Orion Delta™ 3D Printer Manual
Second Edition – Firmware 0.91 and Higher
V3.00, February 23rd, 2015
MatterControl v1.2.2

This guide will take you through the steps to set up and operate your new 3D printer from SeeMeCNC. You'll find instructions on calibration, software, maintenance and more all in this manual. As a new SeeMeCNC™ owner, you'll also find a ton of great resources on the forums at forum.seemecnc.com

READ ME FIRST!

READ THIS MANUAL COMPLETELY BEFORE UNPACKING AND POWERING UP YOUR PRINTER!

Hazards and Warnings

The SeeMeCNC Orion Delta 3D printer has motorized and heated parts. When the printer is in operation always be aware of possible hazards.

Electric Shock Hazard

Never open the electronics bay of the printer while the printer is powered on. Before removing the access door, always power down the printer and unplug the AC line cord.

Burn Hazard

Never touch the extruder nozzle or heater block without first turning off the hot end and allowing it to completely cool down. The hot end can take up to twenty minutes to completely cool. Also, never touch recently extruded plastic. The plastic can stick to your skin and cause burns. Also before of the heated bed which can reach high temperatures capable of causing burns.

Fire Hazard

Never place flammable materials or liquids on or near the printer when powered on or in operation. Liquid acetone and vapors are extremely flammable.

Pinch Hazard

When the printer is in operation, take care to never put your fingers in the moving parts, including the belts, pulleys or gears. Also, tie back long hair or clothing that can get caught in the moving parts of the printer.

Static Charge

Make sure to ground yourself before touching the printer, especially the electronics. Electrostatic charges can damage electronic components. To ground yourself, touch a grounded source.

Age Warning

For users under the age of 18, adult supervision is recommended. Beware of choking hazards around small children.

Table of Contents

Attention! If your Orion Delta™ has a serial number of 100169 or less, you may need to download the First Edition of the user guide here:

http://www.geneb.org/orion/ORIONUserManual.pdf

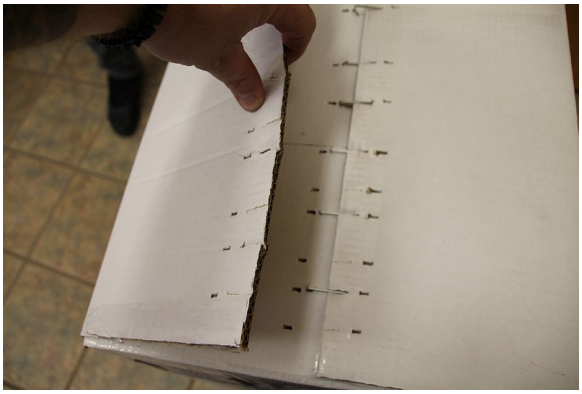
The First Edition covers the older firmware that originally shipped with the Orion Delta™ 3D printer.

Before downloading the First Edition, please make sure that you actually need it. Compare the LCD idle screen (shown on power-on, after the initialization sequence takes place) on your machine to the first LCD screen photo shown in the “The LCD Control Panel” chapter. If they match, you've got the newer firmware and do not need the second edition. You can follow this edition to reach the power-on point in order to definitively identify which firmware your machine was shipped with.

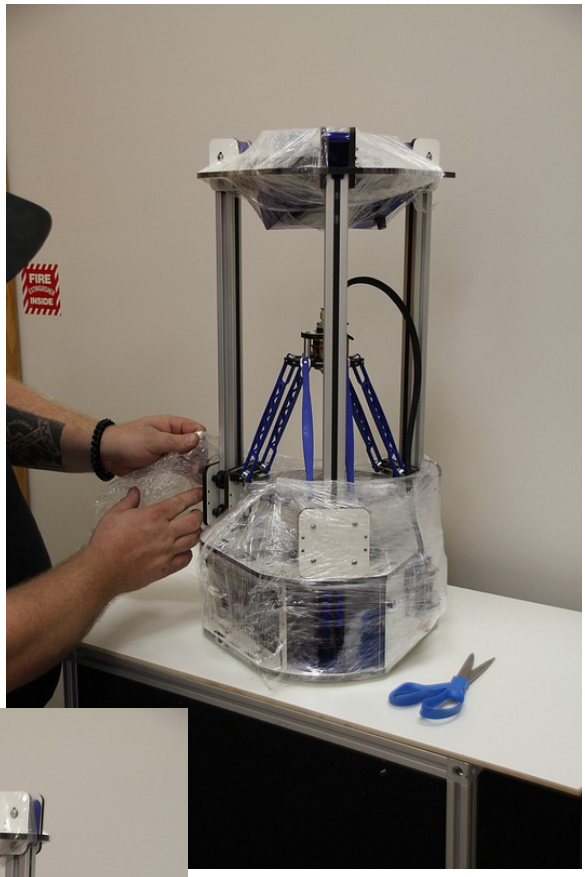
Un-Boxing your new Orion Delta™ 3D Printer

With your box standing in the upright position, carefully cut the packing tape along the top edges and across the taped seam in the box.

After you've cut the tape, open the top of the box being careful of any packaging staples. You'll find the machine tucked inside and wrapped in a protective expanding foam shell. Remove the machine and foam all as one by pulling straight up out of the box. Be careful not to drop the machine once it's out of the box and the foam is still around it.

