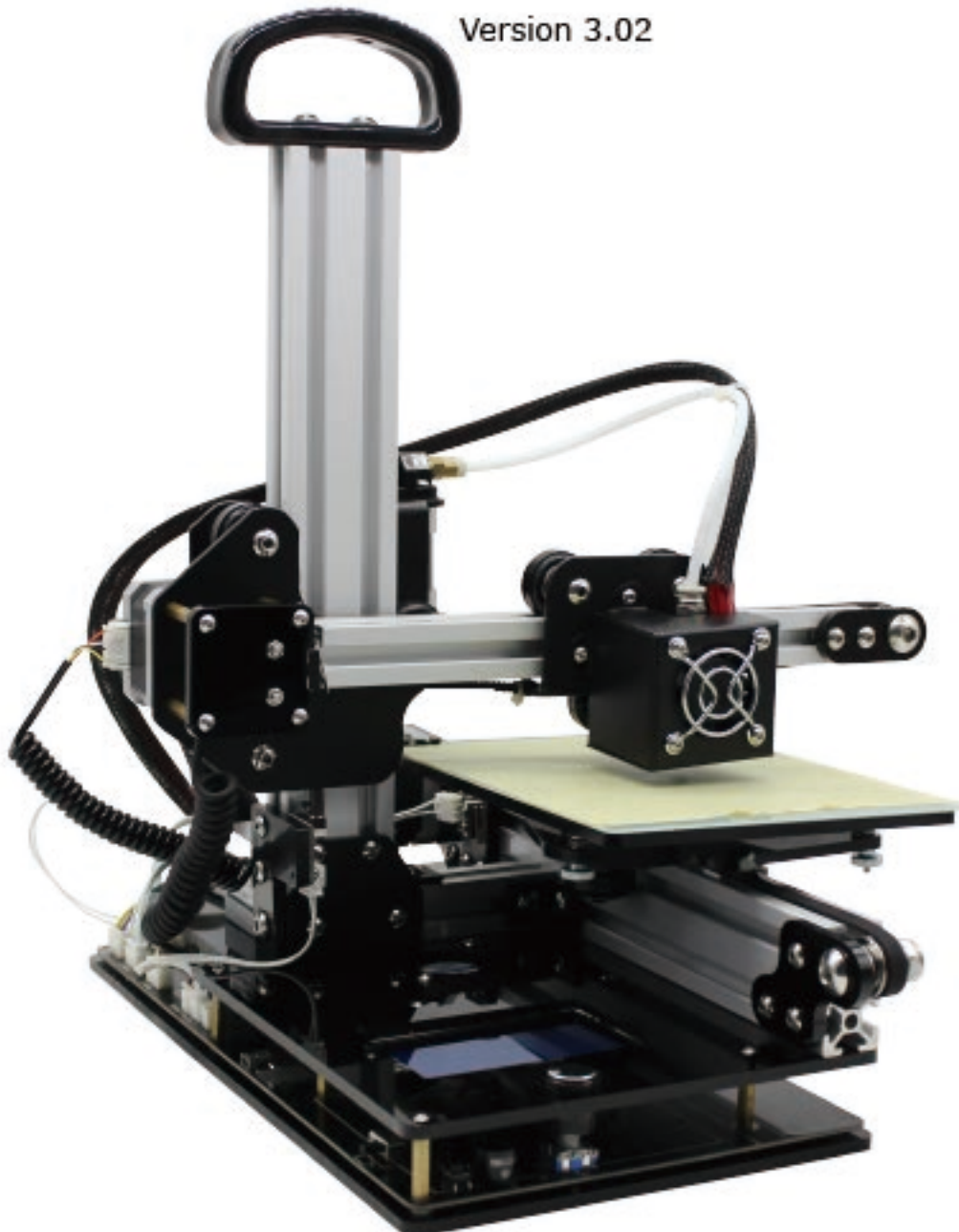


SKY3 WRITERD

SkyWriter 3D Desktop Mini

Quick Start Manual

Version 3.02



Please read all contents of this manual before operating your new 3D printer.



