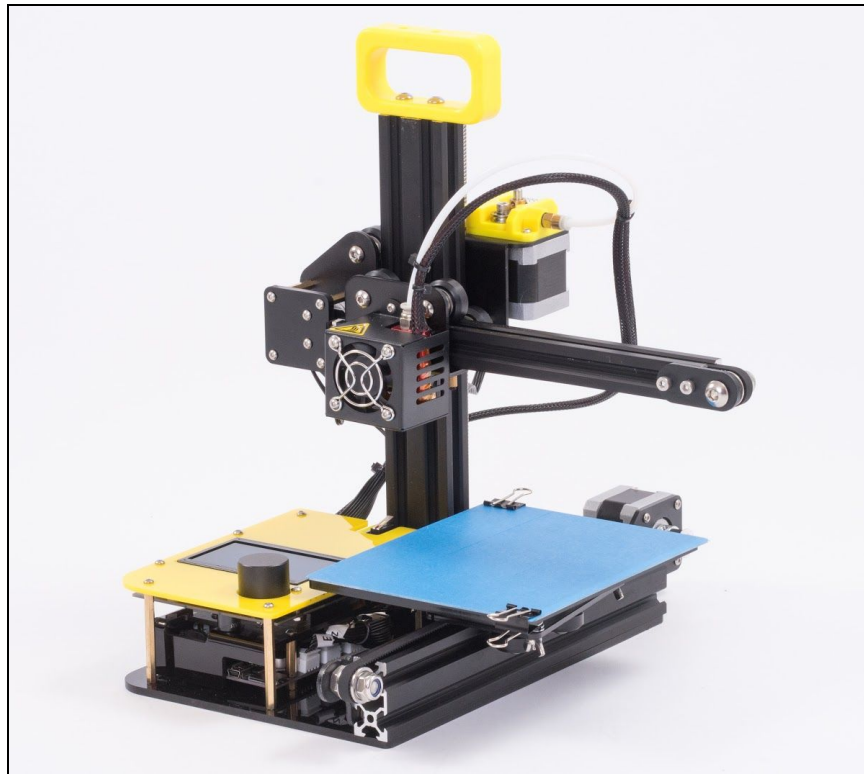


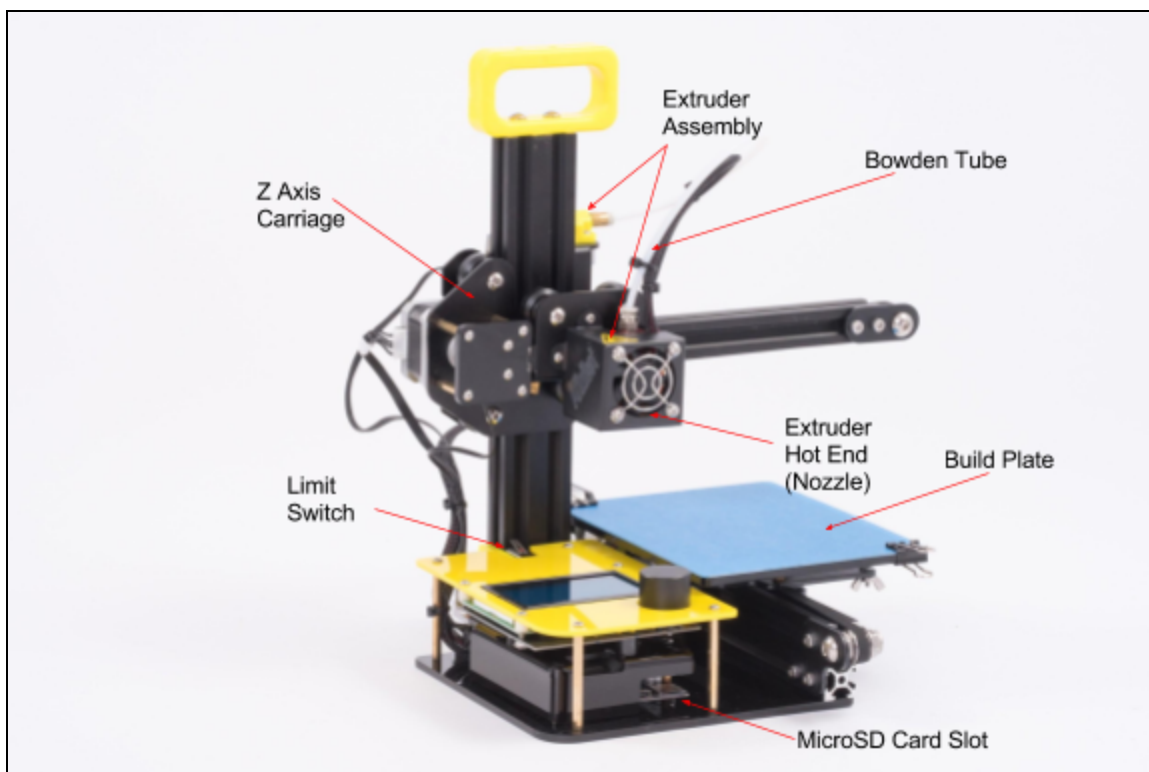
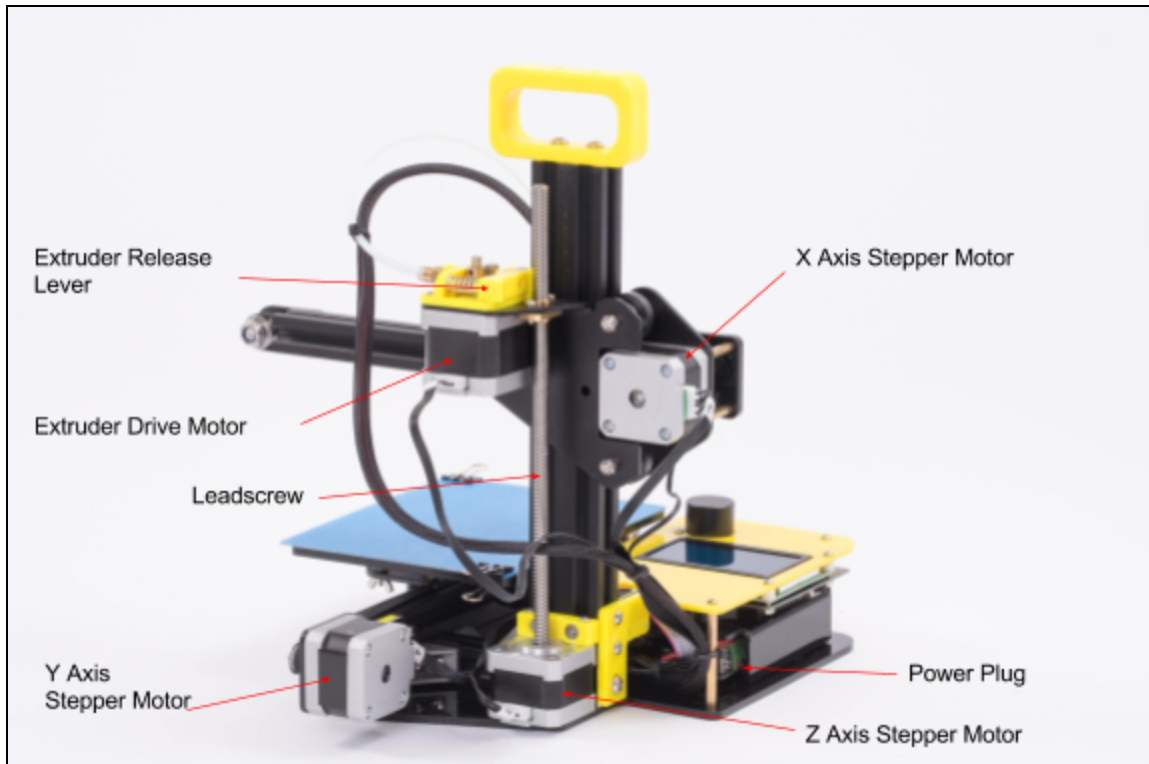


## NWA3D A5 User Manual



1. NWA3D A5 3D Printer Part Diagrams
2. Assembling the Spool Holder
3. Leveling the Build Plate
4. Loading and Unloading Filament
5. Operation: The Four Steps in 3D Printing
6. Troubleshooting
7. Additional Resources

