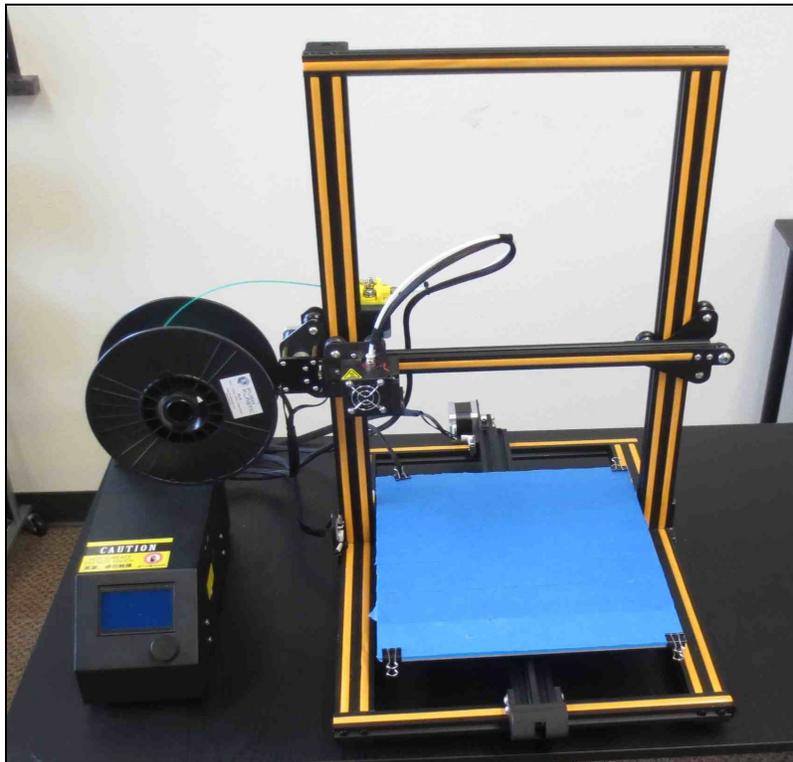




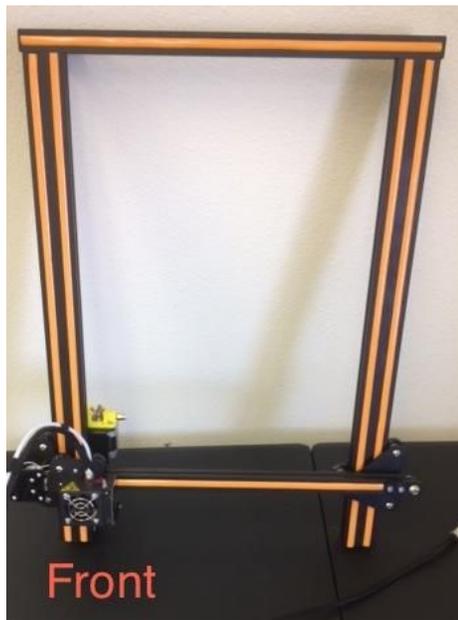
Afinibot A31 User Manual



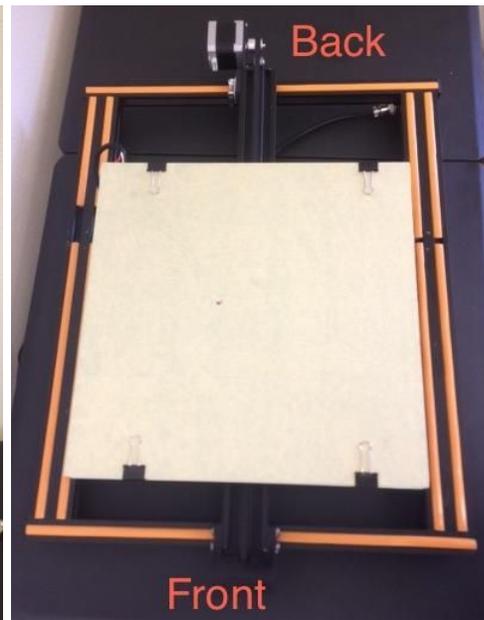
1. Assembly
2. Fine-Tuning
3. Leveling the Build Plate
4. Loading and Unloading Filament
5. Operation: The Four Steps of 3D Printing
6. Troubleshooting
7. Additional Resources

1. Assembly

1. Remove the parts from the box and remove any tape and padding from the parts. Inspect the parts to make sure they were not damaged in shipment.
2. Install the gantry frame (A) to the base frame (B).
 - a. On the gantry frame, make sure the nozzle assembly is to the front, and the long vertical lead screw is to the back. On the base frame, make sure the black belt cover is on the front, and the stepper motor is on the back.
 - b. Use the M5x20 screws (4) and lock washers (4). Raise the base frame above the table with two blocks (boxes, books, etc.) Install the screws through the base frame into the threaded holes in the gantry frame. Tighten with the M5 hex key (Allen) wrench.