This printer is **NOT** a toy. Adult supervision is recommended for users under the age of 18. 3D printer parts and prints can be a choking hazard to small children; supervise at all times.



Keep hands and long hair away from the extruder, belts, gears, screw-drives and Power Supply at all times unless the printer is unplugged or until the machine has cooled and come to rest.



Electrical shock hazard: Do not operate in wet conditions. Do not attempt to disassemble the power supply or modify any electronic components.



Fire Hazard: Do not place Skywriter 3D Desktop Mini near flammable liquids or vapors under any circumstances.

If anything malfunctions, unplug the unit immediately!

If serious issues arise - contact our customer support, we will help you to resolve any issues.

Customer Support at:

<http://www.skywriter3d.com/contact/>

Step 1: Unpacking Your Skywalker Mini 3D.

Caution: Contents may have shifted during shipping.

- Open the top of the box and carefully remove the rectangular piece of protective foam.
- Remove all packing foam, as well as packing tape, and gently remove the printer and parts box.
- Save all packing materials.

Step 2: Download and Install Printing Software.

*Skywriter 3D printers will only print an object in **G-Code** format. Recommended software can be downloaded and has been included that will convert **.stl** files (a standard 3D model extension) to **G-Code** (a programming language that Skywalker 3D Desktop Mini and majority of 3D printers can process).*

In order to create an easy and tested sample print, or to do a custom print, we have provided the software **CURA** as well as 3D **.stl** files for you to start with.

CURA goes through many updates; the version pre-loaded onto your micro SD card is most likely outdated. We recommend downloading and installing the latest version of CURA for your Windows PC, Mac, or Linux based computer at:

<https://www.ultimaker.com/en/products/cura-software/list>

If an internet connection is unavailable and you are using a Windows based computer, you may install CURA from the Micro SD Card in the steps listed below:

First, locate the Micro SD card and pink Micro SD to USB adapter inside the parts box. Connect to your computer's USB Port.

Once the removable drive is recognized, navigate to and open the folder labeled "CR-7 English" and proceed to folder >> "Step 3 - USB Driver and Software installation" >> "CURA" >> "Software Package" >> "cura14.12.zip"

Double click or extract contents of "cura14.12.zip"

Proceed to open and install "Cura_14.12.exe"

Step 3: Configure the Machine Parameters for your printer.

1. Launch CURA. A Configuration Wizard will help guide you through the setup of this printer.

2. When CURA prompts for your machine, select "Other"

Other (Ex: RepRap, MakerBot, Witbox)

4. CURA will then prompt you to use a predefined profile, ***DO NOT*** select any from the list, simply click "Next"

5. Enter these parameters:

Custom RepRap information

RepRap machines can be vastly different, so here you can set your own settings.
Be sure to review the default profile before running it on your machine.
If you like a default profile for your machine added,
then make an issue on github.

You will have to manually install Marlin or Sprinter firmware.

Machine name	<input type="text" value="SkyWriter Mini"/>
Machine width X (mm)	<input type="text" value="130"/>
Machine depth Y (mm)	<input type="text" value="150"/>
Machine height Z (mm)	<input type="text" value="100"/>
Nozzle size (mm)	<input type="text" value="0.4"/>
Heated bed	<input type="checkbox"/>
Bed center is 0,0,0 (RoStock)	<input type="checkbox"/>

Note: These machine parameters will never need to be changed for this printer.

Step 4: Configure Print Settings:

Once the Machine Parameters are set, enter the Basic and Advanced print settings located on the left of CURA.

Note: These settings can be changed to suit the needs of different prints and are independent from the Machine Settings. These values are a good baseline for many simple prints with this printer. As you gain experience with 3D printing, you may find yourself changing individual settings often to produce better prints.