## 1. NWA3D A5 3D Printer Part Diagrams

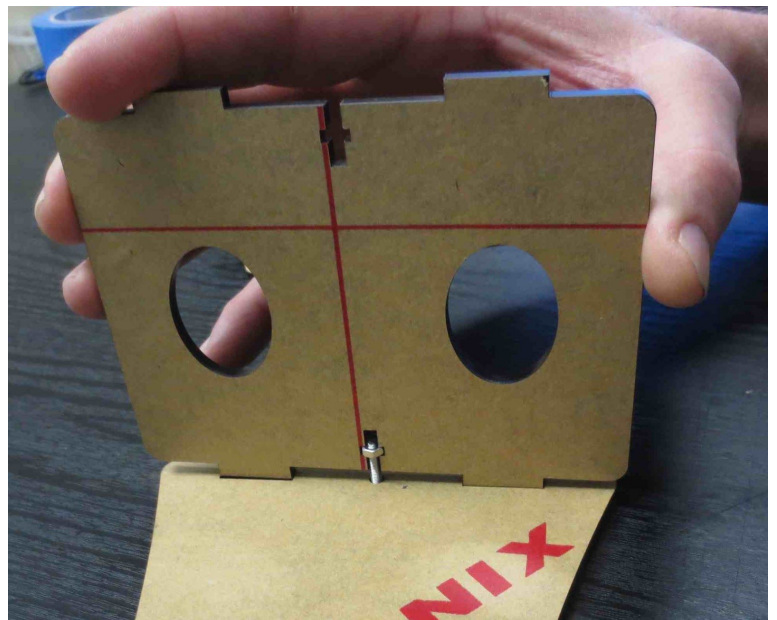


## 2. Assembling the Spool Holder

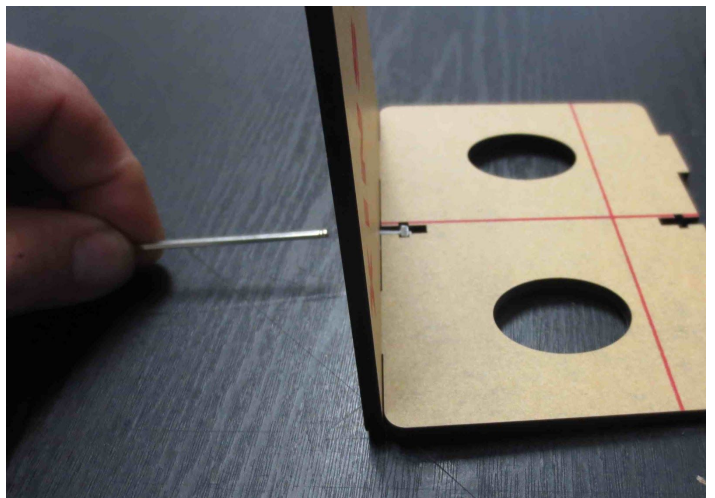
1. Loosen the nuts on the two bolts installed on the spool holder side pieces until they are near the end of the bolt threads—but don't remove them completely.



2. Slide the base of the spool holder into the side piece while at the same time sliding the nut into the small groove in the base.



3. Attach the side piece to the base by tightening the bolt with a 3mm hex key (Allen) wrench. Do not over-tighten.
4. Repeat Steps 2 and 3 to attach the other side piece.