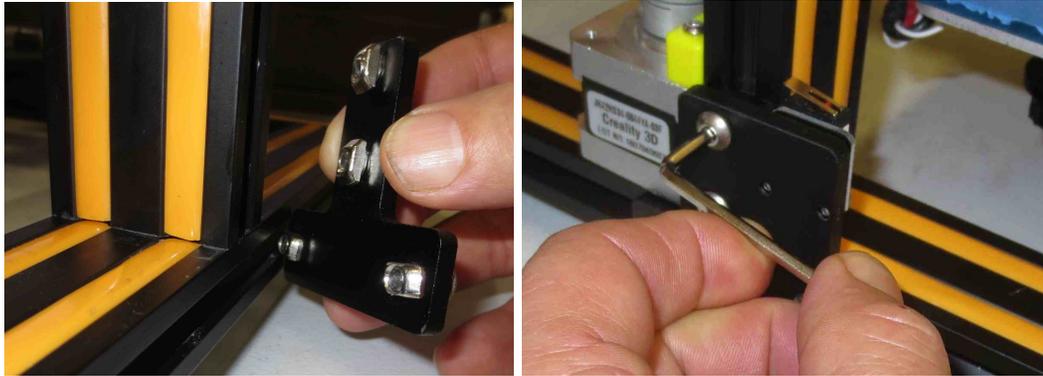


A

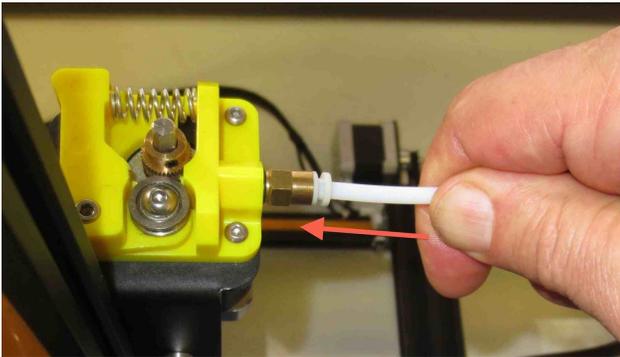


B

3. Install the two T-shaped frame reinforcement plates. The plate with the limit switch goes on the left. The plate without the limit switch goes on the right.
 - a. Loosen the t-nuts by hand and turn them so they will fit inside the grooves on the frames. You want the nuts to be loose so that when you tighten the bolts, the nuts will rotate 90° and grab onto the inside of the groove.
 - b. Align the nuts to the grooves and insert the T-shaped plates into the frames.



- c. Tighten all eight bolts using the M4 hex key (Allen) wrench.
4. Attach the white PTFE Bowden tube coming out of the hot end to the yellow tube coupler on the extruder. Firmly push the tube into the connector on the extruder. You should feel it slide in and lock into place.



5. Cut and discard the tie wrap holding the nozzle to the gantry frame.
6. Install the filament spool holder.
 - a. Attach the bracket to the top of the power & control box using two thumbscrews. Tighten the screws by hand. Do not overtighten.
 - b. Remove the first nut from the double-nut side of the filament holder tube. Install the tube into the bracket and re-install the nut to hold it in place. Tighten the two nuts by hand so that the tube is snugly attached to the

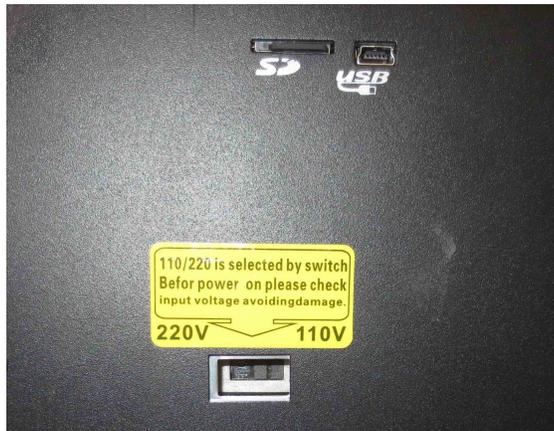
bracket. Do not overtighten.

- c. Tighten the nut on the other end of the tube by hand. Do not overtighten.



7. Connect all electrical cables.

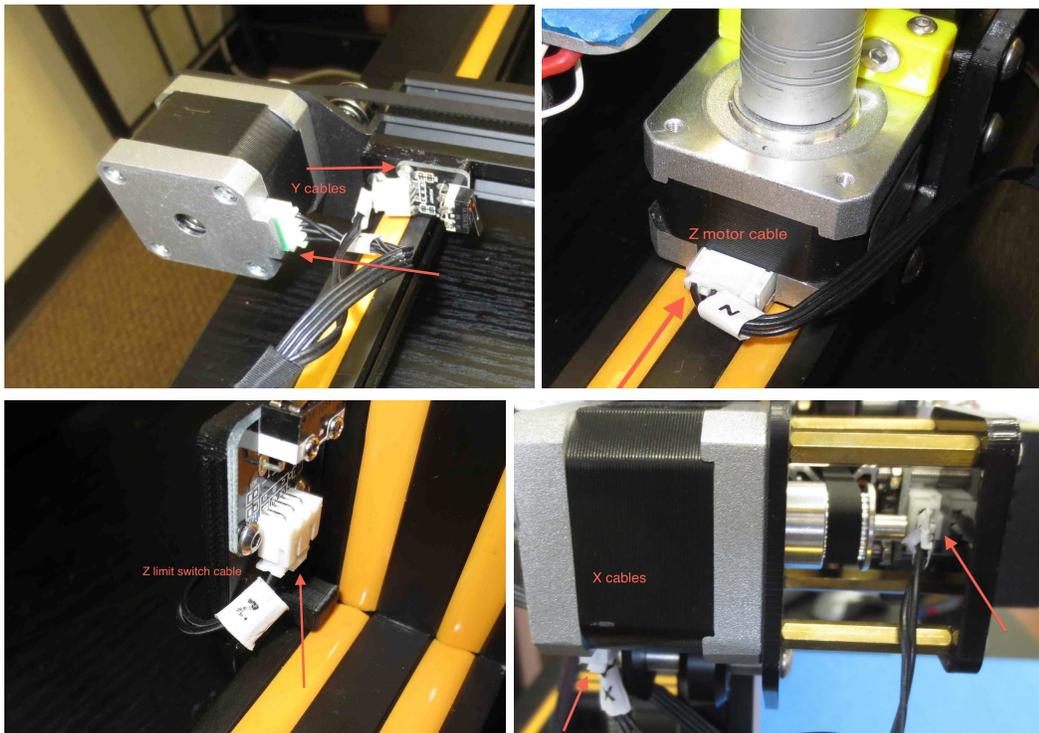
- a. Put the power & control box on the left side of the printer with the screen facing forward.
- b. On the right side of the power & control box, make sure the red voltage select switch reads “110V.” If it doesn’t, select “110V” by gently sliding the switch with a screwdriver or other tool.

