

CALIBRATION + OPERATIONAL MANUAL

version 1.1

Contents

| | |
|--------------------------|----|
| Before you Begin | 3 |
| Software Package | 4 |
| Squaring the Gantry | 5 |
| Moving the Axis | 6 |
| Leveling the Bed | 7 |
| Your First Print | 8 |
| Troubleshooting | 9 |
| Switching Filament | 14 |
| Unjamming a Hotend | 15 |
| Slicing Software Setting | 16 |
| Skeinforge Settings | 18 |

Before You Begin

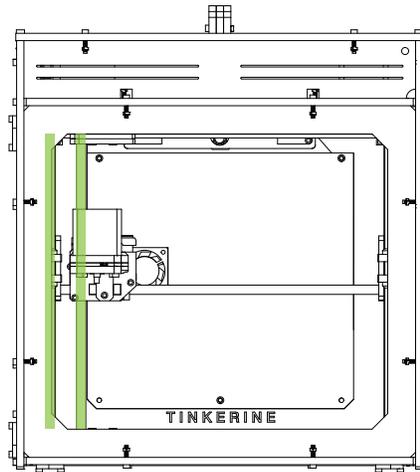
- ⓘ Before operation of the Ditto+/Litto 3D Printer, it is imperative that you read the calibration manual fully and ensure you understand every part of the instruction. While we provide warranty for all parts of the 3D printer, this does not cover damage caused by not following the instruction manual or other end-user error.
- ⓘ Before printing, you may wish to apply a layer of blue painter's tape over the print bed. Blue painter's tape can be found in most home hardware store and will allow PLA to stick onto the bed during the print. Re-apply a new layer of painter's tape after a couple of prints to ensure that the filament is firmly deposited onto the print bed.
- ⓘ During the calibration process, if you are unsure of any instructions or have any questions on how to proceed. Contact us at support@tinkerine.com and we will be able to assist you with any questions.

Software Package

-  Ditto+ and Litto 3D printers uses a customized slicing software to process your 3D models. Visit the Tinkerine Suite page at <http://www.tinkerine.com/tinkerine-suite/> to find the most updated version to download.
-  Tinkerine Suite installs via a executable installer. Download Tinkerine Suite and follow the step-by-step installation process. During this process, Tinkerine Suite will also install all require drivers needed in order for your computer to detect the printer.
-  Your printer's electronic comes preloaded with the newest firmware that is required to operate your 3D printer.
-  In the future, if you wish to update your firmware, you can do so under Tinkerine Suite by going to the Expert tab and selecting file to upload. You will need to connect your printer to the computer using the supplied USB cable.

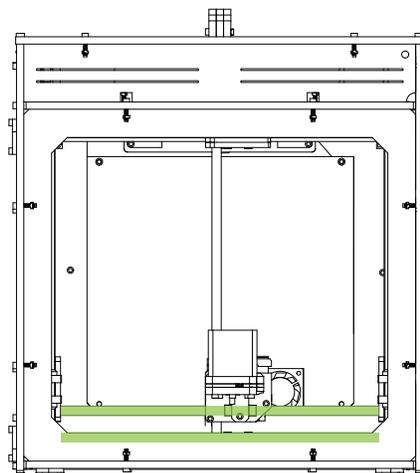
Squaring the Gantry

Y-Axis Crossrod parallel to Y-Axis Rod



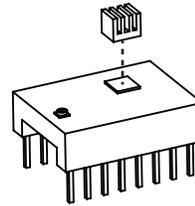
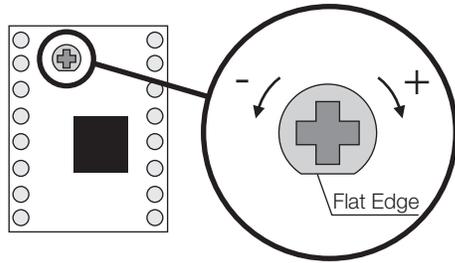
Squaring the gantry system ensures the rods are parallel to each other so the carriage can travel smoothly in both the X and Y axes. Start by centering the carriage on the Z-Platform. Move the carriage straight to the left until it reaches the edge. (You should hear a click from the X-endstop triggering. If not, adjust the position of the endstop until you do.) Measure from the front left corner, the distance between the outer rod and one of the inner rods that the carriage sits on. Do this for the rear left corner as well. Make sure the two measurements are the same. If they are not, adjust the alignment by manually pushing the sliders in opposite directions. Make sure to loosen the sets screws on the pulleys as you will not be able to adjust the position of the sliders if they are tightened. Tighten the set screws on the four pulleys when you're done.