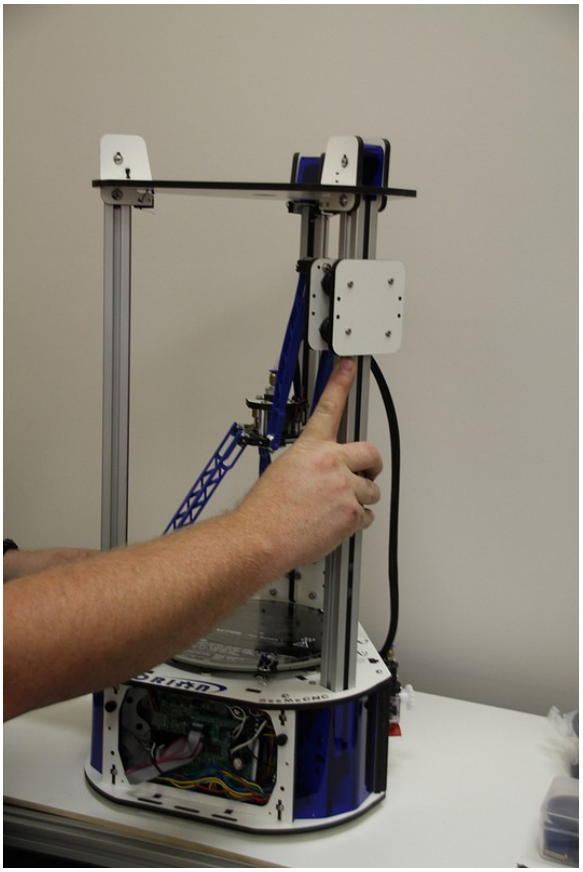


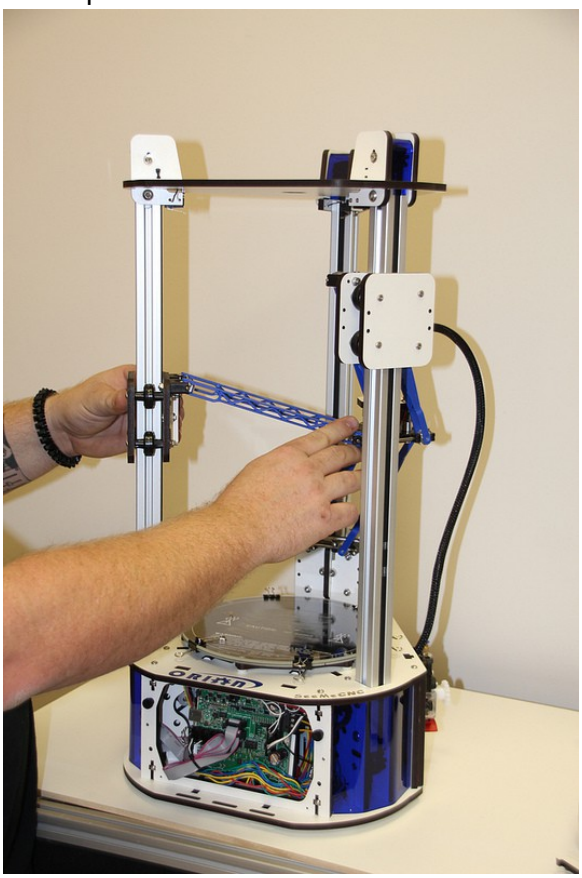
Carefully cut and remove the stretch wrap film that is holding the power/USB cables, Orion control panel and accessories on the top plate, as well as the stretch wrap holding your filament to the table top and glass build surface.

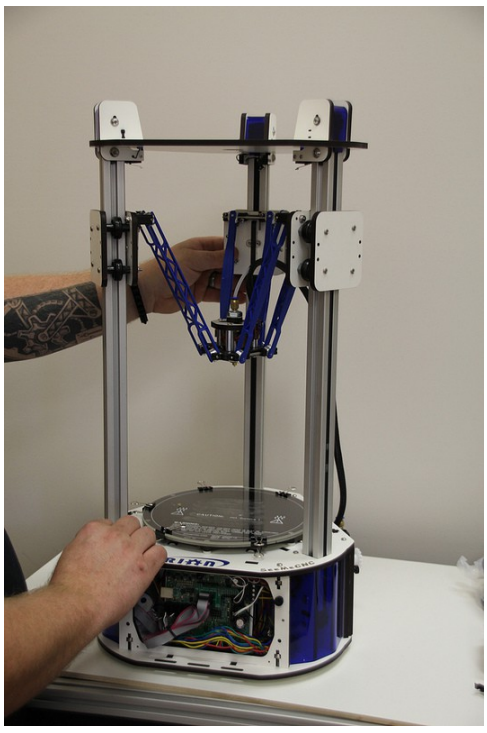


In order to prevent damage during shipping, the hot end on the Orion Delta™ is locked in place over the bed by lowering the delta arms as low as they'll go on each tower. These arms need to be raised in order for the Orion Delta™ to be used.

The process is very simple – just grasp the Cheapskate bearing for each arm and raise them one at a time to the top of the Orion as shown in the photos below.