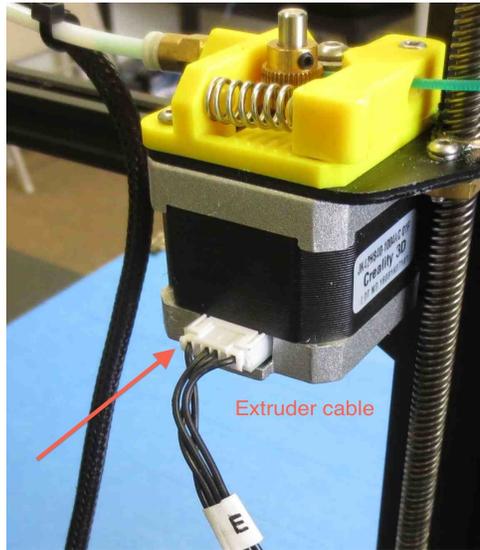


- c. Connect the heater cables to the power & control box. The cable for the build plate heater has four pins. The cable for the nozzle heater has eight pins. Rotate the connector until the pins slide easily into the socket. Do not force them. After the pins are fully inserted, hand-tighten the knurled nuts so they don’t become unplugged.



- d. Connect the stepper motor and limit switch cables.
 The Y cables go to the motor and switch at the back of the base frame that move the build plate. The Z cables go to the motor and switch on the left side of the base frame that raise and lower the nozzle. The X and E (extruder) cables go onto the gantry frame. The X cables go to the motor and switch that runs the belt and the E cable goes to the motor that feeds the filament. Hint: Install the smaller limit switch connectors first. Small hands are an advantage. Try not to bend the pin connectors as you push them in.





- e. Connect the power cord to the back of the power & control box and to a standard 110-volt electrical outlet.
- f. Turn the power on using the switch on the back of the power & control box.
- g. To operate the controls, rotate the button to scroll and push the button to select.

2. Fine-Tuning

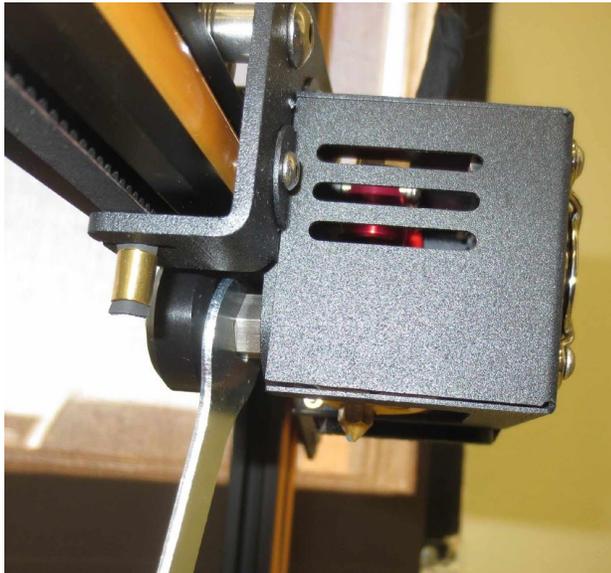
1. Adjust the tension of the build plate assembly on the Y-axis on the base frame.
 - a. Check the tension of the build plate assembly. Hold the build plate carriage under the build plate and try to wiggle it left and right. If the build plate assembly wiggles back and forth on the Y-axis track, it is too loose.
 - b. To tighten the build plate assembly, rotate the three eccentric nuts that connect the three wheels to the right side of the build plate assembly. Use the open-end wrench to rotate the nuts slightly—a quarter turn is usually enough—until each of the three wheels is snug against the Y-axis frame and the carriage no longer wobbles.



- c. Re-check the tension of the build plate assembly. The assembly and the belt should move forward and backward without much effort, but there should have no side-to-side wiggle or play.
2. Adjust the tension of the gantry on the right pillar of the gantry frame.
 - a. Check the tension of the gantry assembly at the right pillar of the frame. Hold the gantry and try to wiggle it up and down. If the gantry wobbles on the right pillar frame, it is too loose.
 - b. To tighten the carrgantryiage, rotate the one eccentric nut that connects the wheel to the inside of the gantry. Use the open-end wrench to rotate the nut slightly—a quarter turn is usually enough—until the wheel is snug against the frame pillar and the gantry no longer wobbles.



3. Adjust the tension of the nozzle assembly on the X-axis on the gantry frame.
 - a. Check the tension of the nozzle assembly. Hold the mounting bracket behind the nozzle and try to wiggle it back and forth. If the nozzle assembly wobbles on the X-axis track, it is too loose.
 - b. To tighten the nozzle assembly, rotate the one eccentric nut that connects the wheel to the bottom of the nozzle assembly. Use the open-end wrench to rotate the nut slightly—a quarter turn is usually enough—until the wheel is snug against the X-axis frame and the assembly no longer wobbles.



- c. Re-check the tension of the nozzle assembly. The assembly and the belt should move left and right without much effort, but there should have no side-to-side wiggle or play.

4. Check the tension of the belt driving the Y-axis (under the build plate). The belt should be taut, with no slack or slop.

- a. If the belt is loose:

Loosen the four bolts at the front of the base frame holding the belt follower pulley to the base frame.

Using an Allen wrench as a lever, push the follower pulley to tighten the belt.

Holding the belt taut, tighten the four bolts.



5. Check the tension of the belt driving the X-axis (on the gantry). The belt should be taut, with no slack or slop.

- a. If the belt is loose:

Loosen the two bolts at the right side of the gantry holding the belt follower pulley to the gantry.

Using an Allen wrench as a lever, push the follower pulley to tighten the belt.