X-Axis Crossrod parallel to X-Axis Rod



Now center the carriage again, then move it straight to the front until it reaches the edge. (You should hear a click from the Y-endstop triggering. If not, adjust the position of the endstop until you do.) Do the distance measurement for this side in the same way. Remember to tighten the pulleys when you're done.

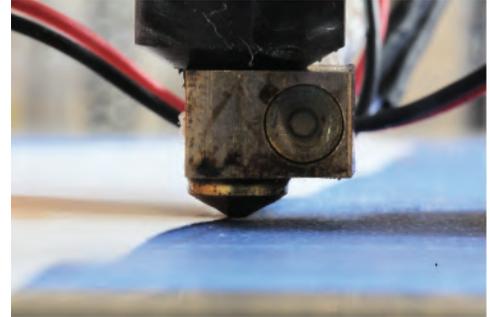
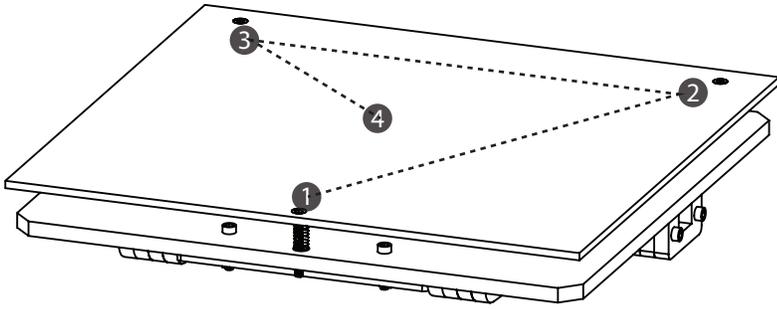
Moving the Axis



◀ Peel away the protective film on the bottom of the heatsink to reveal an adhesive layer. Attach the heatsink on the chip of the stepper driver as shown in the diagram.

- ⌚ To provide the motors with the proper amount of current, the A4988 stepper drivers need to be calibrated. Turning the dial on the driver clockwise increases the current to the corresponding motor (indicated on the RAMPS), and turning it counter-clockwise decreases the current.
- ⌚ The above diagram shows the off position where the current is zero. Notice the flat edge of the dial points to the 6 o'clock position. For starters, turn the dial clockwise until the flat edge points to the 7 o'clock position if you have white coloured stepper drivers, and 1 o'clock position if you have blue coloured stepper drivers.
- ⌚ Before you can start printing, you will need to perform a homing calibration on all three axis (X,Y,Z) of the printer. The endstops on each axis determine the home position of the bed and hotend, so it is very important that they are correctly set to ensure quality prints.

Leveling the Bed



Proper leveling: Roughly the thickness of 1 pieces of paper.



Move the carriage to each of the four points and check the bed level. Adjust the three points by tightening or loosening the nut.

- t** Before starting the first print with your 3D Printer, it is important that the Z-Platform is properly leveled. Start by manual turning the z-axis lead screw so the bed moves upward toward the hotend, bring it just close enough so that the carriage can move around without scrapping anything on the bed. Move the extruder/carriage to the front center edge, level that position by adjusting the thumb screw underneath the spring. Tightening the thumb screw lowers the platform, and loosening it raises the platform. You should aim for a small gap of roughly the thickness of a piece of paper between the platform and the extruder nozzle.
- t** Once the front center position is leveled, move the carriage to the rear right and rear left corner and level it the same way. After all three corners have been leveled, bring the carriage to the centre of the platform to do a final check; it should be at the proper level already. If it is not, check the three corners again to ensure you have leveled them correctly.
- t** After the Z-Platform is completely leveled, turn the Endstop Trigger Adjustment Knob (M4x20 screw) until it just triggers the Z-axis endstop (You should hear a click). Using Pronterface, home all axis, then check if the platform remains leveled as before. If not, adjust the Endstop Trigger Adjustment screw until the proper level has been achieved. Your calibration goal for the Z-Platform is to stop the print head just above the print surface so that you can slide a sheet of paper through with slight resistance. If the bed homes earlier than expect, turn the knob clockwise to bring it down. If the bed homes too late and your hear the motor skipping, switch off the printer to stop the motor and bring the adjustment knob up by turning it counter-clockwise.