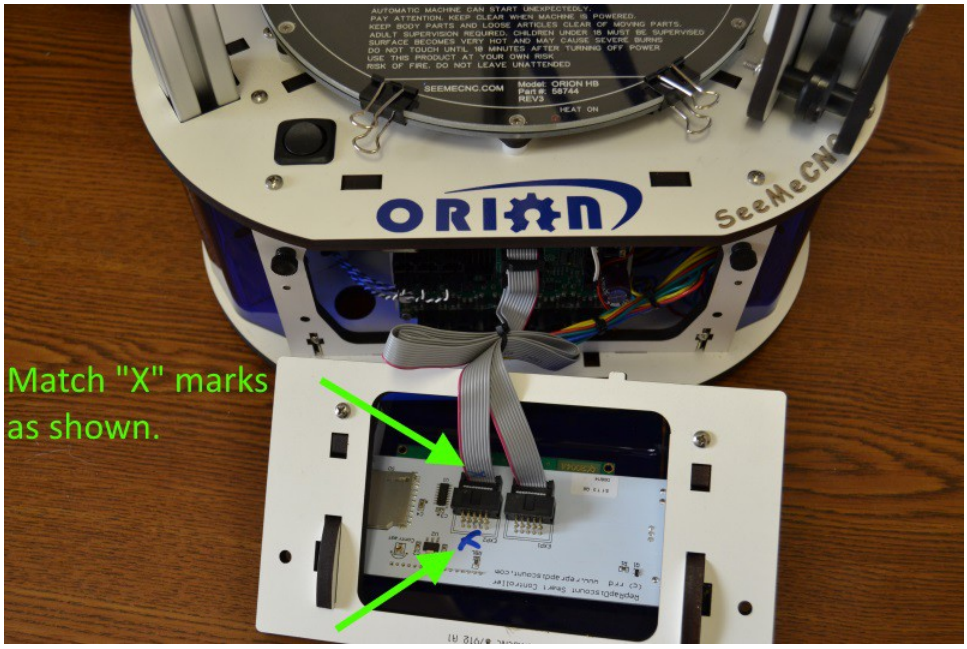


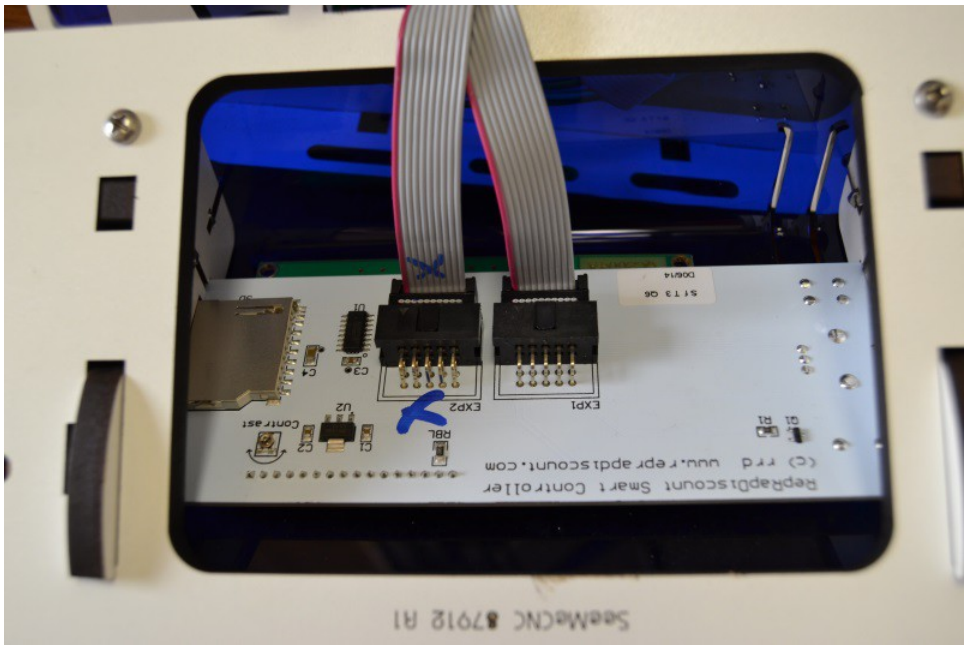


Installing the LCD Control Panel, USB Cable and SD Card

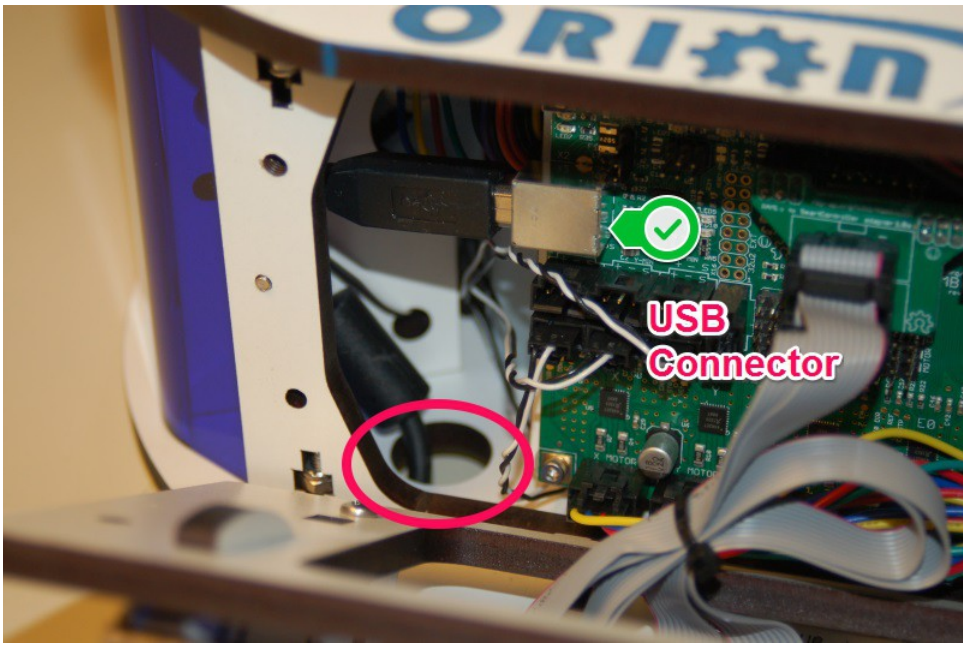
Note that the blue acrylic face plate on your Orion Delta™ will likely differ from the photos shown here, but the installation instructions are identical.

Gently pull the pair of flat ribbon cables out a bit from the Orion Delta™ and install them as shown. The “first” cable is marked by an “X” and has a matching “X” on the back of the LCD controller.





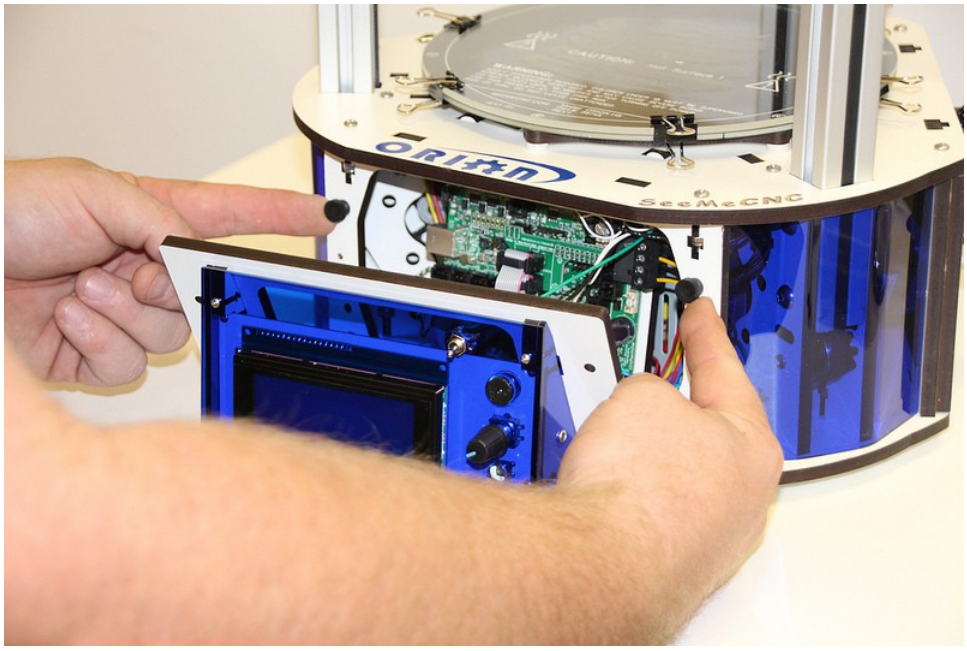
The USB cable only needs to be used if you wish to manually control the machine from the software on your PC. You can print and do most calibration standalone, without the USB cable attached. We recommend hooking it up now, so if you need to connect it to your computer to make changes etc., the cable is already installed.