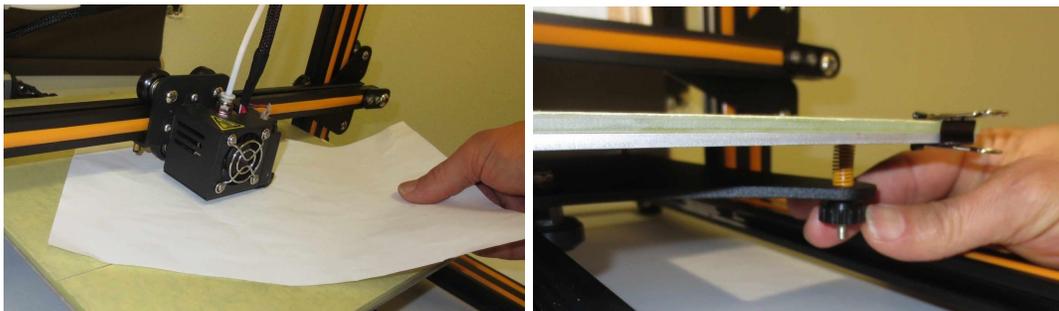
Holding the belt taut, tighten the two bolts.



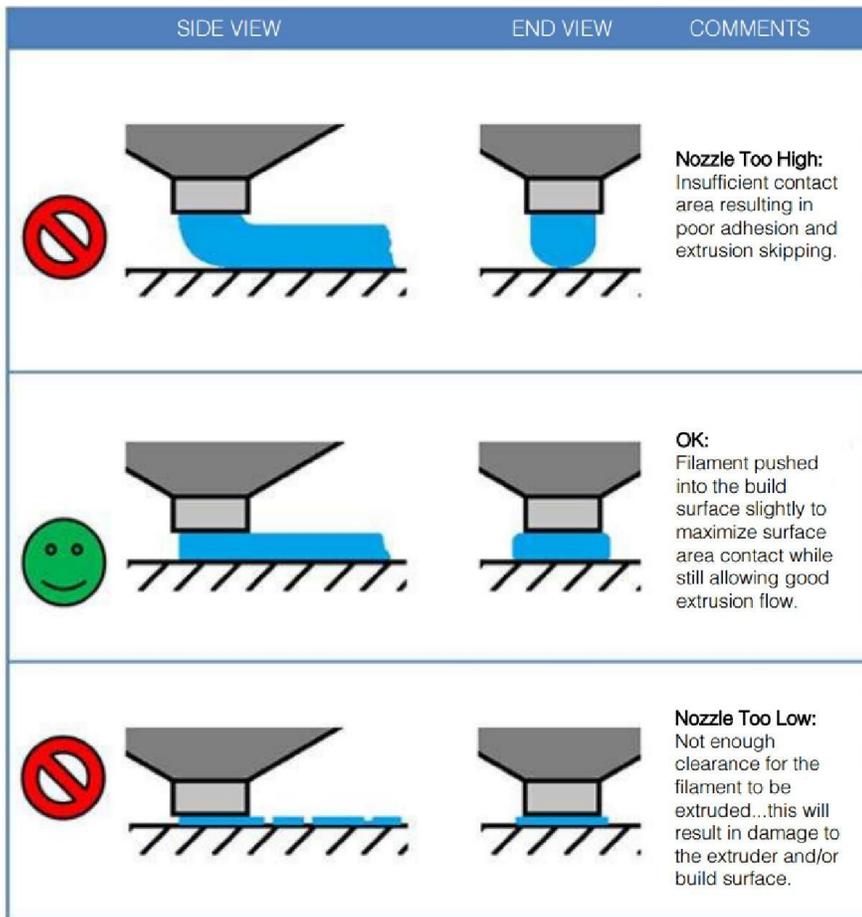
3. Leveling the Build Plate

To build good parts, the build plate needs to be level, and the nozzle needs to be about 0.1–0.2mm from the build plate in all locations. If you have a LokBuild print surface, we recommend a distance of 0.20–0.25mm. This is about the thickness of two pieces of paper (or one piece folded in half to double its thickness). If you use painter's tape as your print surface, we recommend a distance of 0.1mm. This is about the thickness of a single piece of paper. You want to adjust the height of the build plate so that you can barely slide the paper between the nozzle and the build plate with only a little resistance.

1. Select Control > Temperature > Bed. Turn the bed temperature to 55°. Then navigate back to the Info Screen. This will heat the bed to actual printing conditions, making the leveling more accurate.
2. Select Prepare > Preheat PLA. This will heat the nozzle and melt any hardened plastic that might be on the nozzle and affect the leveling process.
3. Select Prepare > Auto home. This will move the nozzle to the home position at the front left corner of the build plate.
4. Select Prepare > Disable steppers. This will allow you to move the nozzle assembly (X axis) and build plate (Y axis) by hand.
5. Pro Tip: The Z axis stepper is also disabled, and you don't want it to move at all. So you should handle the printer gently during the leveling process. Keep it flat and move the nozzle assembly carefully.
6. Wait for the print bed temperature to reach 55° and the nozzle temperature to reach 195°C. The Info screen displays this information below the nozzle and bed icons.
7. Slide a piece of paper between the nozzle and the build plate.
8. Adjust each of the four thumbscrews under the bed until the piece of paper slides, with just a tiny bit of drag, in all locations on the build plate.



9. Select Prepare > Auto home, then Prepare > Disable steppers, and recheck in several places. This is to make sure the Z-axis lead screw didn't rotate during the leveling process.
10. You may need to make fine adjustments to the bed level when you start printing. The first layer of the print will show whether the distance between the nozzle and build plate is correct. Refer to the following graphic.



- a. If the bed is too low, the extruded plastic may not stick to the build plate. Parallel lines of filament on the first layer will look rough, with high tracks and low valleys that are not stuck to the build plate.