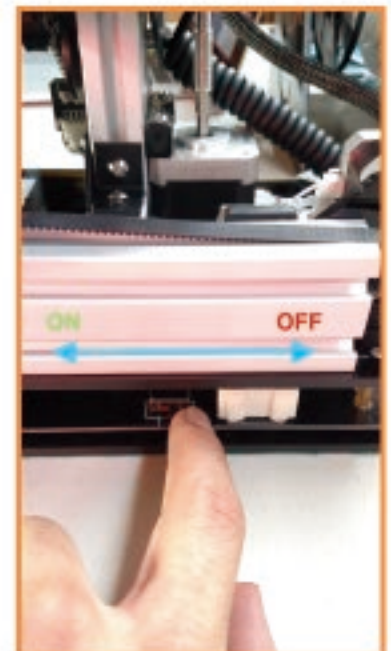
Basic	Advanced	Plugins	Start/End-GCode
Quality			
Layer height (mm)	0.2		
Shell thickness (mm)	1.2		
Enable retraction	<input checked="" type="checkbox"/>		...
Fill			
Bottom/Top thickness (mm)	1		
Fill Density (%)	20		...
Speed and Temperature			
Print speed (mm/s)	60		
Printing temperature (C)	210		
Support			
Support type	None		...
Platform adhesion type	Raft		...
Filament			
Diameter (mm)	1.75		
Flow (%)	100.0		
Machine			
Nozzle size (mm)	0.4		

Basic	Advanced	Plugins	Start/End-GCode
Retraction			
Speed (mm/s)	40.0		
Distance (mm)	4.5		
Quality			
Initial layer thickness (mm)	0.2		
Initial layer line width (%)	100		
Cut off object bottom (mm)	0.0		
Dual extrusion overlap (mm)	0.15		
Speed			
Travel speed (mm/s)	60		
Bottom layer speed (mm/s)	45		
Infill speed (mm/s)	0.0		
Top/bottom speed (mm/s)	45		
Outer shell speed (mm/s)	45		
Inner shell speed (mm/s)	45		
Cool			
Minimal layer time (sec)	5		
Enable cooling fan	<input checked="" type="checkbox"/>		...

You are now done with print settings!

Step 5: Prepare your printer.

1. Locate the power switch and ensure it is in the off position before connecting the power supply.

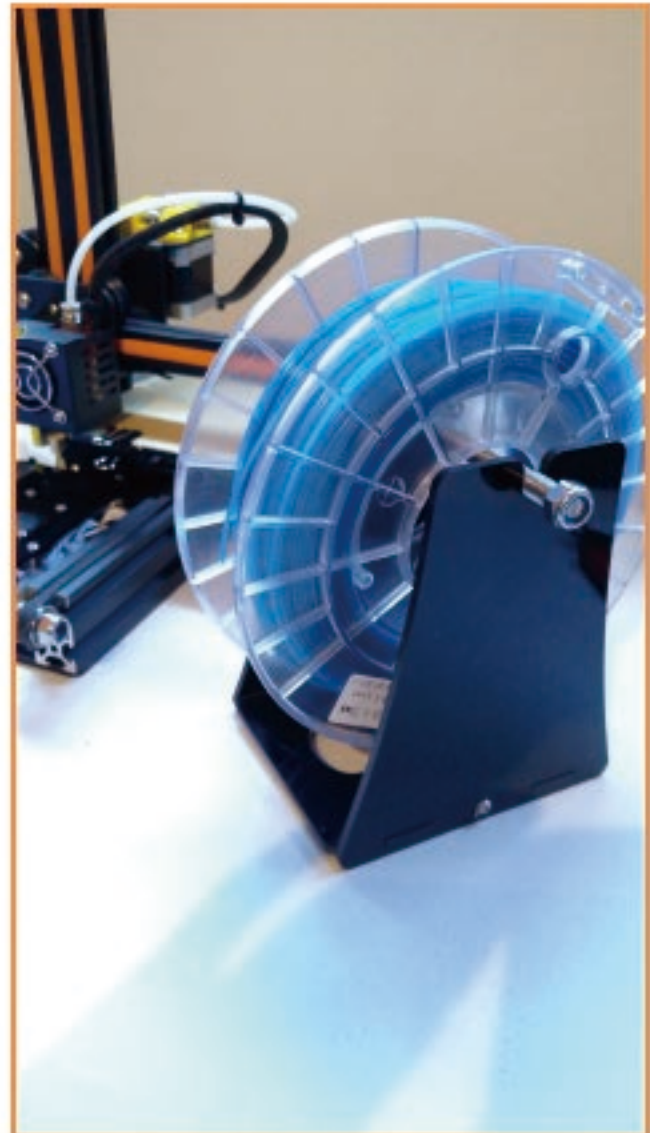
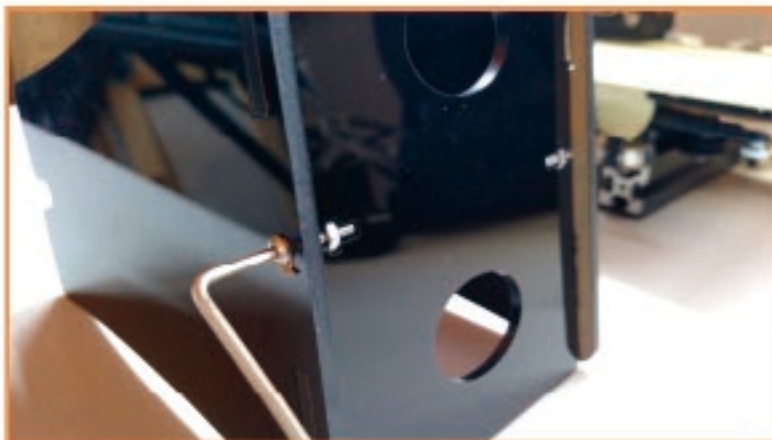