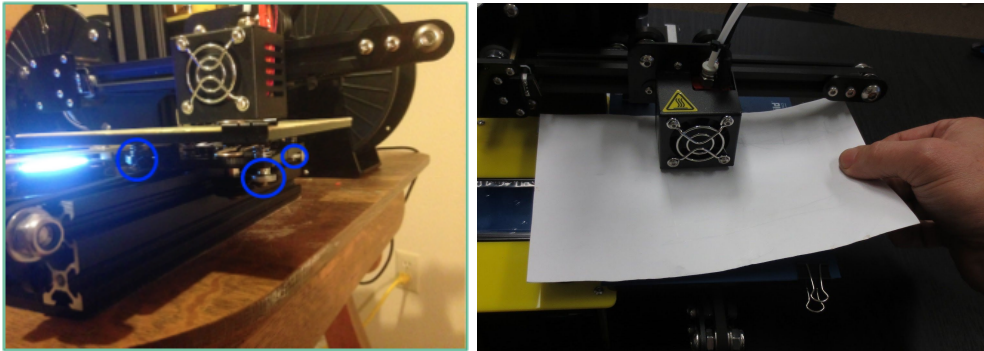


### 3. Leveling the Build Plate

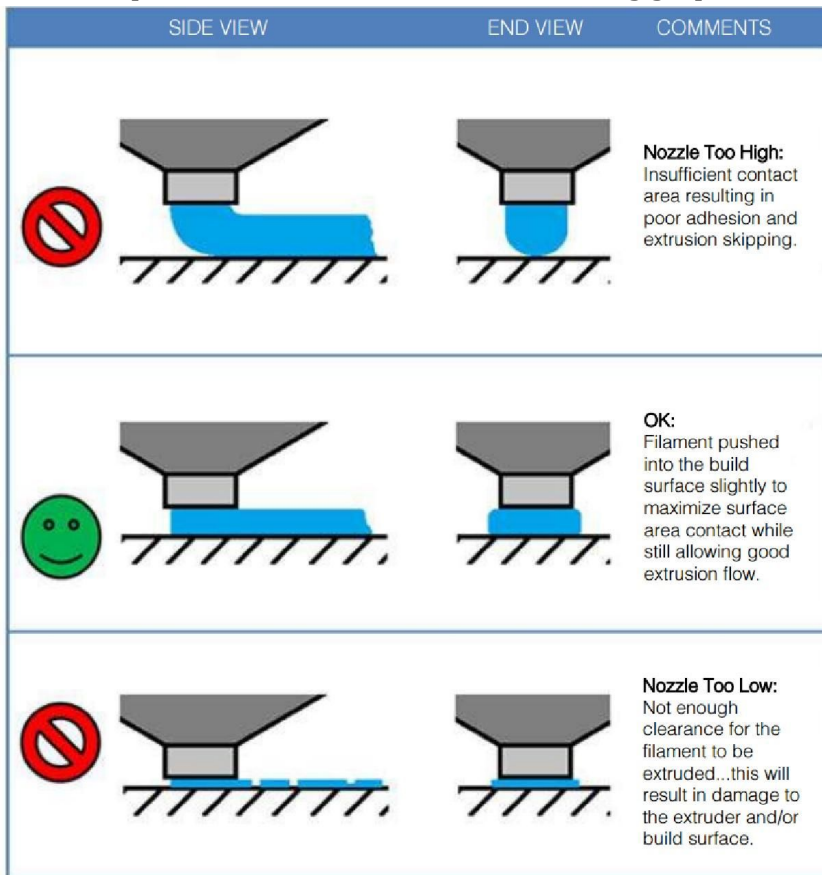
[Click here to watch the video on how to level the build plate](#)

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 0.2mm. This is about the thickness of two pieces of printer 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 printer 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 Setup > Preheat PLA. This will heat the nozzle to melt any hardened plastic that might be on the nozzle and affect the leveling process.
2. Select Setup > Auto home. This will move the X, Y, and Z axis stepper motors to their home positions (0,0,0).
3. When the motors stop, select Setup > Disable steppers. This will allow you to move the “hot end” of the extruder assembly (X axis) and build plate (Y axis) by hand.
4. **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 on a flat surface and don't tip it while leveling.
5. Slide a piece of paper between the nozzle and the build plate. For LokBuild print surfaces, first fold the paper in half to double its thickness to about 0.2mm.
6. Move the hot end (X axis) and build plate (Y axis) so that the nozzle is directly above one of the three wingnuts under the bed. Adjust each of the wingnuts under the bed until the paper slides with just a tiny bit of drag. We recommend adjusting the wingnut on the left first, then the two on the right, then repeating the adjustment at all three locations at least once more.
7. Make sure the paper slides with just a bit of drag at all locations on the build plate, including the center of the build plate. Try and get the amount of tension on the paper to be consistent at all locations.
8. Select Setup > Auto home, then Setup > Disable steppers, and recheck in several places. This is to make sure the Z axis lead screw didn't rotate during the leveling process.



9. The first layer of the print will show whether the distance between the nozzle and build plate is correct. Refer to the following graphic for clarity:

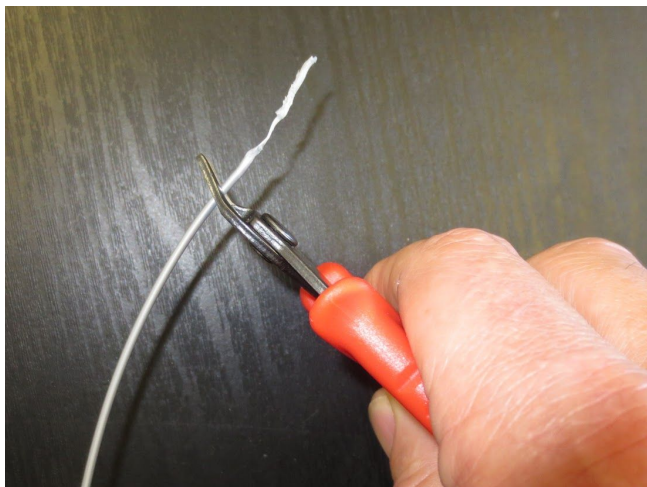


- a. If the bed is too high, the extruded plastic will squeeze out the sides of the nozzle. The nozzle will plow the plastic and leave a first layer that is too thin.
- b. If the bed is too low, the extruded plastic may not stick to the build plate. Parallel lines of filament on the first layer will look rough, with high tracks and low valleys.
- c. 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.