Troubleshooting

| Issues | Description | Solution |
|---------------------------------------|---|--|
| (Operation) Outdated Arduino Firmware | I want to update the firmware for on my printer. | <p>Visit the Tinkerine website or email contact@tinkerine.com to obtain the latest version of the firmware for your printer model.</p> <p>Once you have the proper firmware file. Use Tinkerine Suite to upload the file onto the printer.</p> |
| (Operation) Motors are Locked | Motor are locked, and cannot be moved. | <p>Nema motors are automatically engaged when the printer is connected. Using the LCD controller, disengage the motor by navigating to the Control sub-menu.</p> |
| (Operation) Cannot Extrude Filament | Filament is not coming out when I manually purge the hot end. | <p>A. Are you trying to extrude with an unheated hot end? The software prevents cold extrusion to prevent it from breaking the extruder. Make sure your hot end is heated up to the proper temperature (around 215C) before your purge your hot end.</p> <p>B. Is the hob properly tensioned to the filament? If the hob is free spinning and not biting onto the filament, no filament will be pushed out. Remove the filament, loosen the extruder motor and re-align the motor + hob so that it can grip the filament properly.</p> |

Troubleshooting

| Issues | Description | Solution |
|--|---|--|
| (Operation) Motors Making Clicking Noises / High Pitch Frequency | I'm hearing a high pitched noise/ clicking noise from my motors. | The potentiometer on your Pololu drivers are turned on too high! Giving your motors too much power will overheat them cause them to operate noisier than needed. With a ceramic screw, adjust the potentiometer (Counter clock-wise for lower, clock-wise for higher voltage). Adjust it so that the motor does not emit any foreign noises while giving the motor enough current to move the axis or extruder without skipping. |
| (Calibration) Backlash on prints | Loss of motion in one or more axis, causing prints to become displace/mis-aligned. | <ol style="list-style-type: none"> 1. Check tension on all GT2 belt, and re-tension any belt you feel that is too loose, causing the loss in motion. 2. Check and tighten all set screws inside of pulley. This includes the 8 pulleys on the smooth rod, and 2 pulleys attached to the NEMA motors. 3. Check to see if motors are getting enough power. Adjust the potentiometer on the Pololu driving accordingly if motors are under voltage. |
| (Calibration) Skipping | Interrupted movement of a motor, causing either an uneven flow of filament or temporary skip in the axis. Skipping can also be caused by an interrupted movement from other motion components found on the printer. | <ol style="list-style-type: none"> 1. Determine the source of the skip. <ol style="list-style-type: none"> A. Extruder Motor – Check to see if the filament is properly seated against the hobbed bolt, if the bolt is too far away, the hobbed bolt will not latch onto the filament properly and slip. You may also need to remove the extruder motor to see if the hob is clean and not clogged with filament fragments. B. Belt – Check to see if the belt is too tight and restricting fluid movement of the axis or catching on another gantry component and causing friction. C. Pulley – Inspect the pulleys to see if they are all correctly position and aligned with the belt that is attached. Inspect all pulleys for visible damage that causes improper rotation. CONT. |

Troubleshooting

| Issues | Description | Solution |
|--------------------------------|---|---|
| | | <p>D. Sliders – Inspect all sliders along the axis and check to see if the delrin are moving smoothly. Also make sure the linear bearing on the carriage are operating smoothly as well.</p> <p>E. Axis Motor – If belt and pulleys are working smoothly, your axis motor that is skipping may not be receiving enough current from the respective driver. Using the ceramic screwdriver that is included within your electronics kit, turn the potentiometer on the Pololu clockwise. Increase the voltage ever so slightly each time, as giving an motor too much current will overheat it.</p> <p>F. Electric Interruption – During the point of the skip, was the printer connected to a power source that had a high power device turned on? A temporary flux to the power supply of the electronics devices on the printer may cause a skip on the print.</p> |
| (Calibration) Dragged Filament | Hotend's nozzle is digging/dragging down previous layers. | <ol style="list-style-type: none"> 1. Bed level is too high. Readjust the Z-Axis endstop trigger screw, bringing the bed down away from the hot end nozzle. 2. Check that your filament diameter within the slicing software is dialed in correctly. Incorrect filament diameter will cause the extruder to over/under extrude filament per layer. |

Troubleshooting

| Issues | Description | Solution |
|-----------------------------------|--|--|
| (Calibration) Curling and Warping | Corner of the print are not sticking to the bed and is curling upward. | <p>Curling and warping of print are usually caused by plastic cooling at different rates. There are some ways to prevent or minimize this issue.</p> <ol style="list-style-type: none"> 1. Print with a clean bed – Is it time to clean or swap out your print surface? A dirty print surface or old piece of tape will cause the filament to not stick to the bed. 2. Z-Height – Is your Z-axis properly calibrated? If the bed is too low, the extruded filament will not be able to make a good contact with the print surface. Bring your bed height up and make sure the first layer is deposited firmly onto the surface. 3. Ambient Temperature – Are you printing outside in the cold? Or is your printer sitting beside a window with a cold draft coming in? A cold ambient temperature will increase cooling rate of the plastic, causing to curl or warp drastically. Move your printer somewhere a bit warmer. |