2. Connect the A/C Power Adapter to a wall outlet, then connect the Power Adapter to the printer.

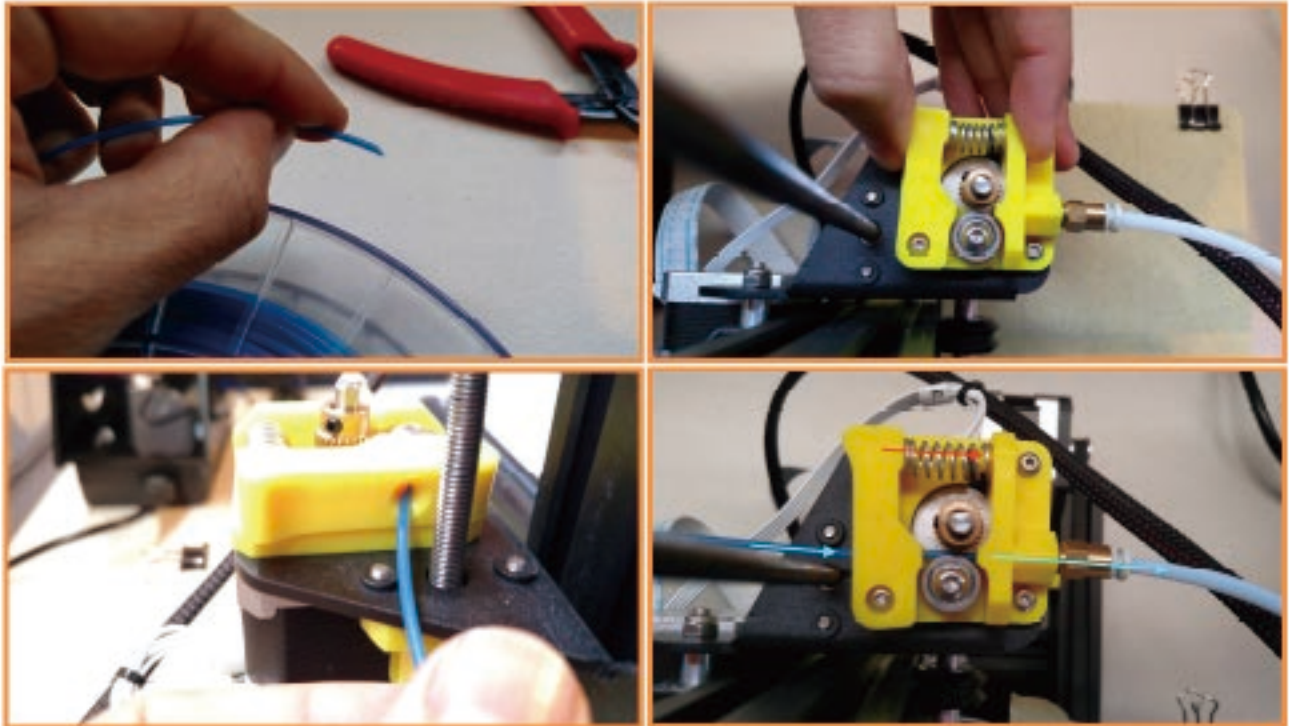
3. Slide the power switch to the on position.