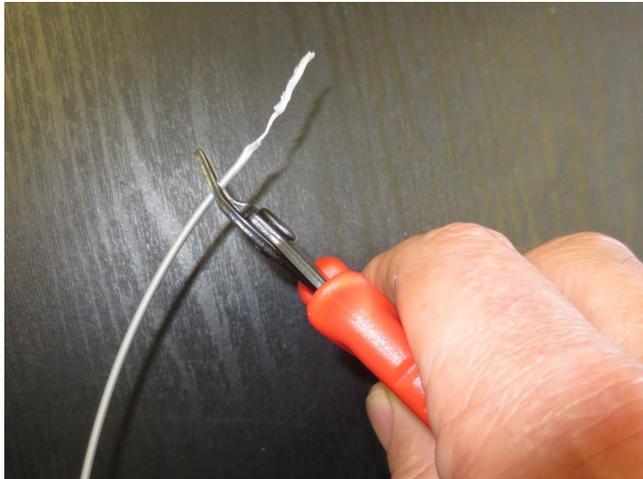


- b. If the bed is too high, the extruded plastic will squeeze out the sides of the nozzle. The nozzle will plow through the plastic and leave a first layer that is too thin.
- c. You can try to carefully adjust the thumbscrews during the first layer of the build while the plate is moving until the distance between the build plate and the nozzle is producing smooth extruded lines. Be careful of the moving parts to not get pinched.
- d. After you have fine-tuned the bed level during the first layer, you may want to stop the build, clear the build plate, and restart the build.

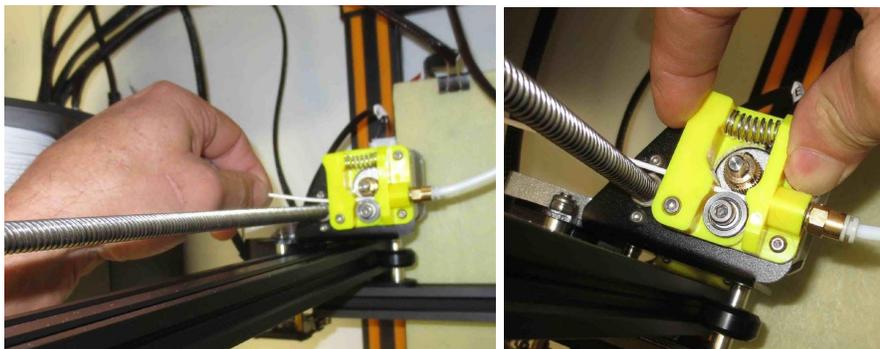
4. Loading and Removing Filament

1. Loading filament

- a. Load a spool of filament into the spool holder.
- b. Cut off the used or bent end of the filament at an angle and it will be easier to feed into the extruder and Bowden tube.



- c. Feed the end of the filament into the extruder with one hand as you squeeze the extruder release lever with your other hand.



- d. Push the filament all the way through the Bowden tube until it stops when it hits the back of the nozzle.

- e. Select Prepare > Preheat PLA.
- f. When the printer is fully heated, check to make sure the filament is fully loaded. This also removes all of the previous material left in the printer from other prints. To do this, manually push filament through the heated nozzle until the new color begins coming out of the nozzle.
- g. Pro Tip: Never leave the printer preheated with material inside of it for long periods of time. This will cause the filament to bake inside of the nozzle assembly and can cause it to clog.

2. Removing filament

To remove the filament cleanly, you should do a “soft pull.” This involves heating the nozzle to 100°C, then pulling the filament out of the printer.

- a. Select Prepare > Preheat PLA and wait until the nozzle temperature reaches 100°C. (You can also select Control > Temperature > Nozzle, spin the temperature to 100°C, and then navigate back to the Info Screen.)
- b. Wait for the nozzle to reach the set temperature.
- c. When the nozzle temperature is 100°C, squeeze the extruder release lever with one hand and pull the filament out of the extruder with your other hand in one fluid motion.
- d. Cut the end of the filament to remove the partly melted plastic.
- e. Pro Tip: Feed the end of the filament into the hole on the side of the spool. This will insure it doesn't get tangled and cause a build to fail the next time you use it.
- f. Remove the filament spool from the spool holder.



5. Operation: The Four Steps of 3D Printing

3D printing involves four steps:

1. Create or find a 3D file
2. Slice the file for 3D printing
3. Send the file to the printer
4. Print!

1. Create or find a 3D file

- a. You can create a 3D model—a digital geometry—using any CAD (computer-aided design) software program. We recommend free programs, such as [TinkerCAD](#), [OnShape](#), and [Blender](#). Another is [SketchUp](#), which has free licenses available for schools. These programs take some time to learn, and can be challenging. But they are also rewarding, especially when you see your own creation being printed on your 3D printer!

Every CAD program allows users to save or export digital files for 3D printing. You want to save your file as an STL file (“filename.stl”). The STL file is the standard file format for 3D printing. In some programs, it’s as easy as the Save command. In others, you need to select the File > Export option, and choose the STL format. OBJ files can also be sliced and 3D printed.