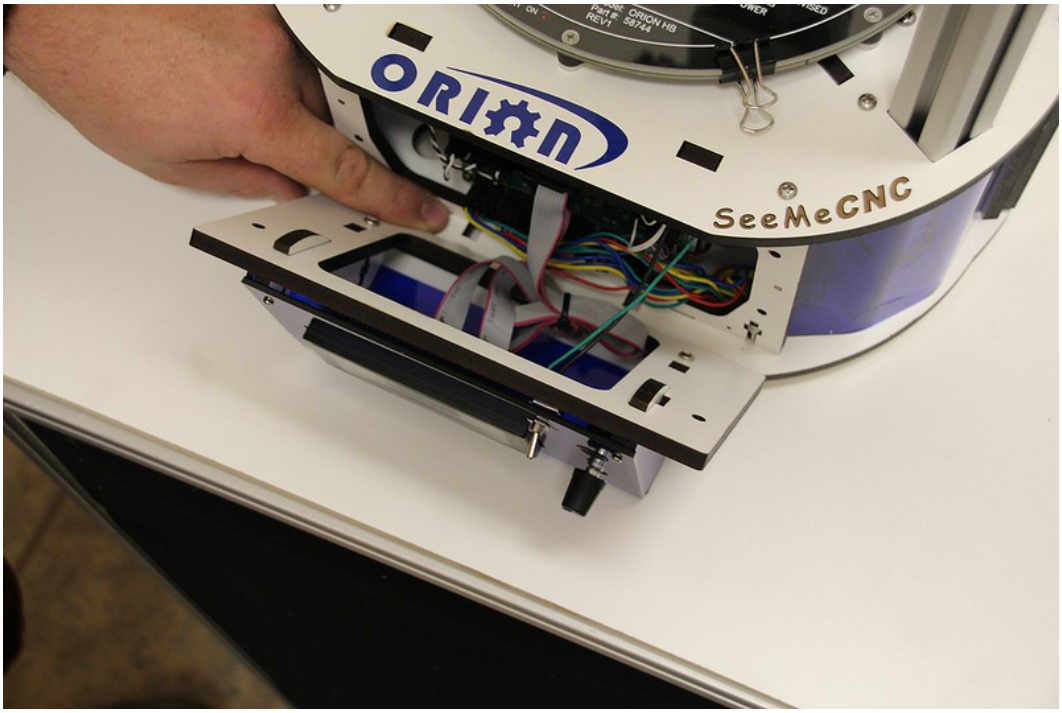


Pass the end of the USB cable up through the hole in the bottom left of the base and plug it into the USB input on the electronics board.

Next, replace the front panel by putting the bottom in first, then tilting the top in. Please be careful as it's a tight fit. Re-install the two black thumb screws and tighten them finger tight.

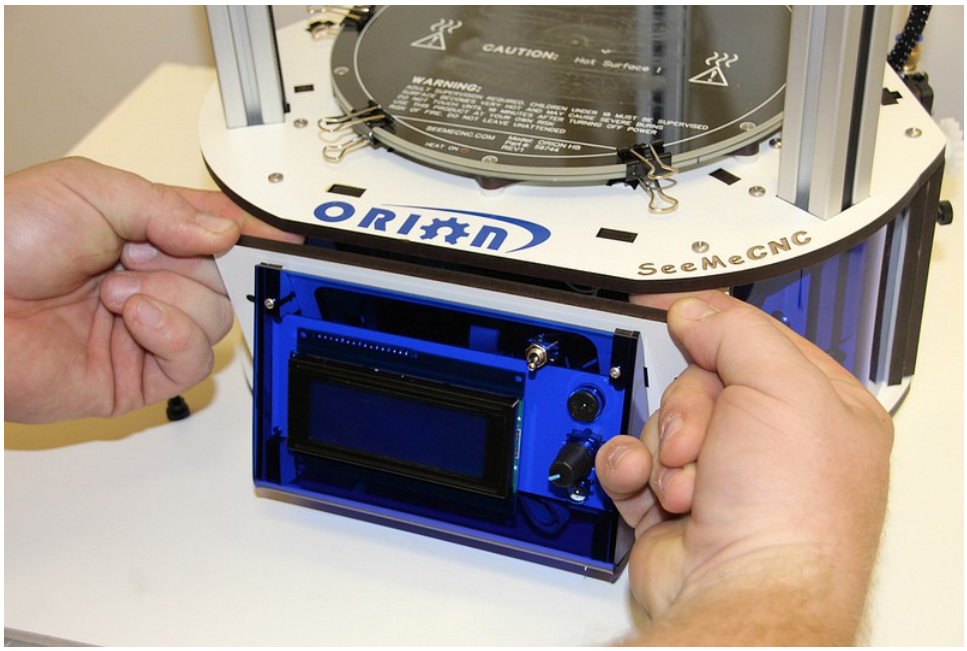
Remove the two black thumb screws as shown and set them aside.



