from the motor shafts (to provide room for flex). Tighten both bolts on each coupler using a 2.5 mm allen wrench. Make sure they are **very tight** to avoid any slippage.



- With software installed, make sure all electrical connections to the printer are disconnected (this includes the USB cord from the computer).



- Inspect the 3D printer to make sure there are no loose wires, loose bolts or obstructions that would impeded the printer's movement in any of the three directions (X (left/right),Y (front/back),Z (up/down)).
- Make sure that nothing has broken or fallen off from shipping. Please note that the filament rod was loosely attached to the printer frame for shipping convenience and will need to be repositioned to hold the filament roll.



- Place the borosilicate glass plate onto the heated bed surface, making sure that the orientation of the glass is correct such that it lies between the heated bed bolts on the left and right sides.



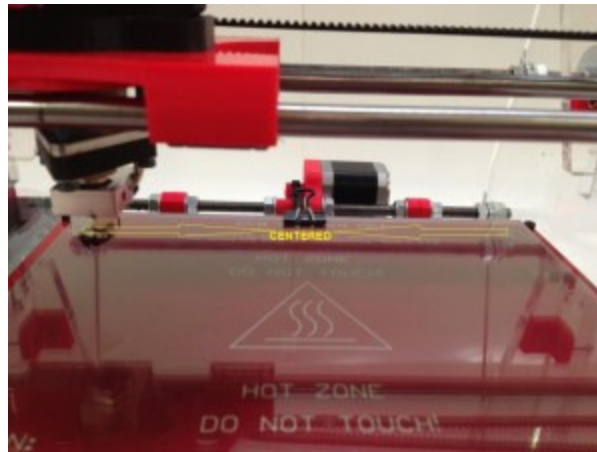
- Important: this is a very special piece of glass made to withstand the hot temperatures generated by the heated bed. If broken, do not attempt to replace this plate with standard glass.
- Install the alligator clips to secure the glass plate onto the heated bed.



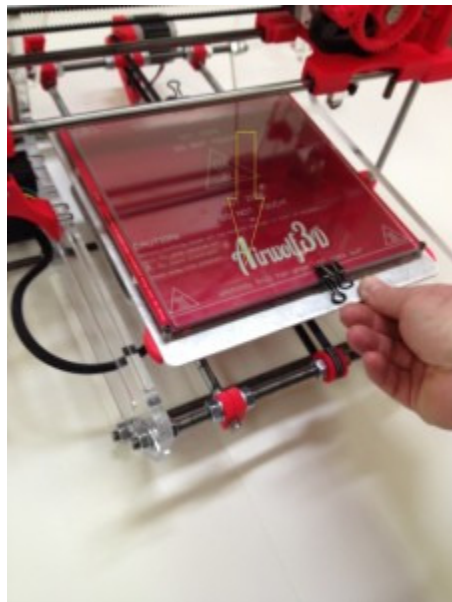
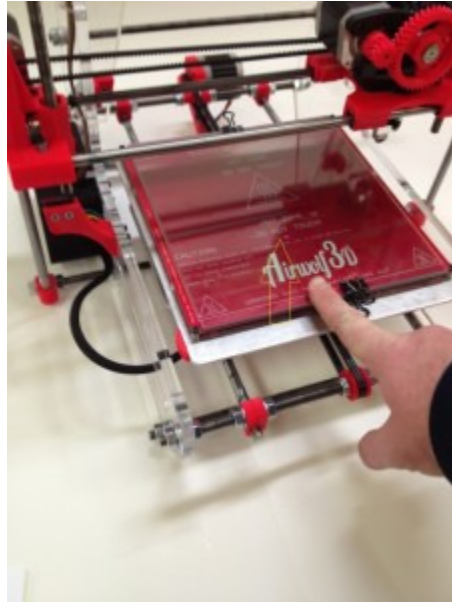
- Note that the glass plate is covered with PET tape that must be facing up. **DO NOT RUN THE HOT END INTO THE GLASS PLATE** because replacing this tape (required for a good “stick”) is at first difficult if you are not familiar with laying films. We carry the very best replacement tape, so feel free to contact us when it comes time to replace it. Further, additional coated glass plates are available for extra cost if you want to continuously “cycle” through builds. In the alternative, you can always use blue painter's tape (preferable for PLA), however for ABS, the stick is not as strong as with the PET tape.
- **IMPORTANT** Place the front clip approximately where the heated bed lettering states “LED.”



- Look under the bed to make sure that the heater wires are **NOT** contacting the front clip.
- Install the rear clip onto approximately the center of the rear portion of the glass (two clips can also be used to gain a stronger grip). Important: the orientation of the clips will affect the process of leveling the bed. The bed should already be level if the clips are placed as shown in the photographs.



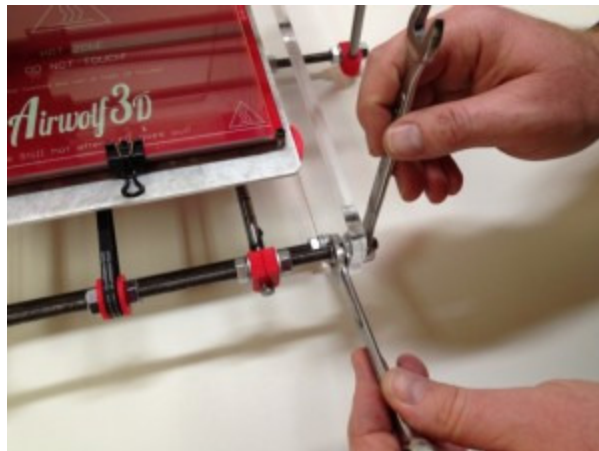
- Move the bed front and back, making sure that the heater wiring (extending from the left side of the frame and into the Gen6 box) does not bind and test whether the Y micro switch activates when the bed is in the rearward position (you will hear a “click”).



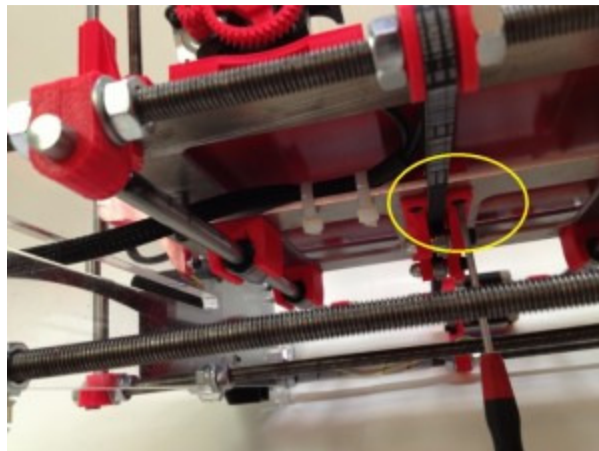
- Move the X carriage left and right to ensure no binding and to confirm that the X micro switch activates similar to above.