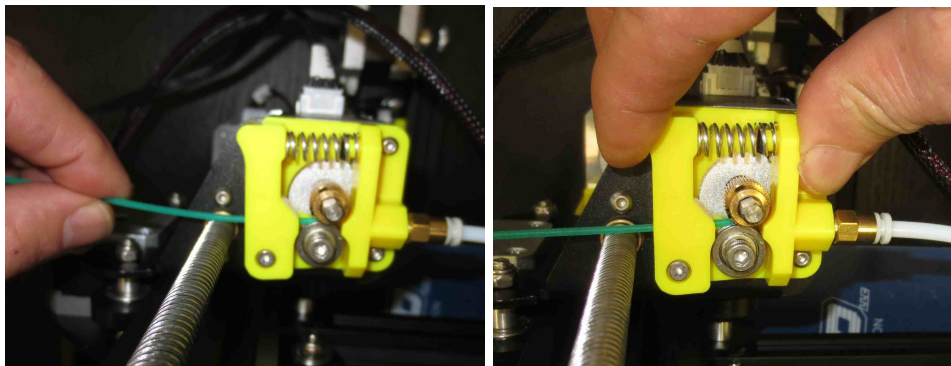
## 4. Loading and Removing Filament

### 1. Loading filament

- a. Load a spool of filament into the spool holder. Slide the bolt through the spool's center hole, and then place the bolt into the recessed grooves on the spool holder.
- b. Cut the end of the filament at an angle so the irregular melted plastic at the end doesn't cause a clog. It will also be easier to feed into the extruder and Bowden tube.



- c. Feed the end of the filament into the extruder by hand as you squeeze the extruder release lever.



- d. Push the filament all the way through the white Bowden tube until it stops when it hits the back of the nozzle.
- e. Select Setup > Preheat PLA.

- f. When the printer is fully heated, you may want to remove all of the filament color left in the printer from previous prints before you start a new print. To do this, manually push filament through the extruder as you squeeze the extruder release lever until the new color starts coming out of the nozzle.
- g. Pro Tip: Never leave the printer preheated with material inside of it for long periods of time. The filament will bake inside of the extruder assembly and create a clogged hot end.

## 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 Setup > Preheat soft pull and wait until the nozzle temperature reaches 100°C.
- b. Wait for the nozzle to reach the set temperature.
- c. When the nozzle temperature is about 100°C, squeeze the extruder release lever and pull the filament out of the extruder.
- d. Cut the end of the filament to remove the irregular melted plastic at the end.
- 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.



## 5. Operation: The Four Steps of 3D Printing

3D printing involves these four steps:

1. Create a 3D file (Make)
2. Slice the file for 3D printing (Slice)
3. Send the file to the printer (Transfer)
4. Select the file on the printer (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

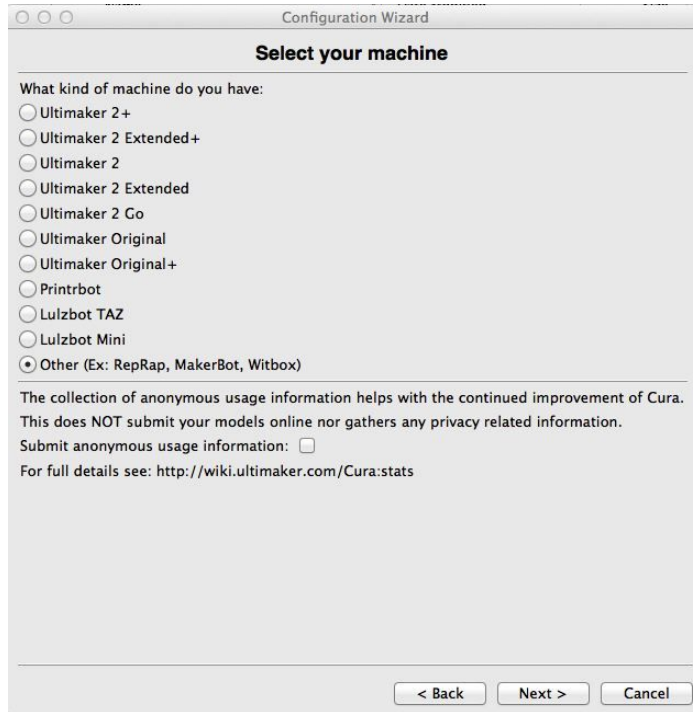
[Click here to learn how to setup and use Cura for the NWA3D A5](#)