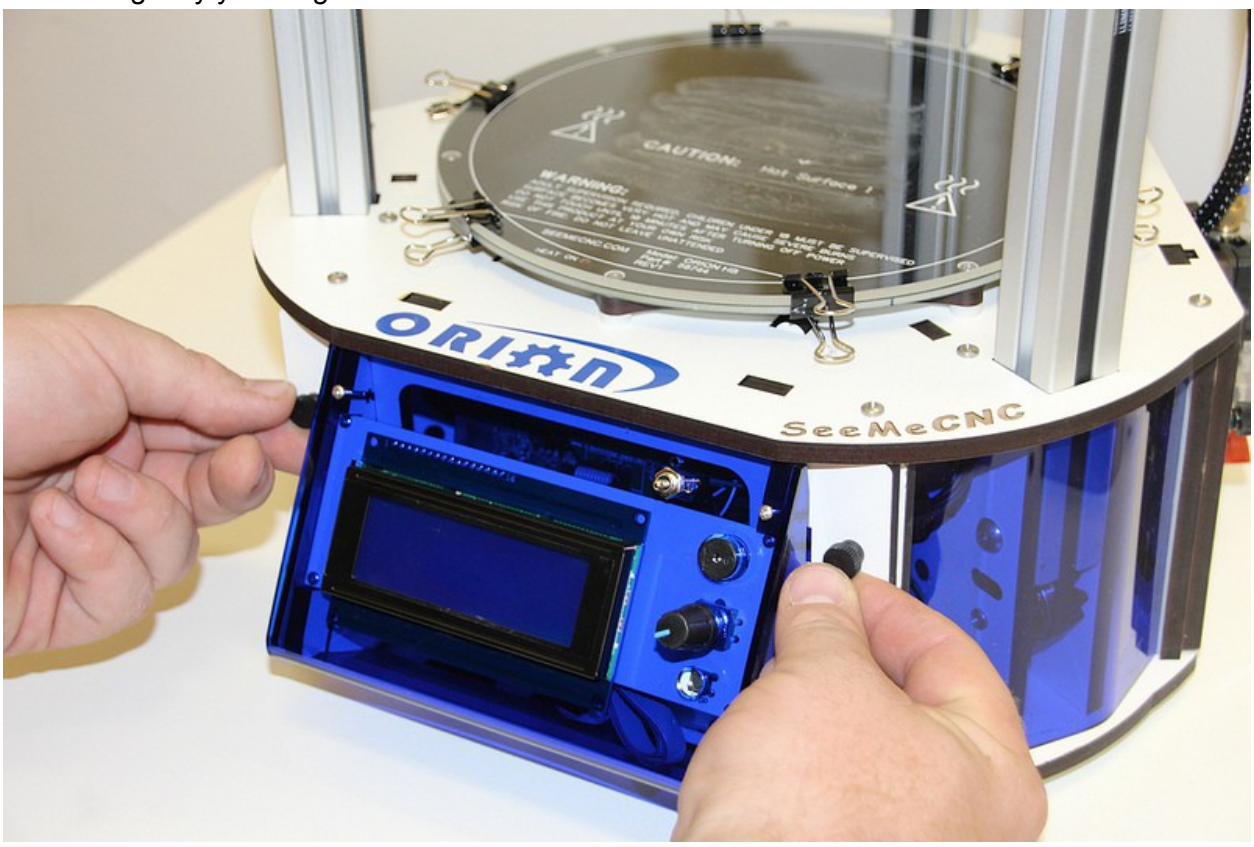


Now you'll install the LCD control panel into the Orion Delta™. Please take care as the parts are a tight fit. Align the LCD mounting plate as shown below – there's small locking tabs that fit into the slots shown.

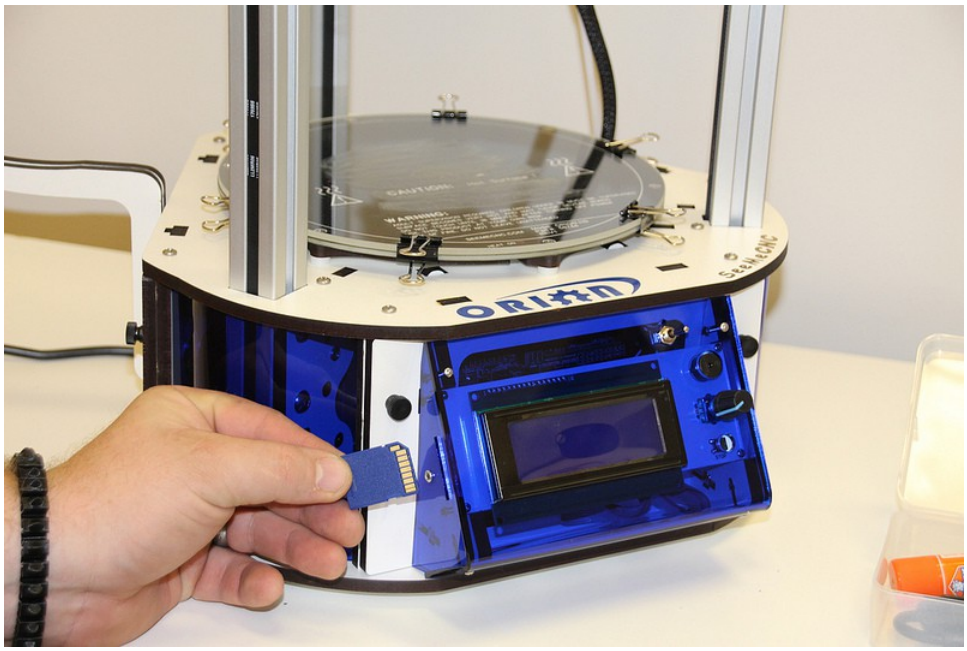
Carefully raise the panel into place and fit it flush against the mating surface, making sure that not wires get pinched between the two panels – this is a tight fit, so please be careful.

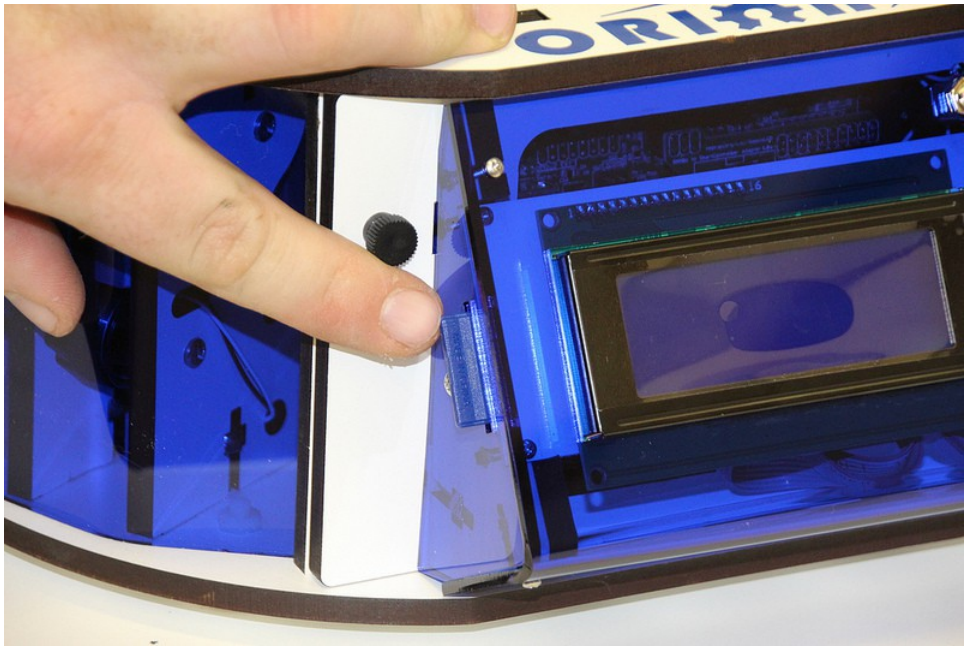


Insert the black thumbscrews you'd removed earlier in the mounting holes and tighten them using only your fingers.



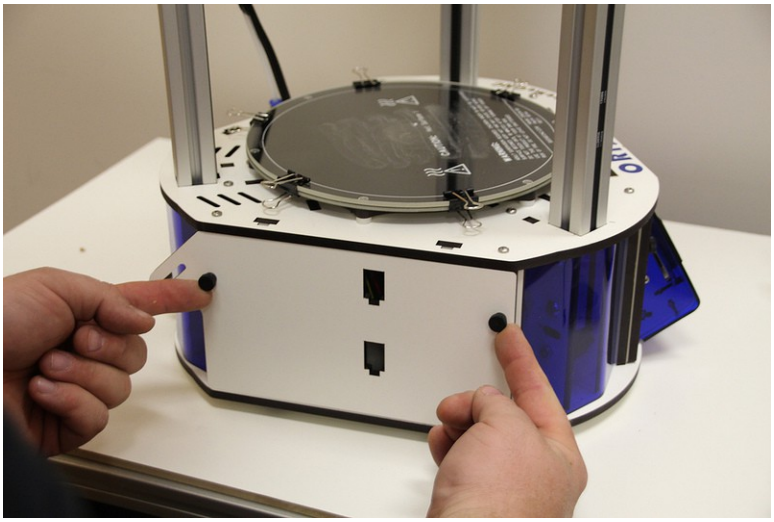
In the accessories box, you should find a small SD card. Insert the SD card into the side of the LCD enclosure as shown below.