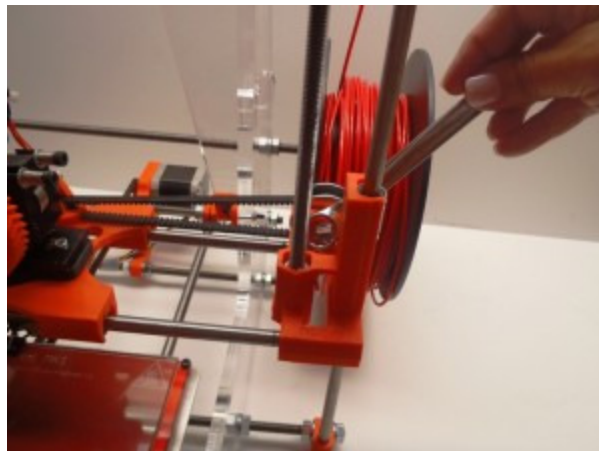
- Acquaint yourself with the machine and observe all the moving components. Make sure the belts are snug.
- To tighten the Y belt, loosen the nuts on the front of the machine holding the M8 threaded rod to the frame and smooth rods. Pull back on the rod to remove slack from the belt and then tighten the nuts accordingly.



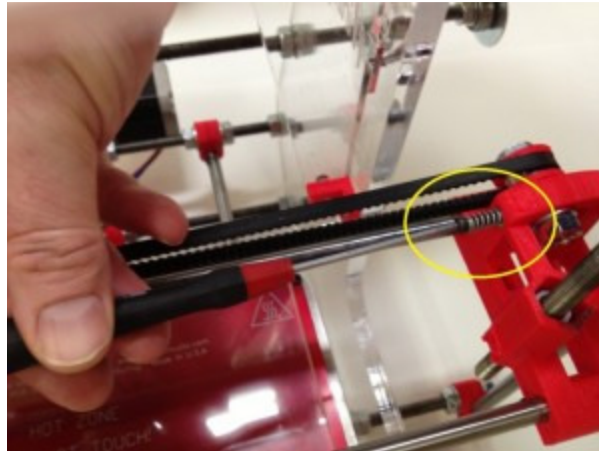
- **Do not over tighten. V.5.5** - you may also tighten the Y belt underneath by loosening the bolts running through the Y idler and then tightening the tensioning bolt and then re-tensioning the Y idler bolts.



- To tighten the X belt, loosen the 13mm nut on the right X Bracket and pull back on the bolt and nut to remove slack from the belt. Do NOT over tighten.



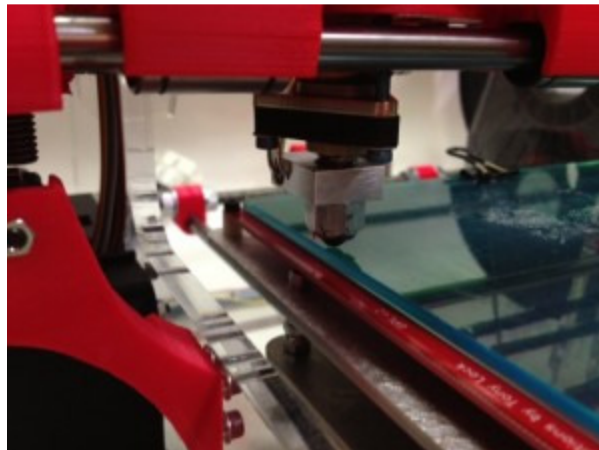
- v.5.5 - tighten the tensioning bolt, but do NOT over tension and then tighten the 13mm bolt and nut accordingly.



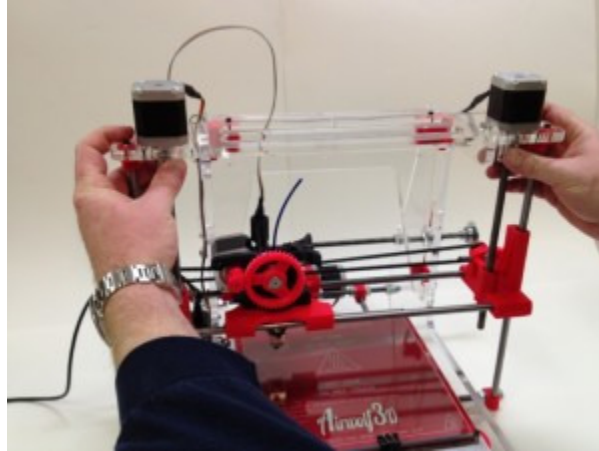
- Pay attention to the smooth rods and threaded rods for the Z axis. Periodically these components will need to be greased to ensure optimal performance. For the smooth rods, a thick automotive-style grease is recommended. For the threaded rods, graphite lubrication is recommended to avoid dirt contamination.

3. LEVEL THE 3D PRINTER

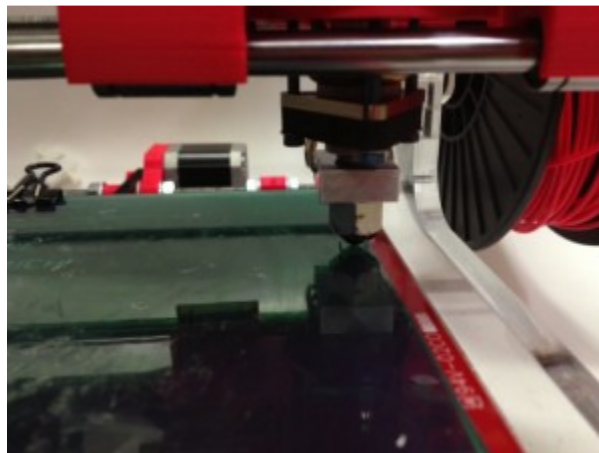
- **prior to turning the 3D printer on** as the X, Y, and Z axis will freely move without power supplied to the steppers.
- First, orient the hot end nozzle approximately above the left rear Y assembly bolt (reason will become clear later).
- Now, uniformly lower the threaded rods in the X Brackets such that there is approximately a 1/16" - 1/8" gap between the nozzle tip and the glass plate (by manually turning the Z couplers).