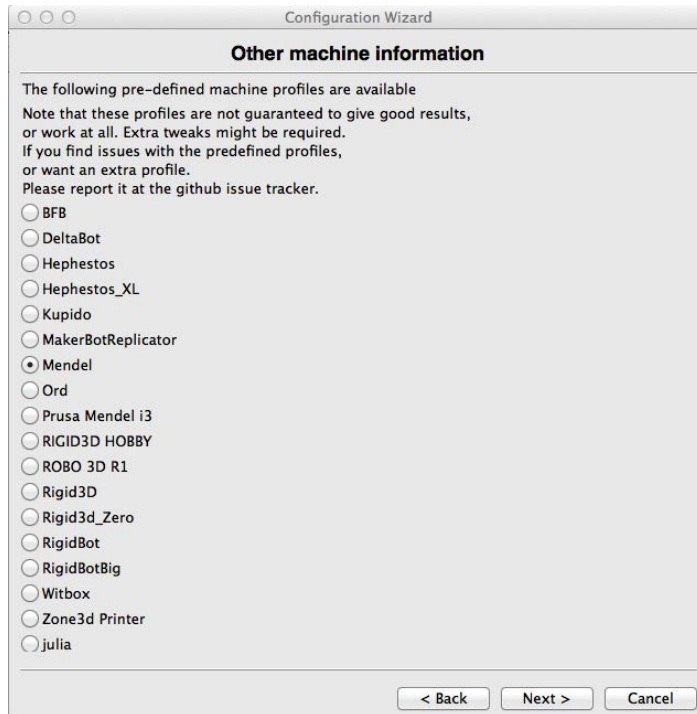
Slicing is the process of converting your 3D model into the layered print language your printer can understand, called G-Code. We recommend using free, open-source slicer programs. Our favorite slicers are [Cura](#) and [Repetier Host](#). Both are free and easy to use. We’ve included Cura and its setup .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 save the G-Code. It’s that easy!

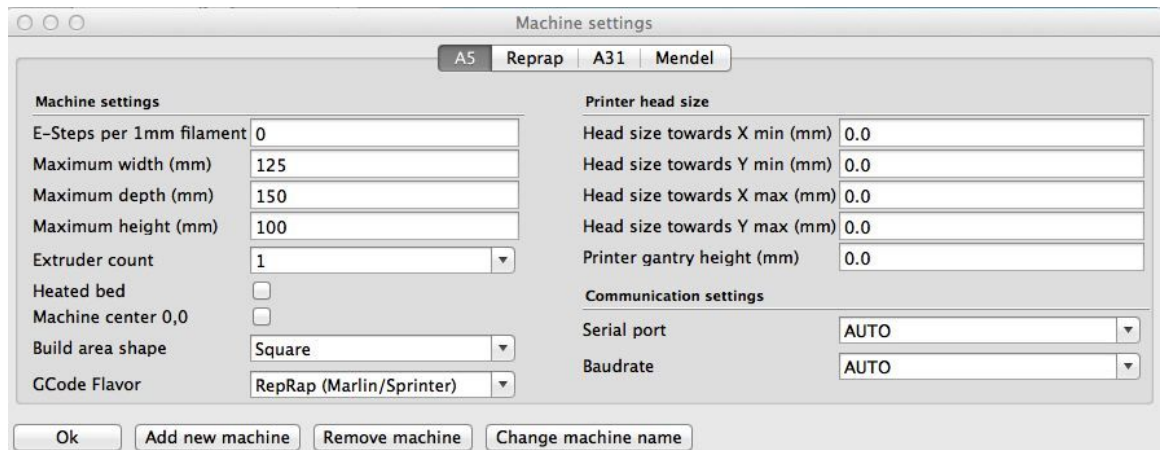


- a. Install Cura on a PC or Mac (both the .exe and .dmg versions are included on your microSD card).
- b. Open the Cura application. You'll be given several options. In the Add new machine wizard, choose "Other" in the "Select your machine" step, and then "Mendel" in the "Other machine information" step (RepRap (Marlin/Sprinter) G-Code Flavor, if it asks).