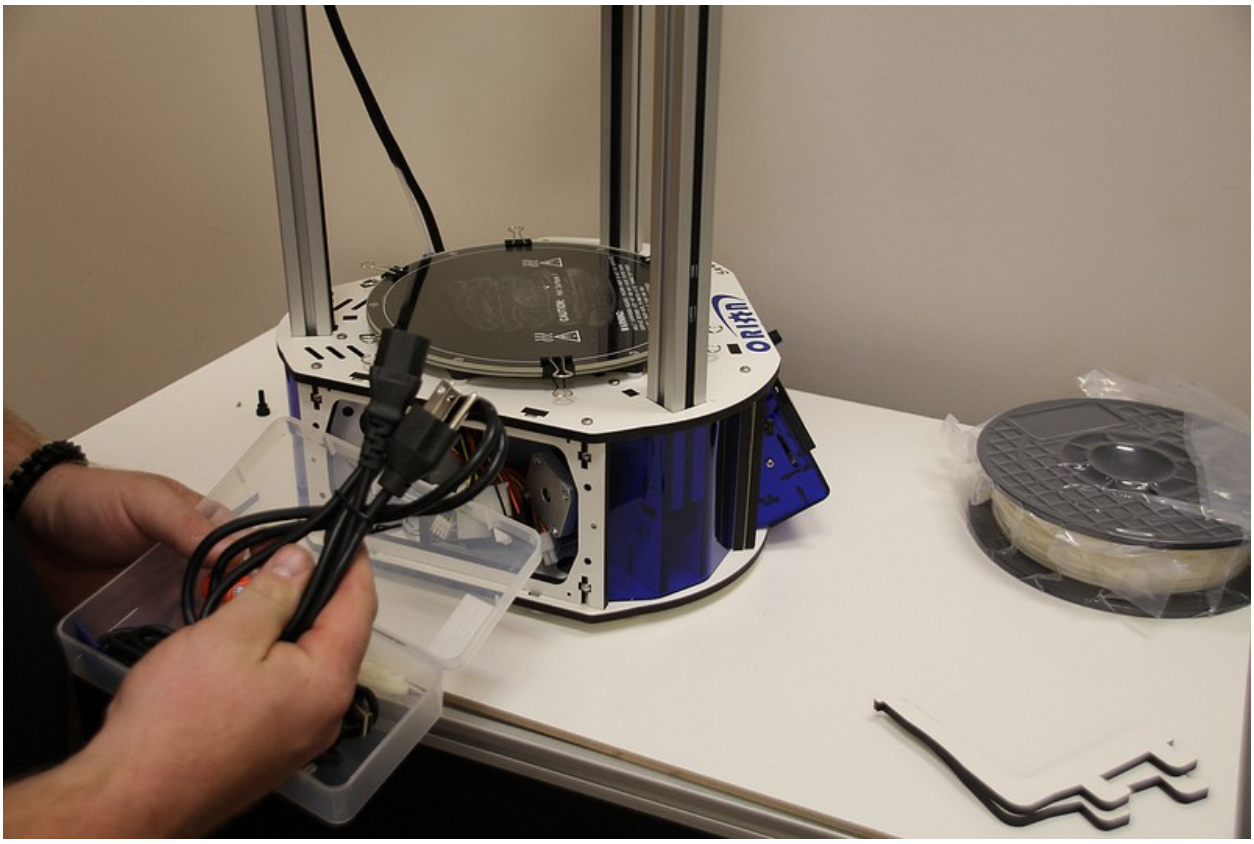


Installing the Power Cord and Spool Holder

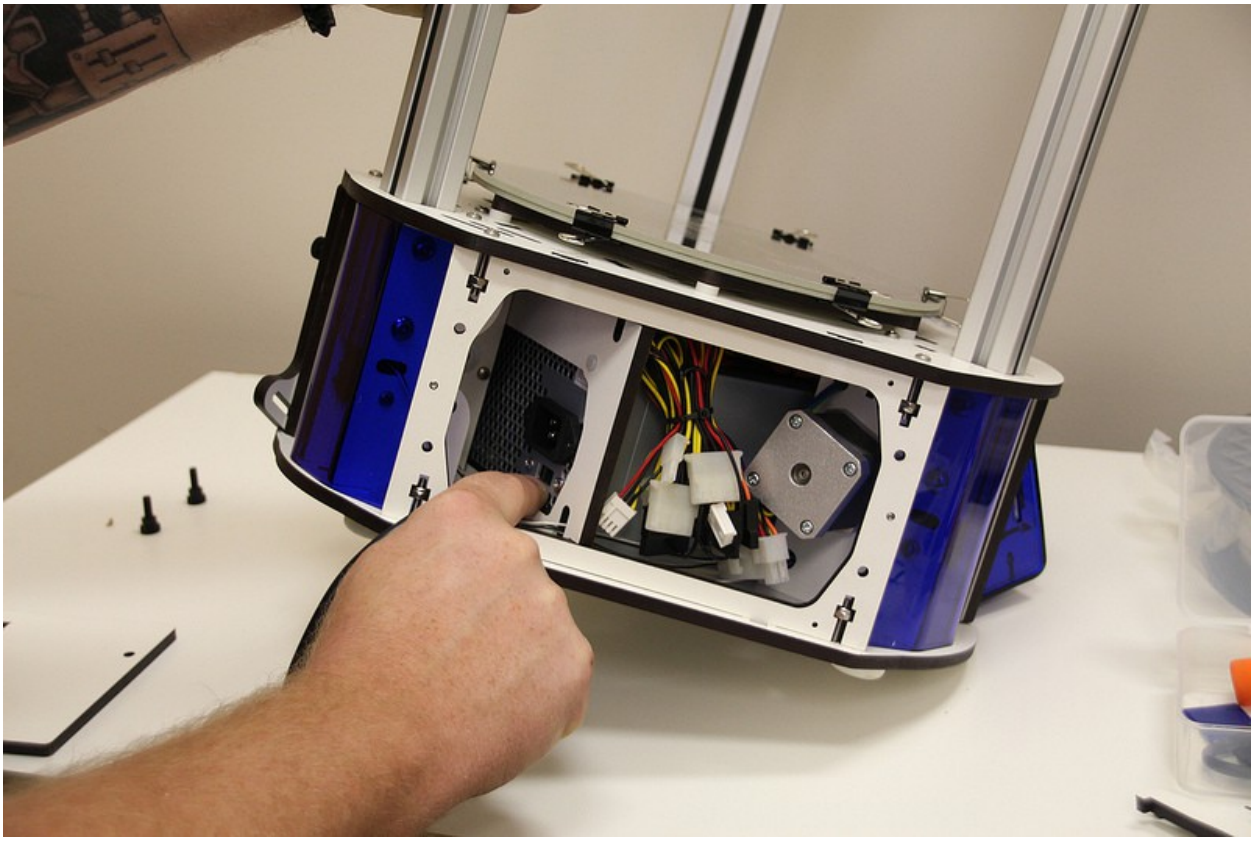
In order to reach the power supply, you'll need to remove the panel that covers it.