4. Set up the Filament Spool Rack and hang the Spool inside the Rack.

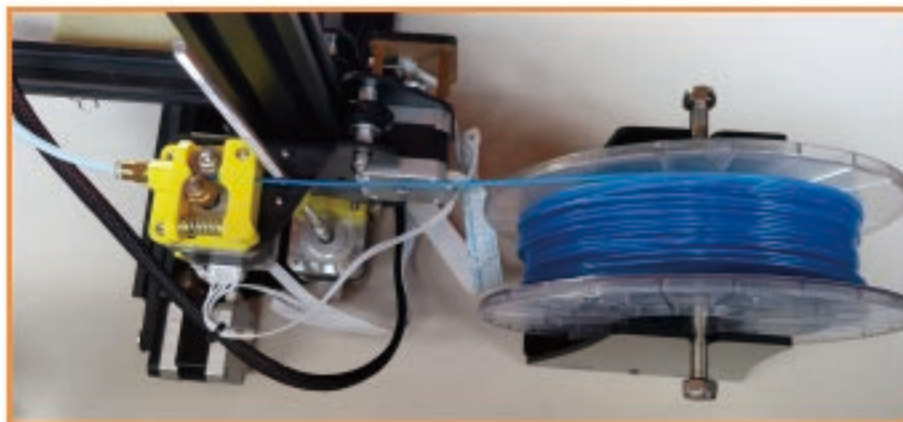


5. Take the leading end of the included Filament and cut a small segment off (a couple of centimeters) at a slight angle. Install the Filament into the Filament Feed by pinching the spring loaded lever and gently guiding the filament through as illustrated in the steps below.



As soon as you feel firm resistance, stop inserting the Filament and release the spring loaded lever.

Set the Filament Spool and Rack next to the printer.

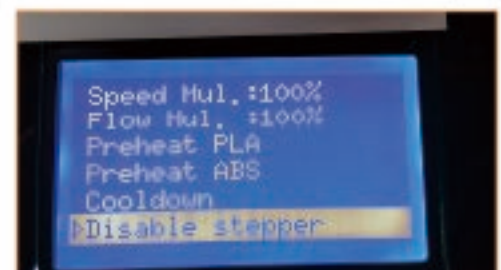
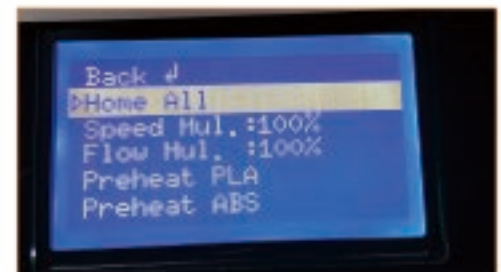
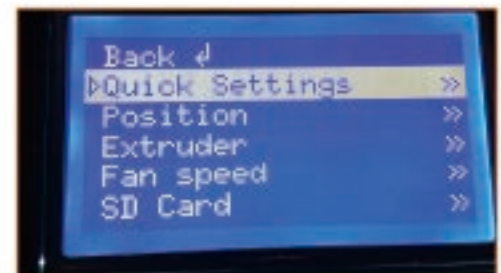


Step 6: Level your printer's bed for your first print.