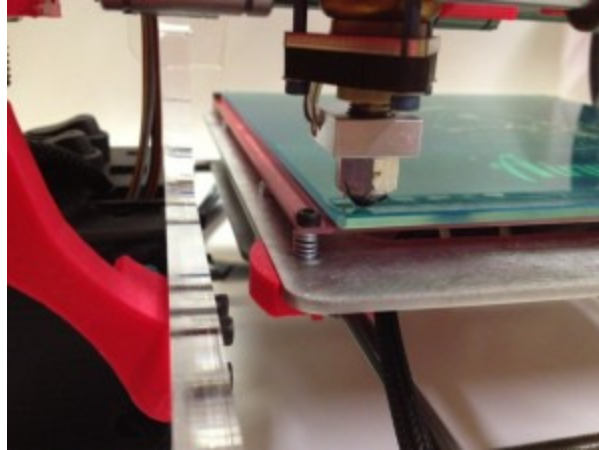
- Next, slowly move the X carriage from left to right and vice versa. Note that the gap between the nozzle tip and glass plate varies from left to right.
- Level the X assembly by holding the threaded rod from one X Bracket while turning the other (you will need to hold one while turning the other as they will move in unison otherwise):



- Move the X Carriage left to right and vice versa to confirm that the X Assembly is leveled.



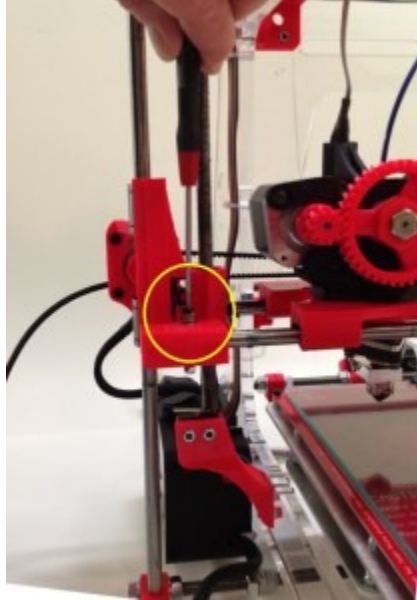
- Now move the bed (Y assembly) back and forth to check if the bed is leveled front to back (Y direction).



- Adjusting the bed is relatively straightforward. After moving the bed back and forth, simply turn the 2.5mm allen bolts on the front corners of the bed to either raise or lower the corners such that the nozzle is spaced about 1/16"-1/8" off the bed.



- Now the printer should be leveled. Fine adjustments can be made after your first test print. The bed will tend to stay almost perfectly level, so any adjustment hereafter should be minimal.
- Before moving further however, adjust the Z axis microswitch bolts such that it is actuated with the nozzle just over the glass plate. This is **IMPORTANT!!!** It will prevent the nozzle from crashing into (and **BREAKING**) the glass when you go to do your first print.



4. CONNECT THE 3D PRINTER

- **IMPORTANT** - Confirm that the power selector on your 240W (360W for v.5.5) power supply conforms to your supplied voltage.
- If your supplied voltage is not 110 or 115, you will probably also have to buy an adapter for the plug or fashion one yourself. Do not attempt to do this unless you are qualified to do so.
- Make sure the power supply is **disconnected from** the wall. Plug the four (4) pin power supply connector from the power supply into the corresponding connector on the 3D printer.



- Attempt to place the power supply behind the 3D printer such that the connectors are oriented on the left hand side of the 3D printer. Make sure that

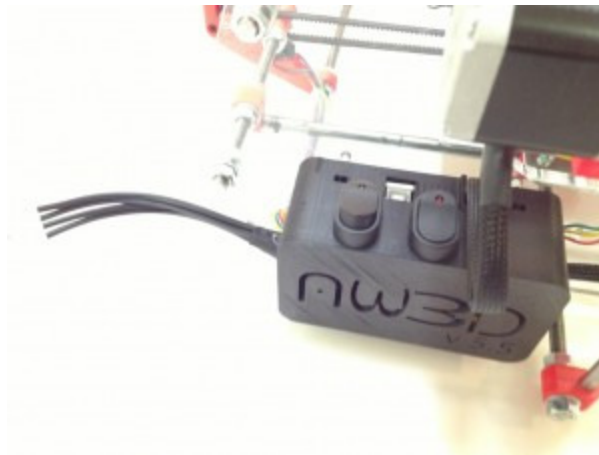
the power supply is lying flat and that there are no obstructions to the circular fan vent or vented opening, which must be facing up.



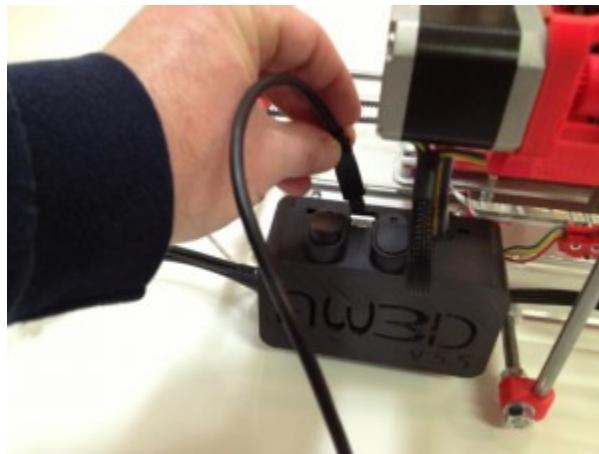
- Plug in the power supply to your wall outlet. Make sure that the power supply cord is out of the way of the moving parts of the 3D printer.
- Check the operation of the power switch (blue LED) and heater switch (red LED).



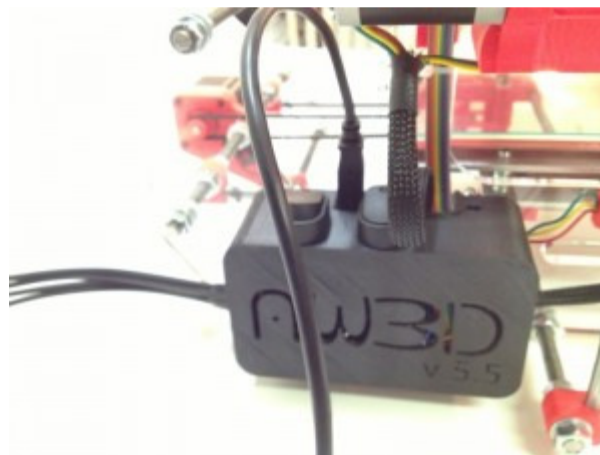
- Turn the power switch on and turn the heater switch off (you will use the heat later). You will know which is on by the LED indicator.



- Connect the small end of the USB cord to the top of the Gen6 box through the square-shaped hole on the top of the box,



between the two power switches and then connect the USB cord to your computer.