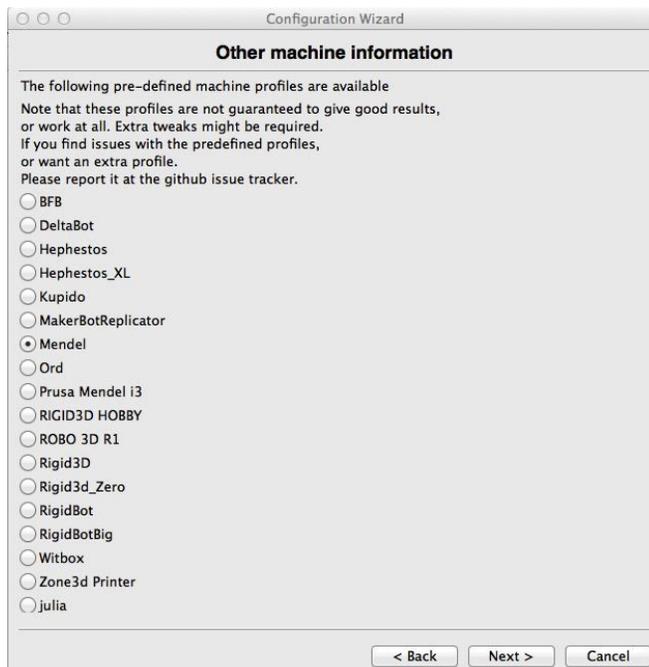
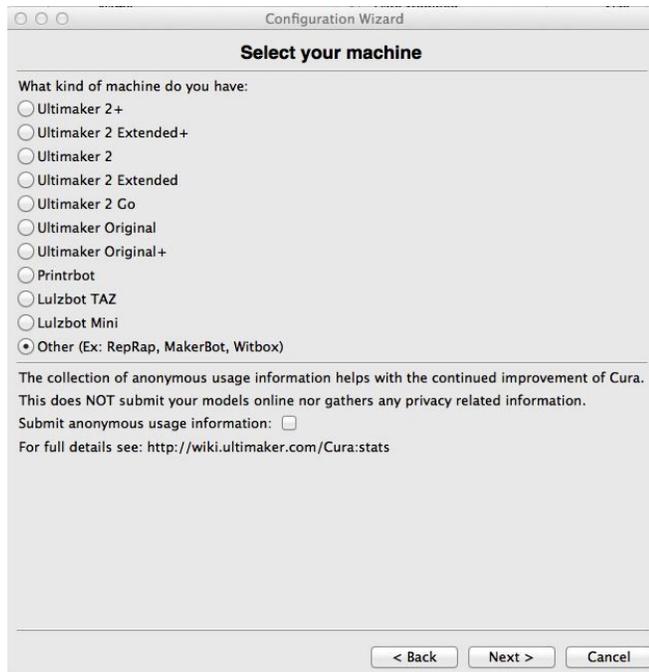
- b. You can find 3D models in many online libraries of digital content. [Thingiverse.com](#), [GrabCAD.com](#), and [Instructables.com](#) are examples, and many other sites exist.

2. Slice the file for 3D printing

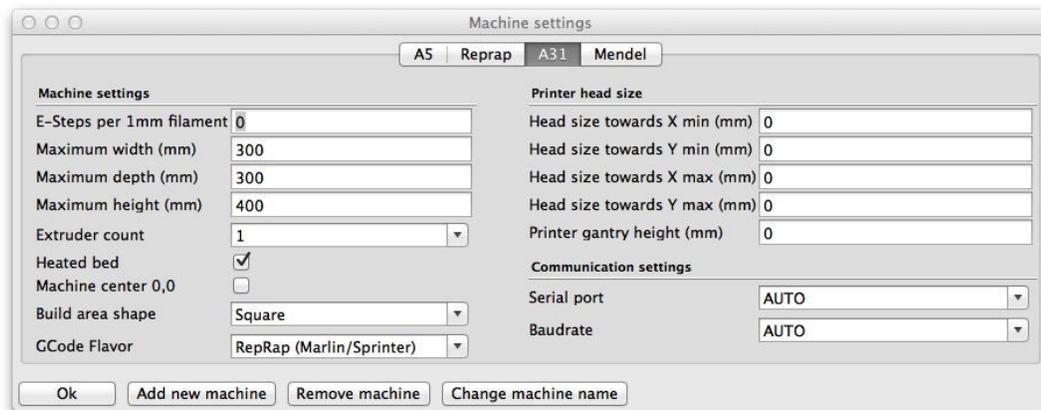
Slicing is the process of converting your 3D model into the layered print language your printer can understand, called G-Code. We recommend using free, open-source slicer programs. Our favorite slicers are [Cura](#) and [Repetier Host](#). Both are free and easy to use. We’ve included Cura and its setup .ini file on your printer’s SD card because it’s easier for beginners. We recommend that you install Cura on a PC or Mac and use it to do your slicing.

There are a lot of slicers that will create G-Code automatically for your printer. All you have to do is input the correct settings for your printer (using the files we included), import your 3D model, and click slice! It’s that easy.

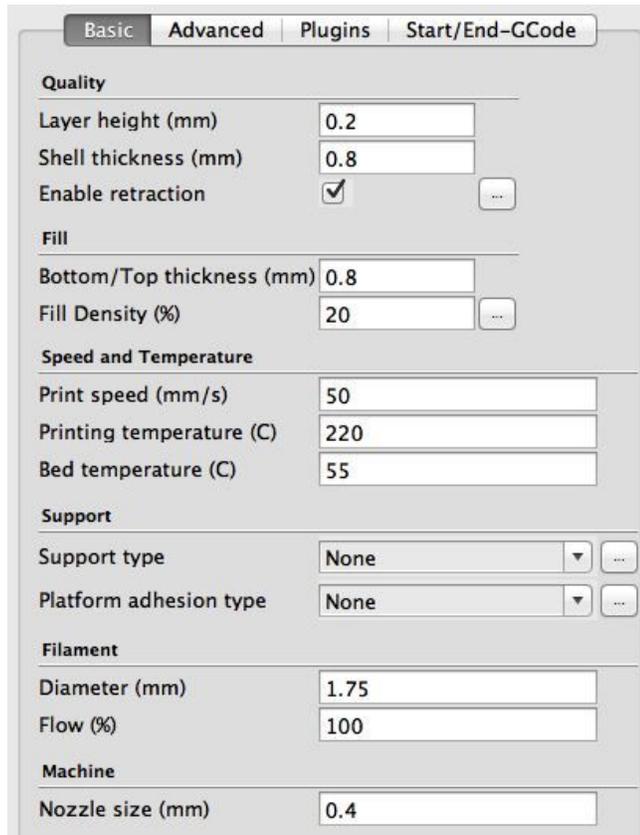
- a. Install the Cura program on a PC or Mac.
- b. Open the Cura application. You'll be given several options. In the Configuration Wizard, choose the RepRap Machine and Mendel in other machine settings (RepRap (Marlin/Sprinter) G-Code Flavor, if it asks).