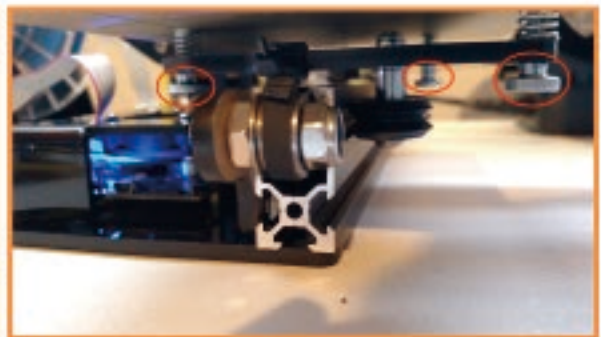
The goal of leveling the printing bed is to make sure that the extruder nozzle is at the same distance above the bed at any location. You do not want your extruder nozzle dragging or rubbing anywhere against your print bed. Conversely, you do not want the extruder head floating too far above the print bed as the Desktop Mini 3D printer needs a surface to print onto; it cannot print without support underneath it.

Please take your time during this process! After following these next steps, you will be rewarded with consistent prints that have great bed adhesion!

1. Take the taped printing plate off of the print bed by taking off the perimeter clips. Remember the configuration in which they were set; this print plate will be re-installed later.
2. With the 3D printer on, use the LCD Control Panel to navigate to the "Quick Settings" menu.
3. Select 'Home All'. This will always reset the printer into its default position.
4. Once the printer has come to a rest, select 'Disable Stepper'. This will allow the extruder and print bed to move in the X and Y directions respectively, without damaging the motors.

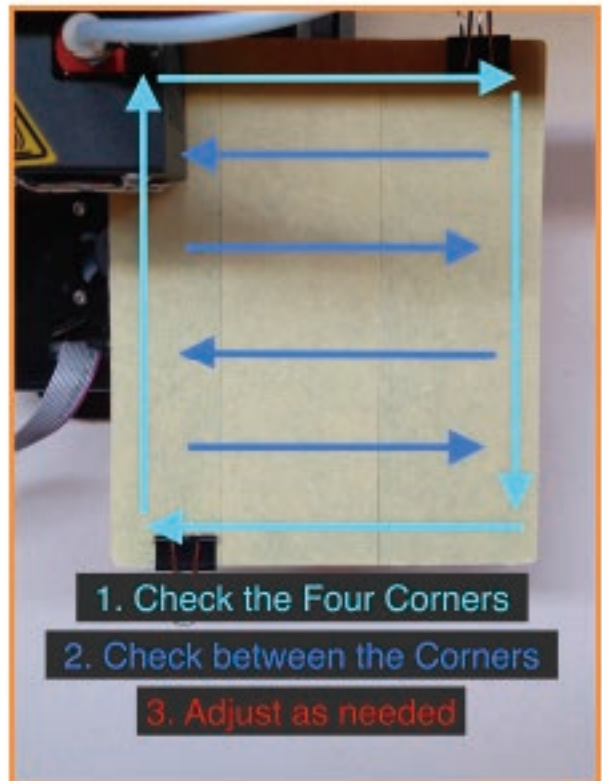


5. Gently re-install the taped printing plate with clips. **Ensure that the extruder nozzle is above the re-installed taped printing plate.** If it is not, lower the bed by adjusting the three silver thumb screws underneath the print bed in the clockwise direction until the tip of the nozzle is clearly above the taped plate. Once done, you are now ready to physically move the extruder and bed for a “Four Corners” adjustment.



6. Start the Four Corners adjustment by sliding the printing bed forward and backwards and ensuring the nozzle does not contact the taped printing plate at any time. Lower the bed at anywhere the nozzle contacts.