- Make sure that your computer is connected to the Internet. It will need the Internet to locate and install the appropriate driver for the Gen6 board. If this process fails, you will need to locate the driver software and perform a manual install as indicated above. Thankfully, this is a rare occurrence.
- Once the Gen6 driver is installed, open your printer interface program (Pronterface preferred to begin with).
- Click on "Connect" and wait for confirmation from Pronterface. Make sure the baud rate is set to **250000**.
- Click on "Monitor Printer" so that you will receive continuous updates on the status of the 3D printer.

5. GETTING FAMILIAR WITH THE 3D PRINTER

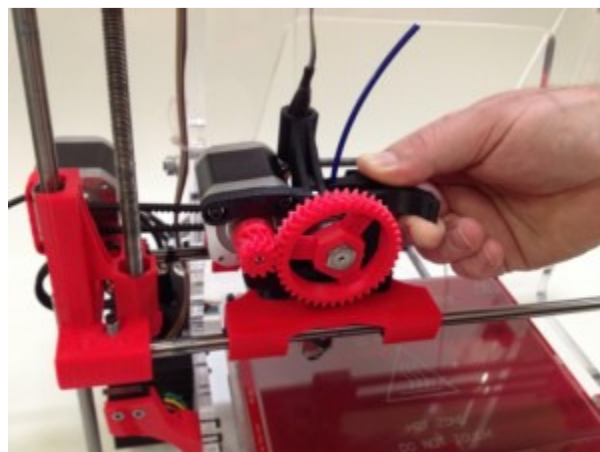
- **IMPORTANT!!! - do not attempt to heat up the nozzle or heated bed yet. Also, make sure the nozzle is still spaced apart from the glass plate to avoid breakage.**
- Toggle the X axis **slightly** by clicking on the inside of the hemisphere in the "+X" direction. Check to make sure that the X carriage moves as directed. If you hear loud gnashing from the motor (skipping steps), this is probably because the X axis is at it's maximum travel in that direction. Do not continue to subject the printer to this. Try small steps in the reverse direction to confirm operation.
- Toggle the Y axis **slightly** by clicking on the inside of the hemisphere in the "+Y" direction. Check to make sure that the print bed moves as directed. If you hear loud gnashing from the motor (skipping steps), this is probably because the Y axis is at it's maximum travel in that direction. Do not continue to subject the printer to this. Try small steps in the reverse direction to confirm operation.
- **BE VERY CAREFUL ABOUT THE Z AXIS!!!.** Toggle the Z axis **slightly** by clicking on the inside of the hemisphere in the "-Z" direction. **IF YOU MOVE THE Z AXIS TO FAR IN THE "+Z" direction, you risk cracking the glass, damaging the hot end, and throwing off the calibration of the 3D printer. PROCEED WITH CAUTION.** Check to make sure that the print head moves as directed. If you see the hot end pressing the glass down,

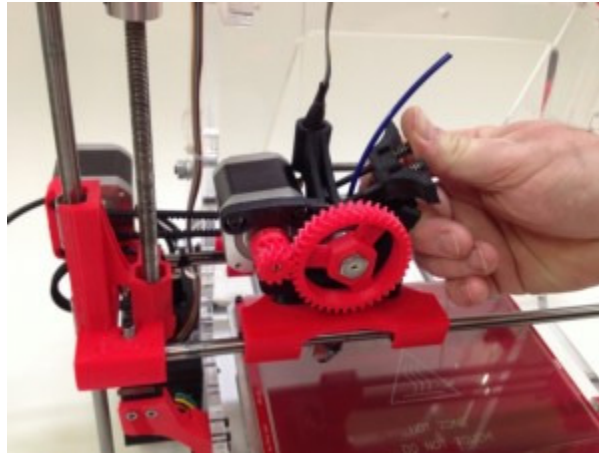
stop **IMMEDIATELY** and unplug the printer and regroup. Once regrouped, try small steps in the reverse direction to confirm operation.

- **SLOWLY** move the Z axis in the “-Z” direction such that the print head moves closer to the glass. This is a critical adjustment that you will become very familiar with. Initially adjust the Height Adjusting Bolt (on the left side X Bracket) such that the micro switch activates **BEFORE** the print head comes into contact with the glass. Adjust the Bolt such that there is a very small distance between the tip of the hot end and the glass. Don't worry, you will eventually become very familiar with this adjustment and it should take seconds once you have become accustomed to the 3D printer.
- Confirm the print bed is level.
 - Through Pronterface, **SLOWLY** move the hot end on the X and Y axis. Check to make sure the hot end is approximately the same distance from the glass bed in all locations.

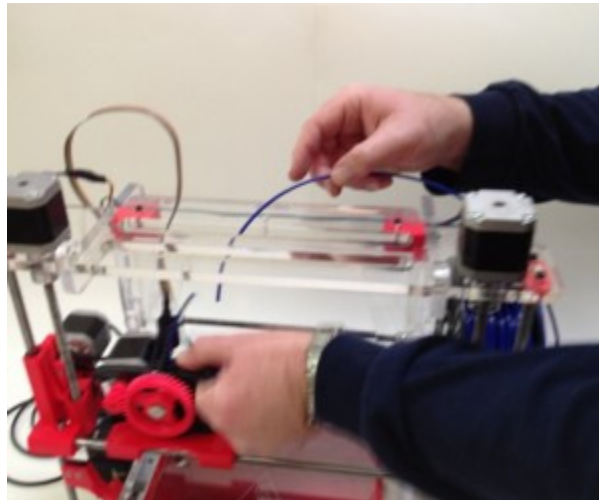
6. GETTING READY TO 3D PRINT

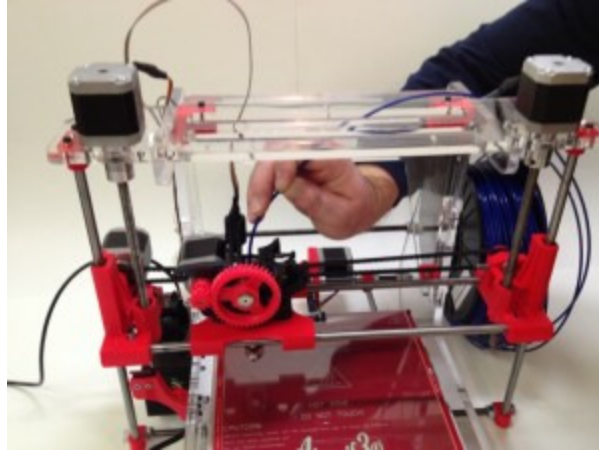
- Through Pronterface, set the hot end temperature to 240 degrees Celsius (240 C). It will take a few minutes to heat up. This is very, very hot so be **CAREFUL. NEVER TOUCH A HEATED HOT END.**
- Flip the Airwolf Quick Release Latch up to expose the filament in the extruder (mid-v.5-v.5.5).