- c. Once you add the new machine, click Machine > Machine settings and change the machine settings to: Max width 300, Max depth 300, Max height 400, Extruder count 1, and check Heated bed. You can also rename the machine to “A31” or “Our Giant 3D Printer” if you’d like. Click “OK.”

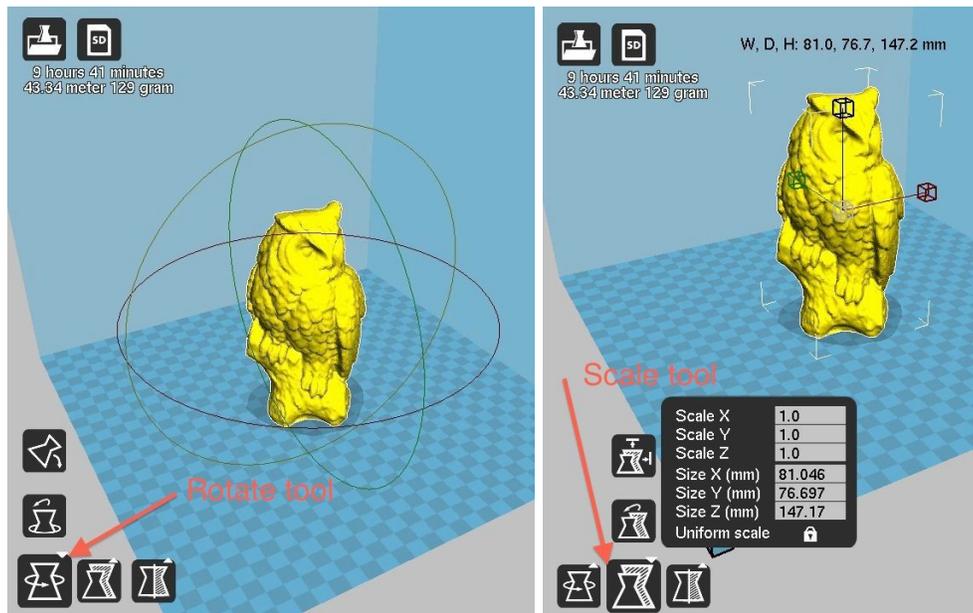


- d. Select File > Open Profile and select “NWA3D A31 Printer Profile.ini” from the microSD card. This is the medium-quality settings for the A31 printer.
- e. The “A31 Printer Profile.ini” settings in Cura for the A31 are shown on a screen capture on the microSD card. They are also shown in the following image.



Section	Parameter	Value
Quality	Layer height (mm)	0.2
	Shell thickness (mm)	0.8
	Enable retraction	<input checked="" type="checkbox"/>
Fill	Bottom/Top thickness (mm)	0.8
	Fill Density (%)	20
Speed and Temperature	Print speed (mm/s)	50
	Printing temperature (C)	220
	Bed temperature (C)	55
Support	Support type	None
	Platform adhesion type	None
Filament	Diameter (mm)	1.75
	Flow (%)	100
Machine	Nozzle size (mm)	0.4

- f. Use these default settings for printing. You may need to change the support type to “Everywhere” depending on the shape of the part you are printing. Also, you can change the layer height from 0.2mm to 0.1mm for a smoother part, although the print will take longer. As you become more advanced with 3D printing, you may want to experiment with many settings. But remember that you can always revert to these default settings at any time, or ask us for help at nwa3d.com/support.
- g. Load the model file in Cura. Rotate the file to the best orientation for printing. You may want to scale the file at this point as well.



- h. When you are satisfied with the Cura settings and the position, orientation, and scale of the model, select the “Save toolpath” (or “SD” if the microSD card is inserted) icon or File > Save GCode. Save the file to the microSD card that came with your printer.



3. Send the file to the printer
 - a. After saving the G-Code to the it, remove the microSD card from the computer.
 - b. Insert the microSD card, upside-down, into the slot on the right side of the control box.
 - c. By using this method, no computer needs to be plugged into your printer and it will run autonomously until the print is finished.

4. Print!
 - a. On the A31 control screen, select Init./Change SD card, the last selection on the bottom of the main screen.
 - b. Select Print from SD.
 - c. Select your print (.gcode) file.
 - d. Watch the 3D printer create your model!

6. Troubleshooting

This section covers the most common printing problems and how to fix them. Because 3D printing is still an emerging technology, a small level of tinkering and troubleshooting is needed. But if this section doesn't solve your printing problems, then contact us by filling out the [Troubleshooting Request](#) on our website. We're here to help you every step of the way to get you 3D printing!

1. Step-by-Step Troubleshooting

a. Step 1: Diagnose the problem.

This may seem obvious, but many problems can be solved if you take a step back and see exactly what the printer is doing incorrectly.

b. Step 2: Determine if the problem is mechanical or digital.

- Mechanical problems are in the actual operation of the printer, such as the motors that drive the X, Y, and Z axes, the motor that pushes the filament, the nozzle heater, and the level of the build plate. The most common mechanical problems are caused by a build plate that is not adjusted correctly, a clogged nozzle, or an unplugged connector on a motor or limit switch.
- Digital problems are in the slice file that you prepare in Cura or other slicing program. Important slice file settings include layer height, print (nozzle) temperature, and print speed.

c. Step 3: Fix the problem.