7. Next, bring the print bed all the way back and gently slide the extruder head side-to-side ensuring that it does not contact the taped printing plate at any point; lower the bed accordingly if it does. Move the bed plate forward a centimeter and repeat the process until you reach the other side of the bed.



8. Once this is done, raise the bed at every point such that the extruder nozzle is just low enough that a business card cannot slide between it and the bed. You want this distance to be consistent over the entire surface of the taped bed plate.

The printer is now set up for its first print!

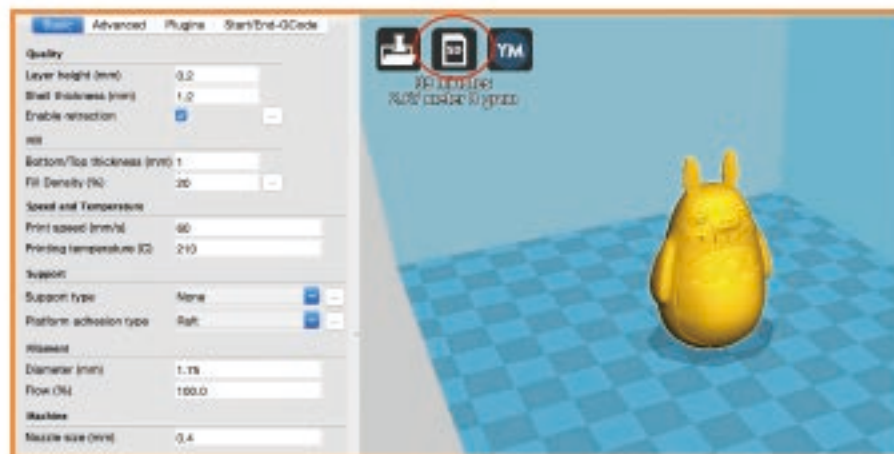
Step 7: Format your first print in CURA.

Open [CURA](#).

With the Micro SD Card and Pink USB to Micro SD Adapter connected to your computer, Navigate to: "Cr7-English">"step 5 Official print">"3 3D Models">"Small Figure">"totoro_solid.stl"

Double Click "totoro_solid.stl" and a figure will appear in CURA.

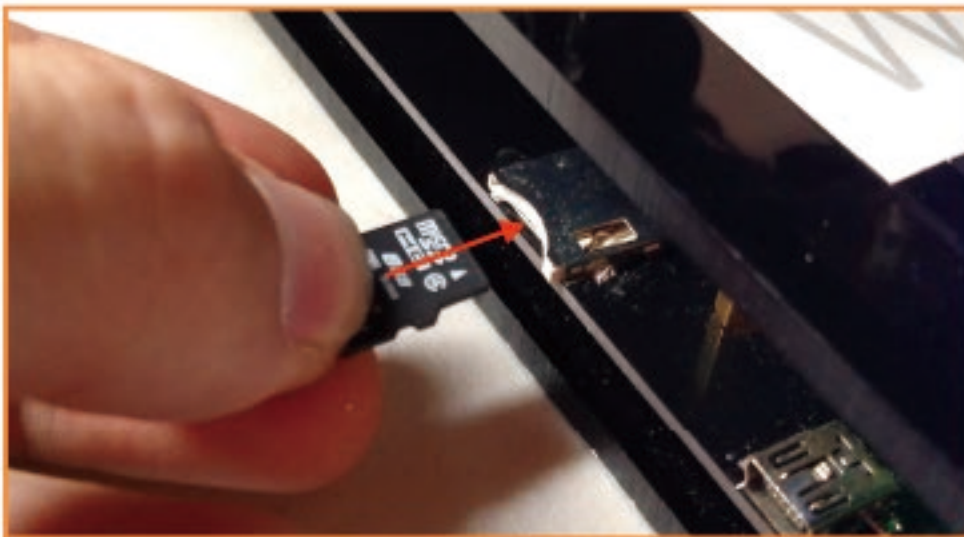
Now, save the figure by clicking "Toolpath to SD" in CURA, and a G-CODE file will be saved back onto the Micro SD Card.