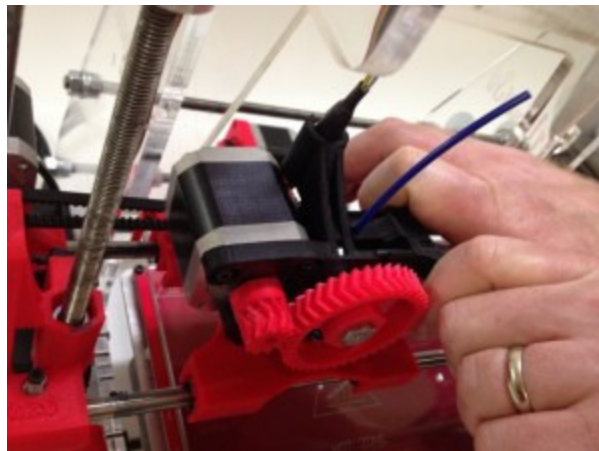
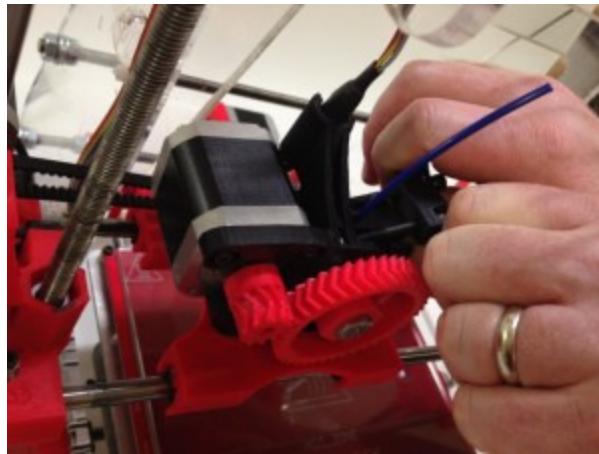


- Loosen the two bolts on the extruder the left the two bolts and flip the extruder tensioner back (early-mid v.5).
- With the hot end heated, you should be able to easily remove the remaining filament (we run the machines for at least 10 hours prior to delivery to confirm that everything is working correctly):
- Insert the supplied 3mm ABS into the hole in the extruder. You may need to cut the tip of the ABS so that it slides all the way down into the extruder.

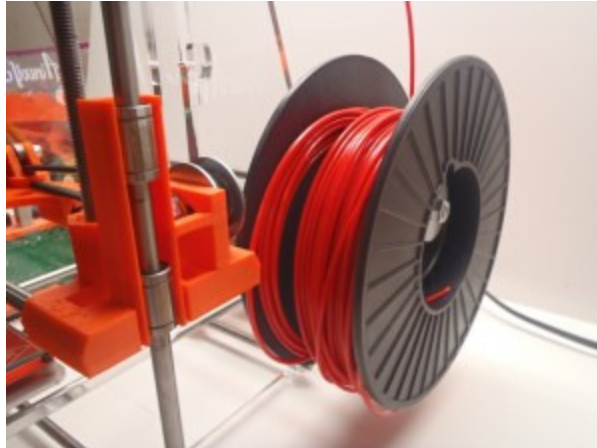




- (early-mid v.5 only) - tighten the two bolts on the extruder to compress the respective springs. This is an adjustment you will become familiar with and it will vary between filaments. There is a compromise between having the filament too tightly compressed against the hobbed bolt ("flattening the filament") and having it too loose such that the hobbed bolt does not grip the filament enough for quick retractions.



- The ABS should have a smooth path from the spool on the right side of the printer through the slotted opening in the top of the printer and into the extruder. Make sure there are no impediments to the rotation of the spool.



- As the ABS is drawn into the extruder, the spool will rotate accordingly. You must however continuously keep an eye on the spool to make sure there are no snags which would cause the filament to hang up.
- **VERY IMPORTANT.** On the spool, make sure the “flush” side is facing towards the printer. If the spool is reversed it may hang on one of the protruding frame bolts and cause the spool to stop rotating.

7. HEATING UP THE HOT END AND HOT BED

- Once the ABS is loaded and the orientation of the spool is confirmed, set the temperature on Pronterface to 240 degrees Celsius. **MAKE SURE** the hot end is not touching the bed (toggle it up the Z axis) so that it does not burn the tape or create too much internal pressure. This is the baseline temperature for ABS. Different ABS may have different optimum printing temperatures. We recommend buying replacement spools in similar configurations from 3Dprinterstuff.net until we have an established supply chain from our company.
- **DO NOT TRY TO “EXTRUDE” ANY MATERIAL BEFORE THE HOT END IS UP TO TEMPERATURE.** This will create binding in the extruder and possibly damage internal components. We have set the firmware to limit extrusion until at least 175 degrees Celsius (for PLA) and to limit the maximum

temperature to 275 degrees Celsius (although the maximum recommended temperature for ABS is 230-240 degrees Celsius).

- You will notice that after a minute or so the hot end will start “oozing” ABS. This is perfectly normal as the plastic is being heated and will naturally proceed downward through the nozzle with gravity.
- **DO NOT USE PLA** initially. PLA is not as forgiving and can get stuck in the extruder if not properly controlled. Disassembling the extruder at this point in your 3D printing learning curve is not something you want to get into.
- Also, PLA has a **MUCH LOWER** printing temperature (185-200 C) that must be minded carefully. Further, PLA varies from supplier to supplier. If you decide to use PLA at some point, you will need a fan to cool the "bell" on the hot end, at least if you are planning on using the 4043D variety (as supplied by ultimachine and protoparadigm). Our friends at Matter Hackers like to cover the nozzle and heating block with kapton tape to avoid overcooling the hot end at the same time. The fan does not have to be on the X Carriage, it can simply be next to the printer and pointed at the hot end. While PLA is necessary for making tall prints without cracks, start off with ABS until you get familiar with the printer.
- Once all the above is confirmed, turn on the heat by flipping the red LED rocker switch.
- Heat up the bed for at least five (5) minutes prior to attempting to print as the parts will not “stick” otherwise. The board draws a continuous current and will heat to approximately 100 degrees Celsius. If the board ever fails to heat (or you notice any other electrical issue), unplug the printer and contact us immediately.
- **IMPORTANT TIP:** Coat the glass plate with a combination of acetone and ABS. We normally mix a small amount of “natural” ABS with acetone in a jar. The natural is used so it does not stain the bottom of the printed part. You can apply the mixture to the glass plate with a small paint brush generally conforming to the area you will be printing on, **TAKING CARE NOT TO DRIP THE MIXTURE** on the clear acrylic panels, as it will stain them. Also, always keep the mixture **AWAY FROM CHILDREN**.