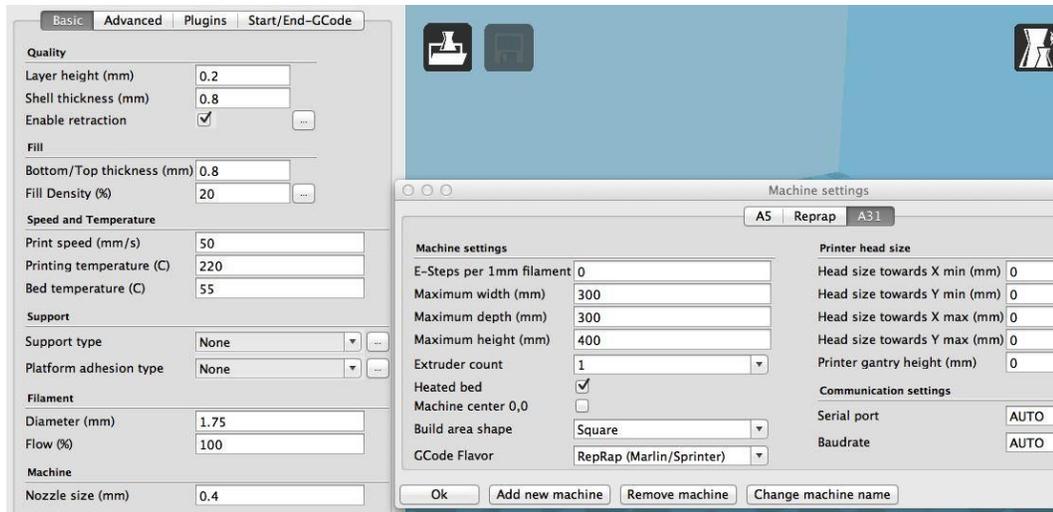
Once you've discovered what the actual problem is, you can fix it with the following procedures. (If none of these fix the problem, [contact us](#).)

2. Digital problems: Slicer settings

- a. Check your slice file in Cura. Make sure the print is centered in the build area, making good contact with the build surface, and isn't too big for the build envelope.
- b. Check the slice settings. Make sure that the layer height is between 0.1mm (high quality prints) and 0.3mm (low quality prints).
- c. Check the speed and temperature. For PLA, they should be set to 30–50mm/s and 230°C.

- d. Make sure the filament diameter is 1.75mm and the flow rate is 100%.
- e. Check the Fill Density (infill) and make sure it is at least 5%. You may need to adjust this value to your liking for your model.
- f. Your part may need supports. If it's rounded you may need to select "Everywhere" for the support type. If it's intricate, it may need to select "Raft" for the Platform adhesion type. A raft is a hatch pattern on the build plate that the model will print on. If the part warps when you begin to build it, you may need to select "Brim" Adhesion support type and reslice the file. (A brim will help the part stick so it doesn't curl up at the edges.)

If your slice settings are really messed up, try re-loading the Printer Profile.ini file from the microSD card. The profile will configure medium quality settings that will automatically reset all the values to ones that will work well. Click File > Open Profile to navigate to the printer profile "NWA3D A31 Printer Profile.ini" that came on the microSD card in the Cura folder.



3. Mechanical problem: Leveling the build plate

Many of the failures in 3D printing are caused by a build plate that is not level or is not the correct distance from the nozzle. See the section "Leveling the Build Plate" in this manual.

4. Mechanical problem: Filament is not coming out of the nozzle

When filament is not coming out of the nozzle, the nozzle may be clogged. You have several options for clearing a clogged nozzle.

- You can pull the clog backward out of the nozzle using the “soft pull” method.
- You can “floss” the clog out of the nozzle using a nozzle cleaner while the nozzle is heated.
- You can heat the nozzle and push the clog through the nozzle.

[This video explains how to unclog a nozzle](#)

NOTE: Never leave your printer nozzle at build temperature (220°C) while the printer is idle. The melted plastic remaining in the nozzle will “bake” onto the nozzle and become a hard carbon blockage.

- a. Try to pull the clog out of the nozzle using the “soft pull” method:
 - Turn the machine on, select Prepare PLA, and wait until the temperature reaches 100°C.
 - When the nozzle temperature reaches 100°C, squeeze the extruder release lever with one hand and pull the filament out of the extruder with the other hand.
 - Inspect the end of the filament. If the end of the filament has the shape of interior of the nozzle—a thicker cone-shaped plug—you have successfully pulled all of the filament out of the nozzle, including the clog.
 - If the clog did not clear using the “soft pull” method, repeat the process.



NOTE: Every time you change filament on your 3D printer, remove the filament with the nozzle at around 100°C. This method usually removes all the old color of filament from the hot end, so when you begin printing again, the new color will begin printing immediately.

- b. Use a nozzle cleaner to clear the nozzle:
 - Select Prepare > Preheat PLA. Wait for the nozzle to reach the set temperature of 195°C for PLA.
 - Raise the Z-axis carriage by selecting Prepare > Move axis > Move 1mm > Move Z and spinning the knob. (You can also manually rotate the lead screw to move the carriage up.)
 - When the nozzle has reached 195°C, carefully insert the nozzle cleaner up into the tip of the nozzle. Squeeze the extruder release lever with one hand and push the filament into the extruder with your other hand.
 - Alternate “flossing” with the nozzle cleaner and pushing the filament through the extruder until a consistent bead of filament is extruded through the nozzle.
 - Reload the filament and move on to step c.
- c. Try to push the clog through the nozzle:
 - Select Prepare > Preheat PLA. Wait for the nozzle to reach the set temperature of 195°C for PLA.
 - When the nozzle has reached 195°C, squeeze the extruder release lever with one hand and push the filament into the extruder with your other hand.
 - Watch to see if any plastic is coming out of the nozzle.
 - Turn the machine off and wait 10 minutes. The nozzle needs to cool down completely.
 - Do a “soft pull” (step a.) to remove remaining filament in the nozzle.



7. Additional Resources

nwa3d.com

[NWA3D's Troubleshooting Form](#)

[How to unclog a nozzle](#)

Simplify3D Troubleshooting page provides good explanations and photographs of common problems in 3D printing:

<https://www.simplify3d.com/support/print-quality-troubleshooting/>