After the file has been saved, safely eject the Micro USB Adapter and Micro SD Card by clicking eject at the bottom of the CURA 3D model space.



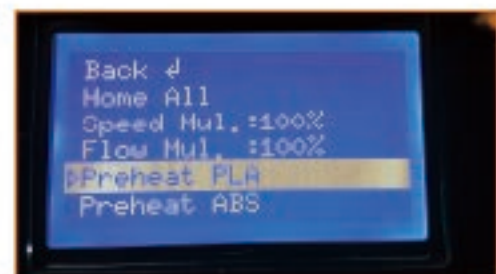
Take the Micro SD Card and insert it face up into the Skywriter 3D Desktop Mini.



Plug the A/C adapter into the 3D printer and power it up.

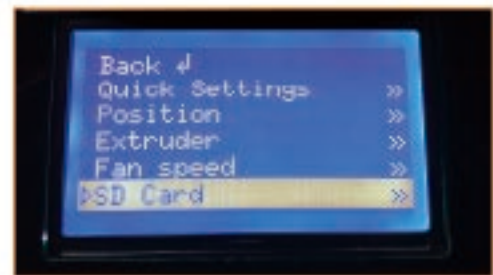
Use the LCD Control Panel to Preheat the Filament.

Go to Prepare and select "Preheat PLA".



Warning: Extruder will now heat, do not touch the extruder while it is heated.

Go to the main menu and select "SD Card".



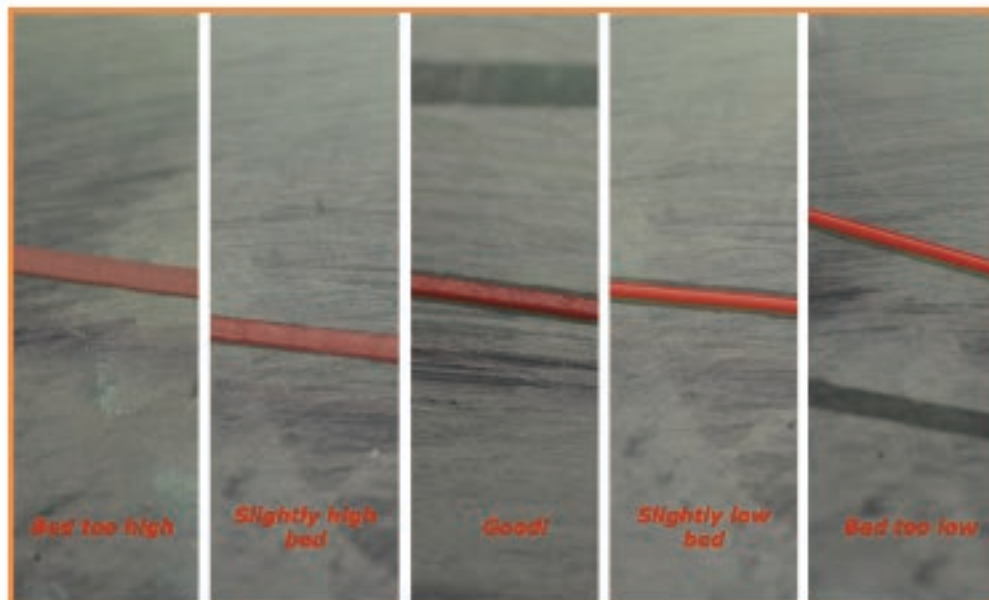
Select "totoro-solid.gcode".



Wait a few moments and your print should begin!

Note: It is important that you do not touch the printer while it is printing. You may still navigate the LCD Control Panel.

*Pay attention to Bed Adhesion as the print begins. **It may take a moment for the Filament to stick to the bed.** If it doesn't at all, the bed is most likely too high. Stop the printer, let it cool and repeat Step 6 as necessary.*



Once the print has finished and the printer has cooled and come to a rest, you may lift the figure off of the blue material Raft.

Congratulations on your first print!