- Once the hot end is up to temperature (as indicated by the Pronterface interface), the bed is leveled, the bed is up to operating temperature, the ABS is properly routed, you have sufficiently familiarized yourself with the printer, and you are **mentally prepared**, you may move to the next step of actually printing something.

8. PRINTING IN 3D

- The normal work flow is to create an object in Sketchup or a similar 3D modeling software such as Solidworks, check that it is “watertight” in a program like Meshlab or Netfabb, and then “slice” that object in a program like “Slic3r” to create "gcode" specially created for your particular printer (do not use gcode from others, as it will NOT work correctly on your printer). While the parts usually do not have to be watertight to print, watertight parts ensure that what you design is what ends up being printed.
- Here, we have done the work for you, at least for your initial prints. You will do the following:
 - Follow the above instructions to get the printer ready to print. Your heated bed should be hot (100 C), the alligator clips should be securely in place, and your hot end should be at about 240 C (and not sitting on the glass bed).
 - Study the Pronterface interface once again. Make sure you know where the “pause” and “disconnect” buttons are located.
 - Click the “load” button.
 - Load “Circle Calibration.gcode” from the supplied flash drive. **MAKE SURE TO LOAD ".GCODE", NOT ".STL."**
 - You will notice that the base of a cylinder will appear on the Pronterface GUI.
 - Place your hand close to the **BLUE POWER BUTTON** on the machine in case you have to immediately shut off the printer (shutting off the printer will not harm it, on the contrary it will likely save it from unnecessary damage). **KEEP EVERY OTHER BODY PART AWAY FROM THE MACHINE.**

9. WITH EVERYTHING DOUBLE CHECKED, PRESS “PRINT”

- The printer will automatically “home” itself through the three micro switches such that the hot end will end up in front of the print bed and slightly lower than the print surface.
- There will likely be a slight pause as the gcode executes and confirms that the hot end is up to temperature.
- The hot end will then move out over the print bed and start to “extrude.”
- As material has probably been flowing out of the hot end as it comes up to temperature and as it remains still until printing, material will not immediately start printing. This is why we have set up the gcode (through Slic3r) to make two “loops” around the part before it starts to print the part. This effectively “primes” the hot end.
- **HIT PAUSE OR DEPOWER THE PRINTER IF ANYTHING GOES WRONG**
 - By hitting pause, the printer will keep the hotend at 240 degrees Celsius. **BE MINDFUL OF THE EXTREMELY HOT TEMPERATURE OF THE HOT END.**
 - **ALWAYS** raise the hot end by toggling the Z axis up when restarting prints. This prevents it from hitting alligator clips or getting caught up on printed parts.
- Most likely you will be getting “strings” of plastic not sticking to the bed or the hotend will be rubbing into the bed and not giving you anything. Follow the instructions above and reset the Z height by twisting the Height Adjusting Bolt until parts start to print correctly. After making these adjustments and removing the gunk from the bed, you can hit “Restart” on Pronterface to give it another (and another) try until you get it right.
- The first layer is the most critical-make sure that everything is firmly stuck to the bed. You want a nice strip of slightly compressed plastic outlining the part on this first layer.
- If the extruder appears to stop extruding, turn off the printer and inspect the “hobbed bolt” in the extruder (you will need to loosen and pivot up the extruder

bolts, as described above). What may have happened is that the hobbled bolt “stripped” the filament. It therefore is necessary to re-trim the ABS filament and to clean the shavings (while the machine is depowered) from the hobbled bolt. Do not however try to remove the filament while the hot end is cold, the filament will stick and you could break an internal component. Therefore its best to have the hot end up to at least 200 C before attempting to remove the filament.

10. WHILE THE MACHINE IS 3D PRINTING

- Make sure that everything is operating smoothly-that the filament is fed correctly, no components are “binding”, and that everything generally appears to be in order. **DO NOT** leave the printer unattended.
- “Circle Calibration” should be a brief print. It will not only show you how the printer works, it will tell you if you have to level the bed or adjust the Z rods (as described above).
- You may also want to print out the supplied parts in case you should later break a plastic component of the printer. **For your first "real" print we recommend printing out the supplied "Gearset" gcode.** This file illustrates some of the many capabilities of the printer and is a relatively slow print which will enable you to observe the printer in action.
- The process to print out all of the parts will take approximately one day and consume less than 1 lb of plastic if done correctly. More importantly, printing out known working files will familiarize you with the printer and the way it works when a properly constructed STL file is properly executed in gcode.
- **For your first "slice" we recommend loading our "Very Precise .2mm .5mm nozzle" Slic3r configuration in Slic3r 7.2b and using this configuration to create the gcode for your part (such as a part you may have downloaded from Thingiverse).**
- **IF ANYTHING APPEARS TO GO WRONG WITH THE PRINTING PROCESS, IMMEDIATELY DE-POWER THE PRINTER.**

11. AFTER THE PRINT IS COMPLETED

- The X axis (only) will home itself and the part will be completed (Repetier Host functions slightly differently).

- The hot end and the stepper motors will power down.
- However, the heated bed will still be at 100 degrees Celsius (very hot).
- Turn off the heated bed power (red LED) and wait for the glass to cool. You may also wish to power down the printer unless you would like to print immediately again.
- Remove the alligator clips and remove the glass from the printer with the printed part fixed to the glass.
- Removing the part from the glass may be difficult (this is a problem many wish they had as most ABS prints end up with “curl up” where corners of the prints lift as the ABS cools during the print).
- One solution is to run the glass under cold water. This may weaken the bond between the PET tape and the part. Another solution may be to slightly “knock” the part from the glass to remove it.
- Before printing again, clean the “loop” from the glass and make sure to apply the acetone/ABS mixture to ensure the next part will stick to the glass.

12. PARTING WORDS

- We have taken great care to ensure that you are provided with absolutely the best RepRap-style printer that is available on the market today. To that end, we have performed a quality check of your printer by printing out an entire set of plastics for the next Airwolf 3D printer. In other words, if the machine is not working correctly, first double check everything above, and then if it is still not working correctly, contact us and we can probably diagnose the problem in a matter of minutes, if not seconds. Rest assured that you have a very solid piece of equipment.
- About breaking things. Don't worry. Just call us and we should be able to supply you with replacement parts in a week or less at a reasonable price.
- Slicing
 - Slicing is the most critical part of the process and a correct slice will enable you to get the best print. Slicing takes an STL file and converts it into gcode instructions for your machine. We have many instructions for

slicing on our website as well as other informative matters related to 3D printing.