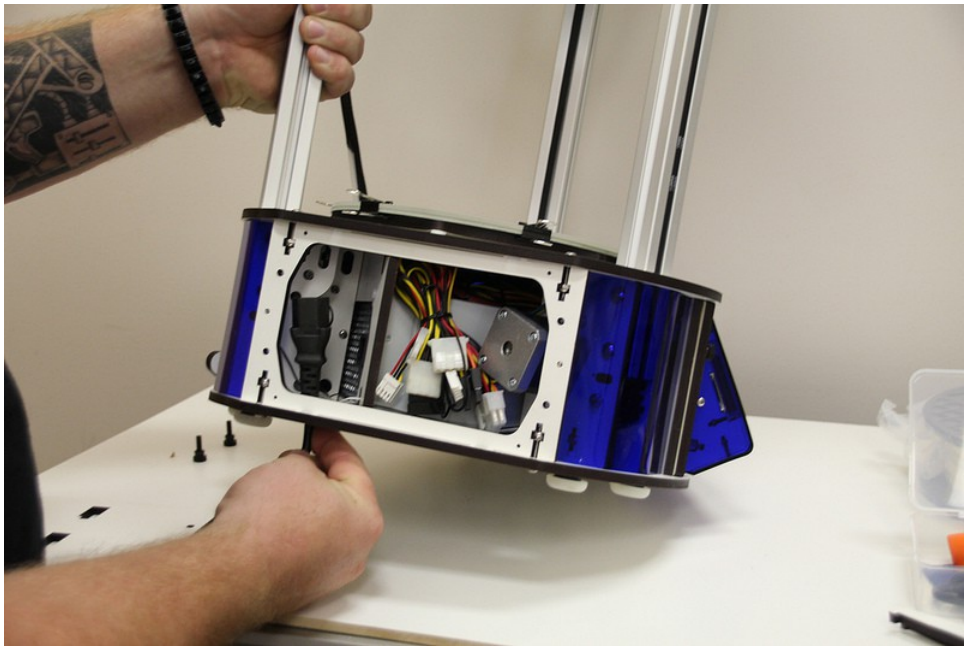
Remove the two black thumb screws and set them and the panel aside. Get the power cable from the box the accessories came in.



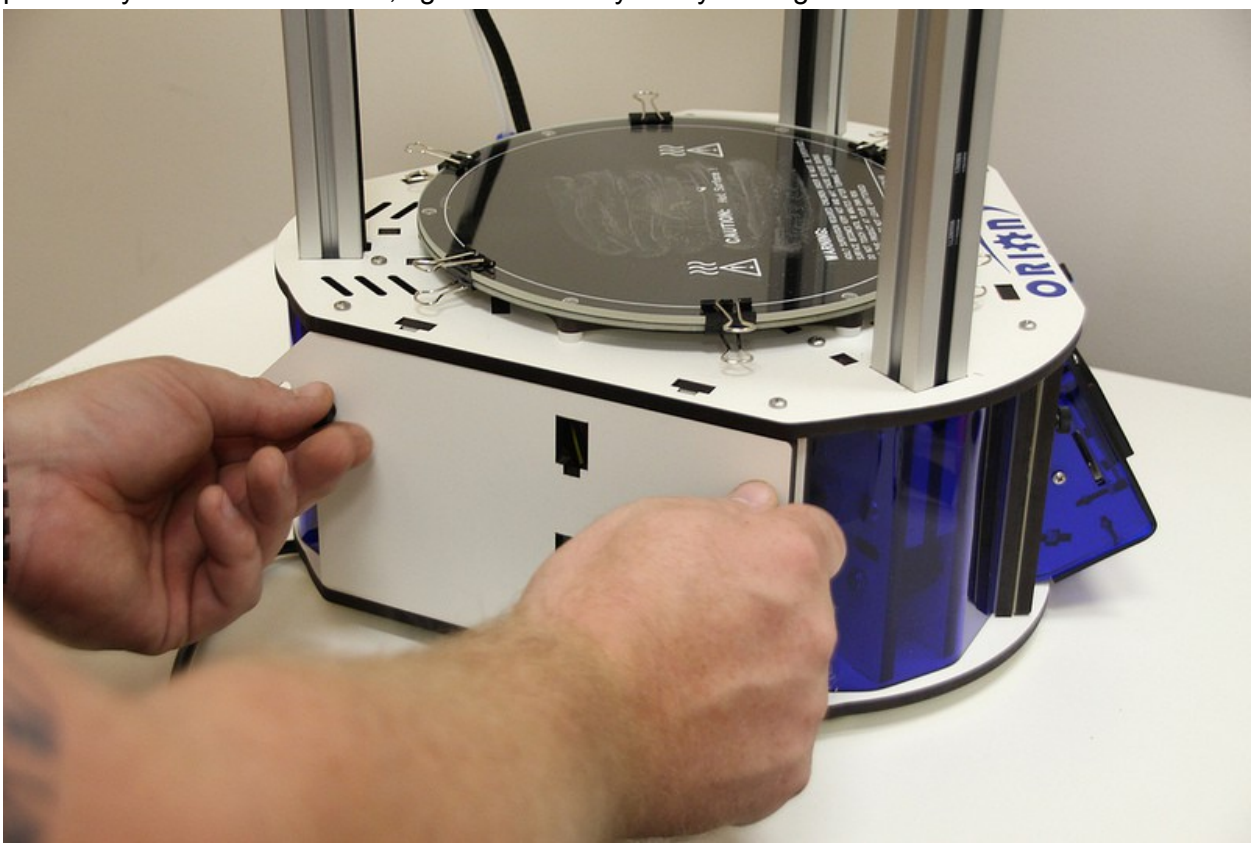
If you're outside the USA and live in a country where the A/C electrical power is 240V, you'll need to flip the switch on the power supply to its 240V setting. This switch is located right below the power socket as shown. You can use a flat tip screwdriver to change its position.