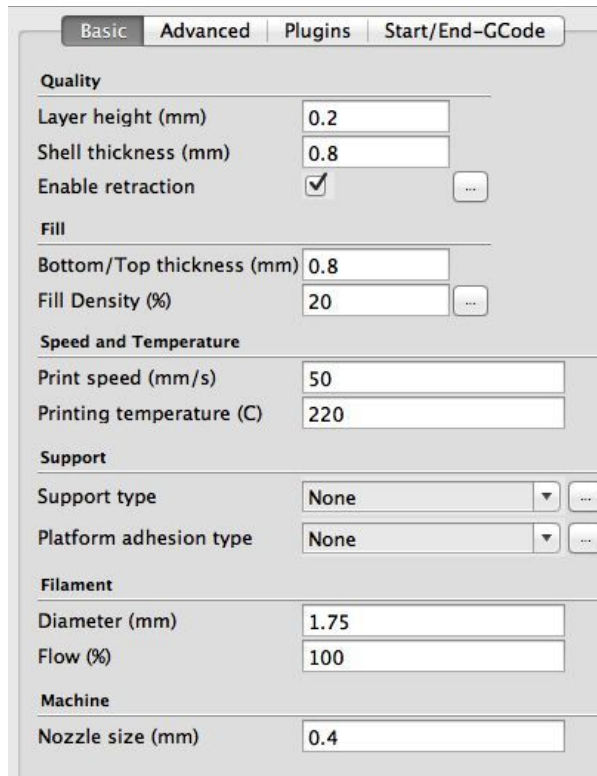




- c. Once you add the new machine, click Machine > Machine settings and change the machine settings to: Max width 125, Max depth 150, Max height 100, Extruder count 1, and Heated bed unchecked. You can also rename the machine to “A5” or “Our A5” if you’d like. Click “OK.”

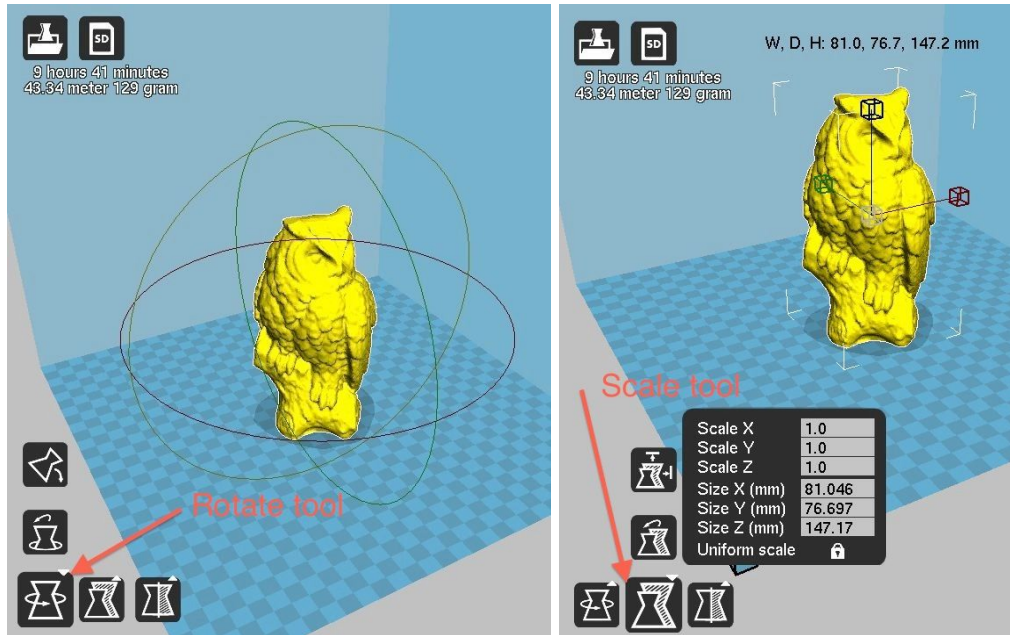


- d. Select File > Open Profile and select “NWA3D A5 Printer Profile.ini” from the microSD card. This is the medium-quality settings for the A5 printer.
- e. The “A5 Printer Profile.ini” settings in Cura for the A5 are shown on a screen capture on the microSD card. They are also shown in the following 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
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 by reloading the .ini file on your microSD card, or ask us for help by filling out a [Troubleshooting Request](#).
- 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. Remove the microSD card from the computer.
    - b. Insert the microSD card into the slot on the front of the A5 control board, underneath the knob.
    - 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 NWA3D A5 control screen, select Refresh 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. Mechanical problem #1: Leveling the build plate

[Click here to watch how to level the build plate](#)

- a. If the nozzle is digging into the build plate or is so close that no filament can extrude, you'll know the build plate is too close.
- b. If your print looks like spaghetti or the nozzle knocked the model off of the build plate, then you'll know the build plate is too far away.
- c. 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.

### 3. Mechanical problem #2: Filament is not coming out of the nozzle

[Click here to watch the three ways to unclog a nozzle](#)

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

- You can pull the clog out of the nozzle using the “soft pull” method.
- You can “floss” the clog out of the nozzle using a nozzle cleaner.
- You can heat the nozzle and push the clog through the 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 Setup > Preheat soft pull, 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 your 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, clip the melted end of the filament off, reload it, and select Setup > Cooldown and wait five minutes for the hot end to cool down. Then repeat the process.