Troubleshooting

| Issues | Description | Solution |
|---------------------------------|---|---|
| (Calibration) Print Adhesion | Filament is not sticking to the bed properly. | <p>1. Adhesion Material – Have you applied something on the print bed for the print to adhere to? Try Blue painter's tape, Kapton, or other unique solutions that you may come across.</p> <p>2. Z-Height – If your print bed is too far away from the nozzle, the filament will not adhere properly to the bed and curl off. Calibrate and bring the bed closer to the nozzle, you generally want the first layer of the print to be a bit 'squished' to make sure the print sticks firmly.</p> |
| (Calibration) Inconsistent Flow | Inconsistent flow of filament coming out of the hotend, prints have skipping like signs and there are breaks and blobbing on the filament deposition. | <p>1. Extruder Motor – Check to see if the filament is properly seated against the hobbed bolt, if the bolt is too far away, the hobbed bolt will not latch onto the filament properly and slip. You may also need to remove the extruder motor to see if the hob is clean and not clogged with filament fragments.</p> <p>2. Check that your filament diameter within the slicing software is dialed in correctly. Incorrect filament diameter will cause the extruder to over/under extrude filament per layer.</p> |

Switching Filament

- t** Perform this procedure when you are switching out a roll of filament. Make sure you read through and understand the instructions before you begin.

- 1** Remove filament guide tube from the top of the extruder.
- 2** With the LCD controller, pre-heat the hotend.
- 3** With the hotend heated, Push down on the filament to extrude a bit of filament before pulling the filament out of the extruder. Trim away the molten tip of the filament.
- 4** Insert the new filament spool on the spool holder and thread the filament through the filament guide, then into the extruder.
- 5** Manually thread the filament through the top of the extruder, past the hob, and into the hotend. If the filament appears to be stuck, bring it back out and trim the end of the filament on an angle and try again.
- 6** The filament should be within the hotend now, you should be able to manually extruder some filament by hand and purge the previous filament out.

Slicing Software Settings

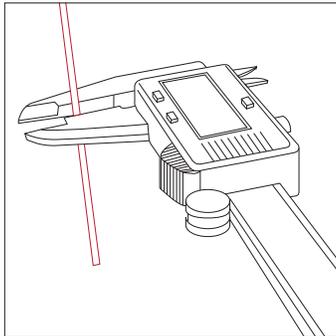
T The process generally refers to as “slicing” is when a slicer software takes a 3D drawing file (most often in .STL format), and translates it into individual layers. The software will then take this information and generate the codes the 3D printer will use to run along a predefined route to deposit filament.

Tinkerine Suite - Our latest and recommended slicing software. Tinkerin Suite offers easy-to-use interface with an array of powerful features.

Coordia: Vanilla version of our slicing software. Coordia offers a USB tethered support for printing without a LCD/SD card controller.

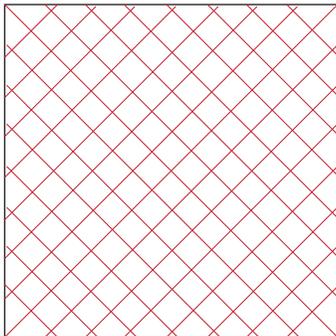
You can read more and download both software at:
<http://www.tinkerine.com/tinkerine-suite/>

Filament Diameter



While PLA/ABS filament come labeled as 3mm or 1.75mm, it is quite rare for them to have a perfect diameter or 3.0 or 1.75. It is highly recommended that you use a digital caliper to measure the diameter of your filament. Measuring the filament and adjusting the diameter setting will allow your printer to extrude/retract the correct amount of filament and provide you with a better quality print. You will generally want to take measurement from 2-3 spots on the filament and entering the average diameter into the slicing software. Note that you should adjust the diameter setting every time you switch to a different spool of filament.

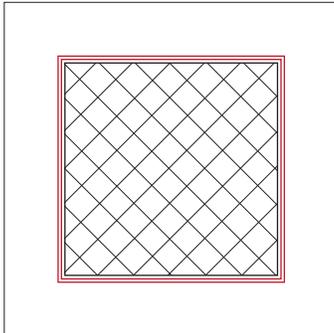
Infill %



Depending on the purpose of your printed object, you will want to change the infill density. By default, the Infill is set to 10% density. So if you are printing a hollow object (0%) to something that is structural dense (50%), adjust the Infill % to fit your need.