- We use Slic3r because it is user friendly, quick, and reliable. In particular, for the printed parts we use **Slic3r 7.2b** as it is very stable. Newer versions have many more features, but will require further experimentation.
- Once you start creating parts on your own, make sure you start with one of the Slic3r configuration files included with the software you receive (located in the same directory as the STL and GCODE files). You may adjust the speeds to control print time and quality, but pay attention to the defaults that we have inserted into the configuration files. These values include extrusion temperature (240 C), nozzle width (.5mm), print height (.25mm-.4mm) bed center location (97mm X, 113mm Y), ABS filament diameter (2.84mm), recommended speeds, and other critical information.
- Another hint: if Slic3r doesn't slice your part correctly, unless you have a very complex part, that part is most likely not "water tight." Use Meshlab or Netfabb to find out (and possibly correct) the problems with the part.
- ENJOY YOUR ADVENTURE!

Additional resources of information available on Airwolf 3D.com FAQ and Wiki:

<http://airwolf3d.com/3d-printer-support/>

Getting Started With Airwolf 3D Printer Slic3r Configurations	http://airwolf3d.com/blog/question/3d-printer-slic3r-configurations
Apply PET film on 3D printer glass	http://airwolf3d.com/blog/question/pet-film-applied-to-3d-printer-glass
How to Create a 3D Object in SketchUp for 3D Printing	http://airwolf3d.com/wiki/how-to-create-a-3d-object-in-sketchup-for-3d-printing/
How to Import an STL File into SketchUp	http://airwolf3d.com/wiki/how-to-convert-and-export-google-sketchup-skp-files-to-stl-for-3d-printing/
Manifolded Edges – How to Check if Your Model is Watertight and Ready to 3D Print	http://airwolf3d.com/wiki/manifolded-edges-how-to-check-if-your-model-is-watertight-and-ready-to-3d-print/
How to Convert and Export Google SketchUp SKP Files to STL for 3D Printing	http://airwolf3d.com/wiki/how-to-convert-and-export-google-sketchup-skp-files-to-stl-for-3d-printing/
Trouble Shooting a Jammed Hotend	http://airwolf3d.com/wiki/troubleshooting-jammed-hot-end/
Slicing 101	http://airwolf3d.com/wiki/slicing-1/
Height Adustment	http://airwolf3d.com/blog/question/height-adustment/
Layer Height	http://airwolf3d.com/blog/question/layer-height/
Getting the Best Prints	http://airwolf3d.com/blog/question/getting-the-best-prints/
Belt Tensioning	http://airwolf3d.com/blog/question/belt-tensioning/

ADJUSTING HEIGHT

Height adjustments are rarely necessary. Sometimes, however, it may be necessary to do so after the 3d printer is moved from one location to another.

The first layer of filament should be somewhat smashed down on the glass plate, like a fluffy pancake. It should not be too flat like a tortilla, however, the extruded filament should not be completely round like a hose. Below are some examples.

In figure 1, the nozzle is so high above the glass plate that it is dragging the filament tube around and the filament is hardly sticking to the glass.

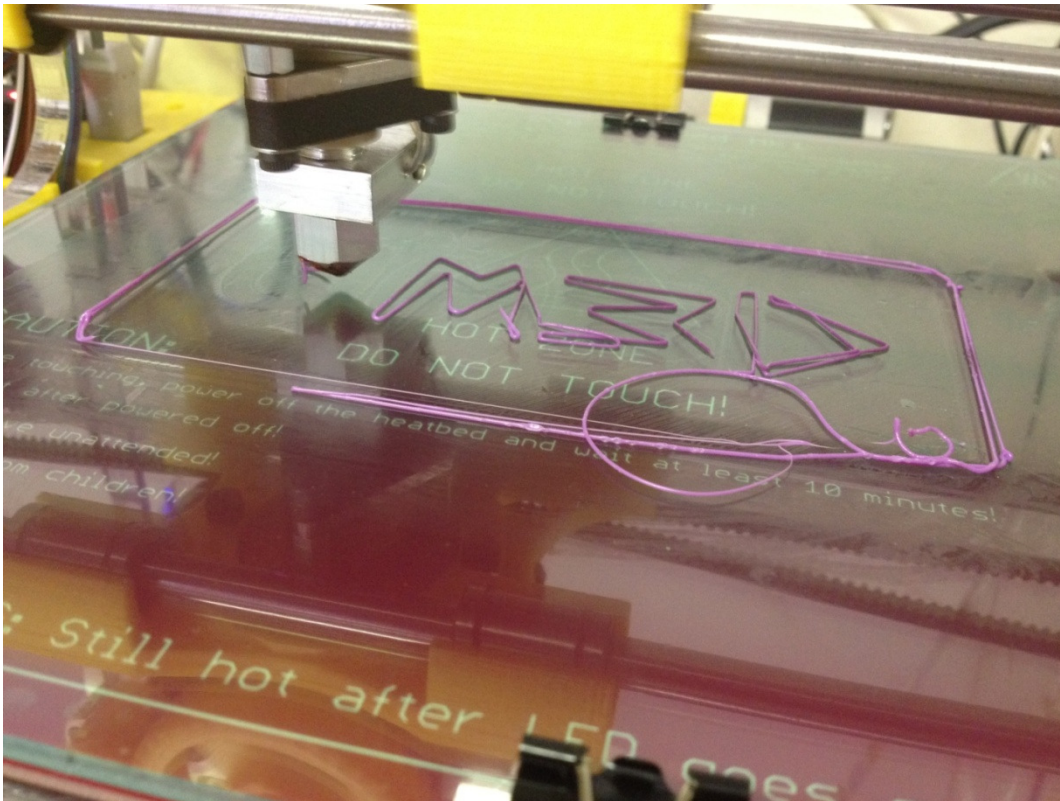
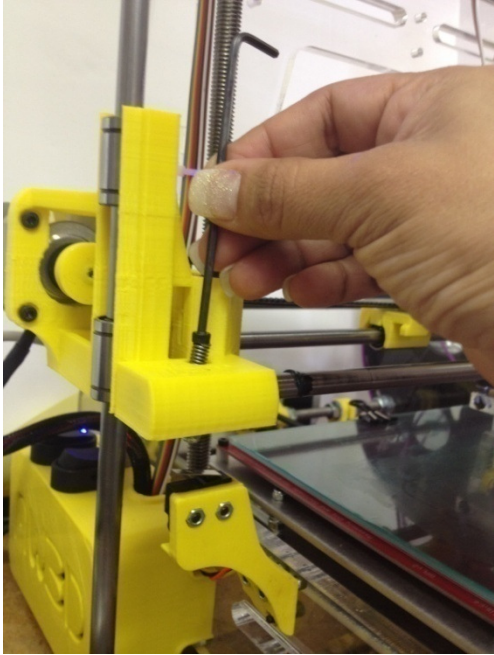


Figure 1 The nozzle is too high above the glass plate.