**NOTE: Every time you change filament on your 3D printer, remove the filament with the nozzle at around 100°C. This “soft pull” 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:
- Remove the filament using the “soft pull” method.
  - Select Setup > Preheat PLA. Wait for the nozzle to reach the set temperature of 220°C for PLA.
  - Raise the Z axis carriage by selecting Controls > 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 220°C, use pliers to carefully insert the nozzle cleaner up into the tip of the nozzle. IT IS VERY HOT, SO BE CAREFUL NOT TO BURN YOURSELF!
  - Squeeze the extruder release lever with one hand and push the filament into the extruder manually.
  - Alternate “flossing” with the nozzle cleaner and pushing the filament through the extruder until a consistent bead of filament is extruded through the nozzle.
  - Remove the filament using the soft pull method.
- c. Try to push the clog through the nozzle:
- Select Setup > Preheat PLA. Wait for the nozzle to reach the set temperature of 220°C for PLA.
  - When the nozzle has reached 220°C, squeeze the extruder release lever and load the filament. When the filament would normally stop, keep pushing to manually push the filament out of the hot end nozzle of the extruder assembly.
  - Watch the filament to ensure it is not curling as it comes out of the nozzle. After pushing several inches of filament through, it should be going relatively straight. If it isn't, turn off the printer and do another soft pull (step a).



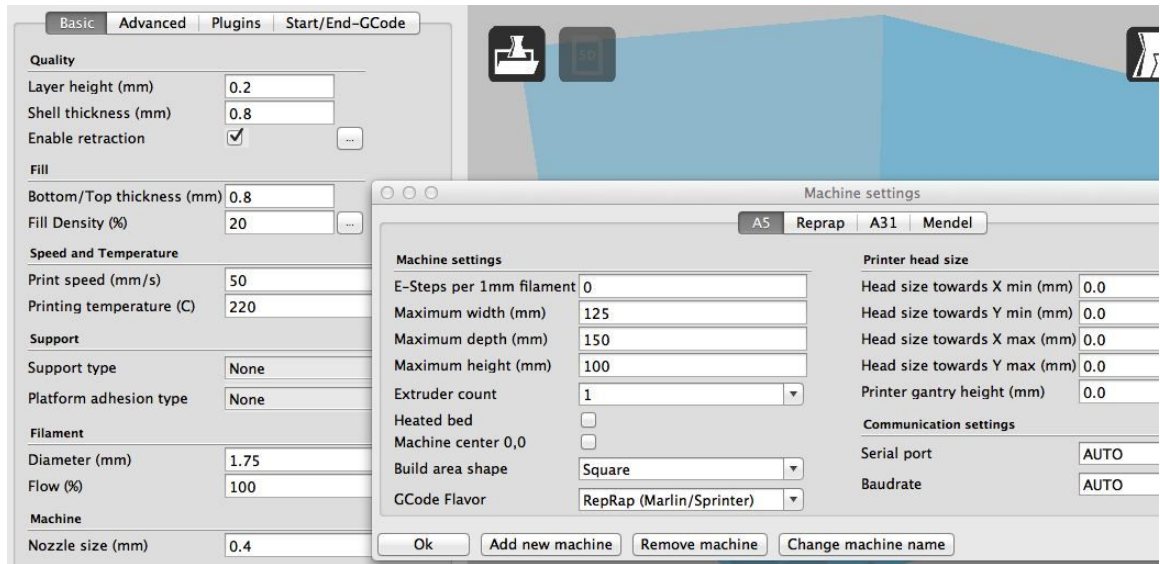
#### 4. Digital problems: Slicer settings

[Click here to learn how to set up Cura for the NWA3D A5](#)

Sometimes, models don't print correctly because the settings in the G-Code are wrong. Here are some steps to make sure your Cura settings are correct.

- 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 220–230°C.
- d. Make sure the filament diameter is 1.75mm and the flow rate is 100%.
- e. Make sure the Heated bed checkbox is unchecked in the Machine settings window (the NWA3D A5 does not have a heated bed).
- f. 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.
- g. 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.)
- h. 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 A5 Printer Profile.ini" that came on the microSD card in the Cura folder. Your settings should look like this:







## 7. Additional Resources

[nwa3d.com](http://nwa3d.com)

[NWA3D Troubleshooting Request](#)

[How to unclog a nozzle video](#)

[How to level the build plate video](#)

[How to set up Cura for the NWA3D A5 video](#)

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

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