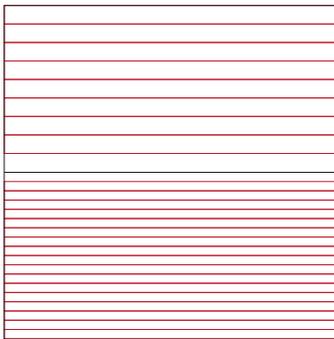
Slicing Software Settings

Perimeter



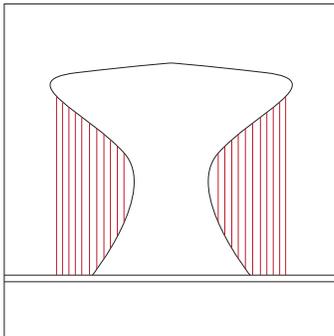
Changing the perimeter setting will affect the thickness of the outer shell. This is especially useful when you are printing a hollow object with no infill, so by adding thickness to the outer shell, you are maintaining the structural integrity of the print.

Layer Height: Resolution



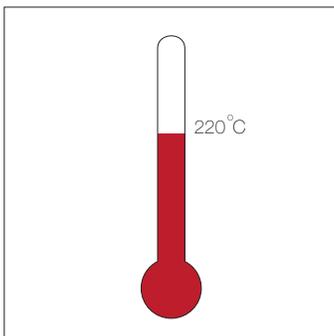
Layer Height, or also referred to as Resolution, determine the detail of the printed object, while the software is defaulted at 0.2mm (200 microns), you can adjust within your slicer setting to print at 0.1mm (100 microns) per layer. While the resolution double in quality, the print time also increases exponentially.

Support



Once you enable it, support material is added by the slicer software to help prints with extreme overhang come out smoothly without drooping. While this may sound like a good function to always have on, users have to beware of the added print time and additional use of material. It is generally a good idea to leave support turned off until the object you are print has overhangs that are over 60 degrees.

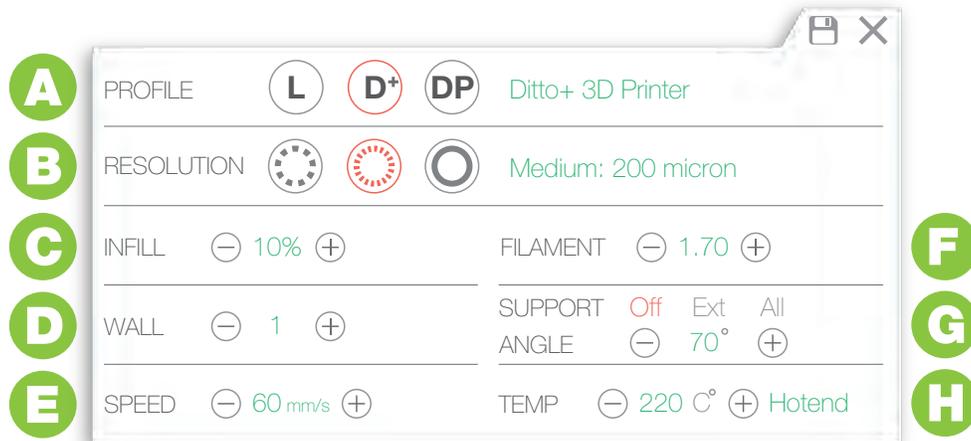
Temperature



While both PLA and ABS plastic have a specific melting point, depending on the source and method of production, the perfect extrusion temperature might vary from different brands of filament. The filament that is provided with your Ditto+ and Litto printers is tested to print between 220-230 degrees celsius, if you are getting issues while using a different brand of filament, you may wish to tweak the temperature.

Recommended Slice Settings

Whenever you play around with the settings within Tinkerine Suite and want to default back to the original settings, check below!



- A** PROFILE: Set this to the printer you are printing with. Ditto+ / Litto / DittoPro
- B** RESOLUTION: Default resolution is set at 200 micron (0.2mm per layer).
- C** INFILL: Infill density is set to 10% by default.
- D** FILAMENT: Set this to the measured filament diameter. Measure with a digital caliper for best result.
- E** WALL: Number of perimeter/wall on the printed object.
- F** SUPPORT: Off by default, activate to add printed support structure on object.
- G** SPEED: Default printing speed 60-70mm/s
- H** TEMPERATURE: For use with Tinkerine's 1.75 PLA filament, set temperature to 220C.