To adjust the height, use the supplied 2.5mm allen wrench and turn the screw counter clockwise to lower the nozzle and bring it closer to the glass plate. If an allen wrench is not available, you may also use needle nose pliers or your fingers to turn the bolt.

Figure 2 Turn The Bolt To Adjust Height

In figure 3, the height is better, however, still not optimum. The ABS is now sticking to the glass, however, it needs to be pressed in a little more and therefore the nozzle should be lowered a tiny bit more.

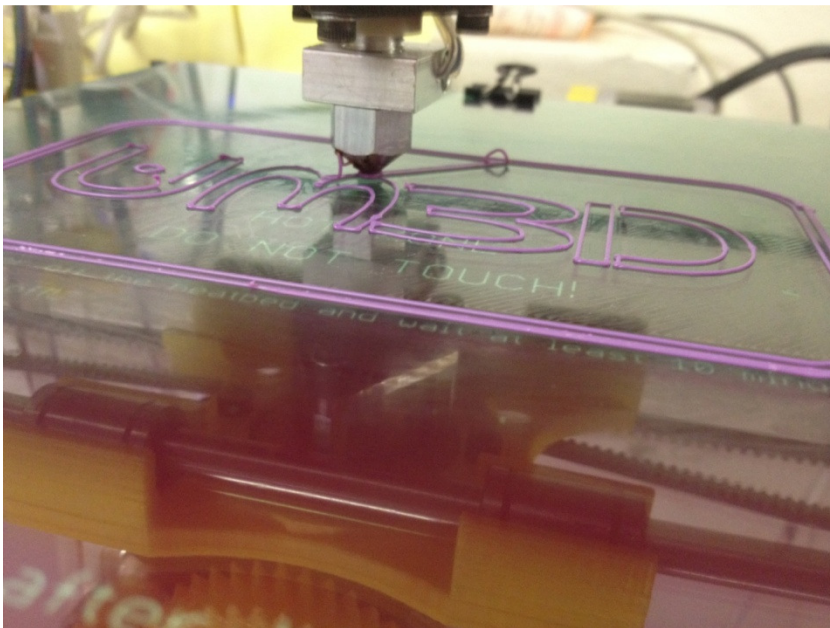


Figure 3 BETTER, Nozzle is Still Too High

In figure 4, the nozzle has been lowered slightly and is now printing at optimum height. The ABS filament is visibly pressed into the glass and will stick to provide best results. Caution: do not bring the nozzle down too low as this may cause the filament to get stuck in the extruder.

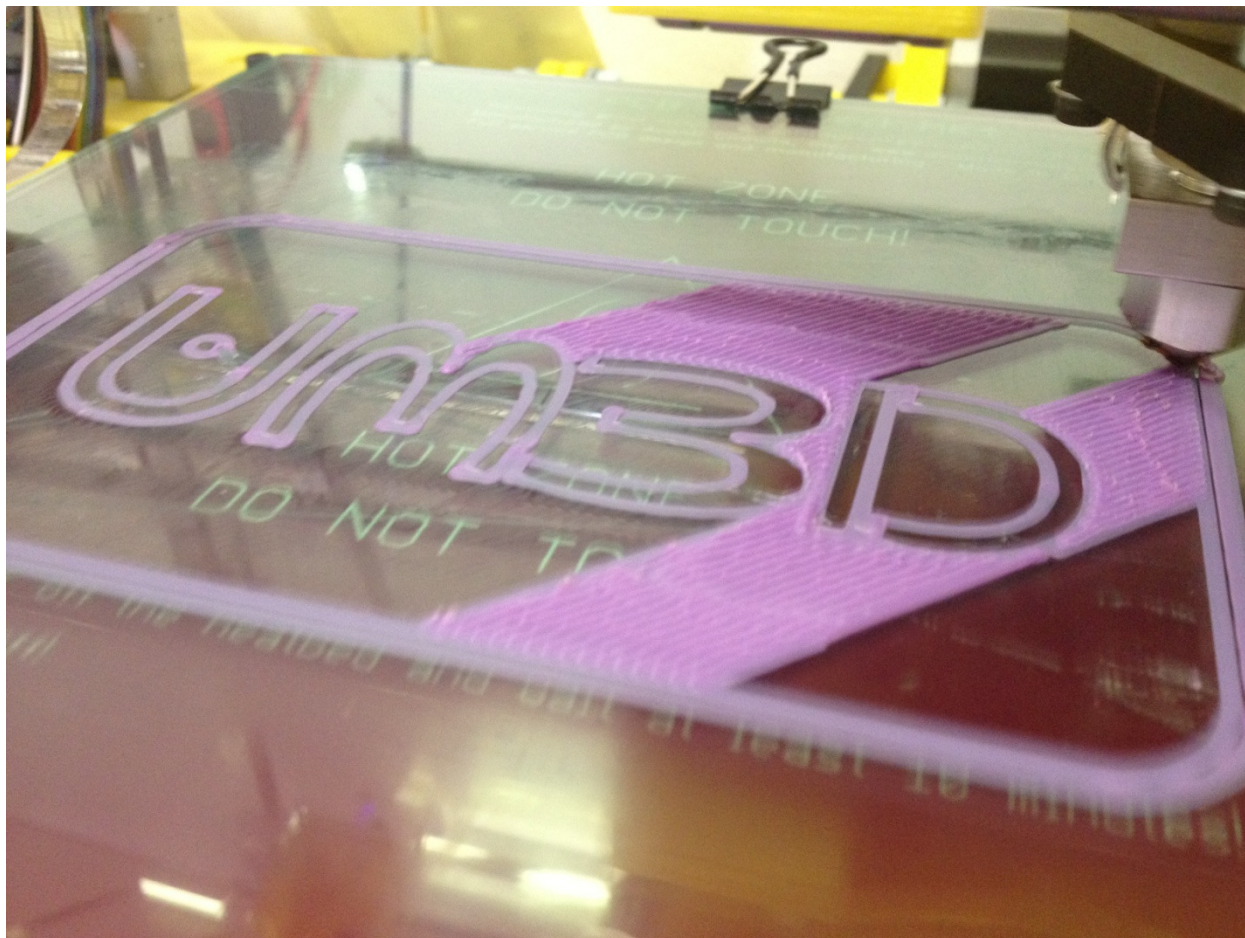


Figure 4 BEST, Nozzle is at Optimum Height