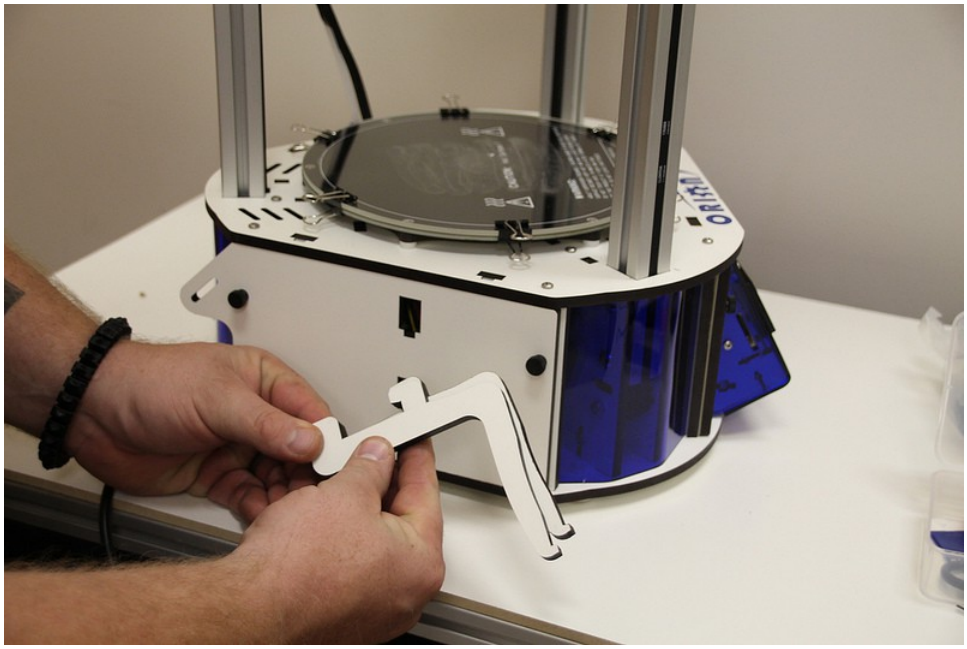
The power cable is installed by routing it through the hole in the base of the Orion Delta™ and plugging it into the socket on the power supply. It's a tight fit, so take your time.



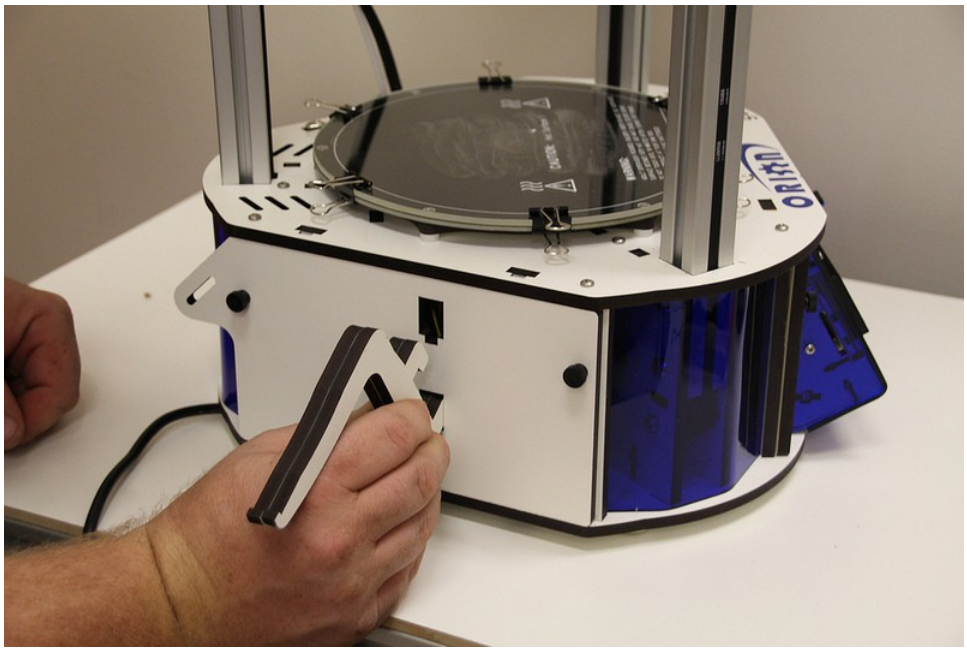
Now replace the door as shown, replacing the black thumbscrews you'd removed previously. As with the others, tighten them only with your fingers.



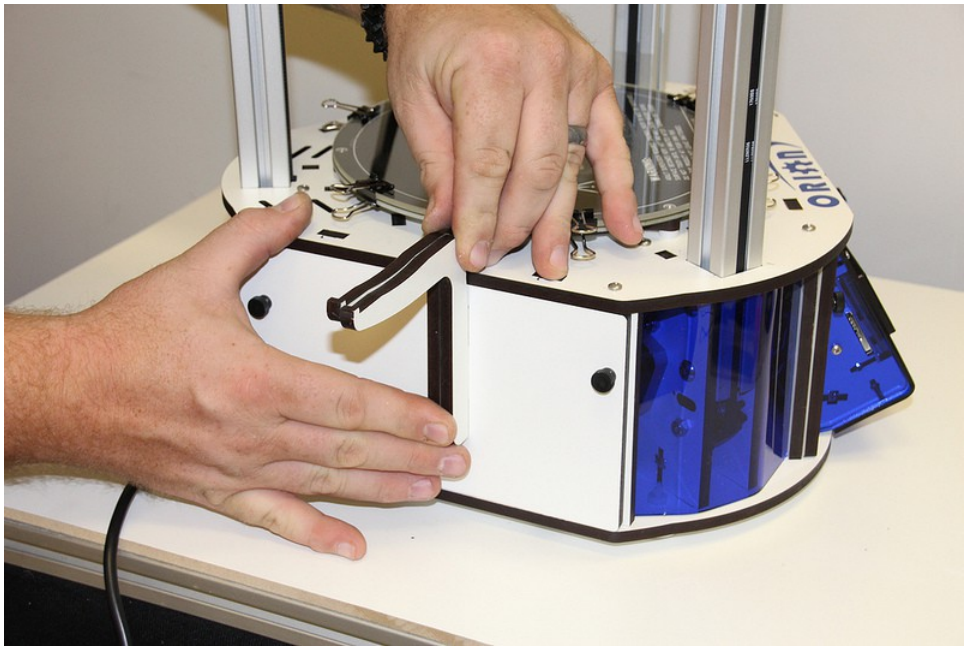
The spool holder is made from two identical laser cut parts.



Holding the spool holder parts together, install in the spool holder mount as shown.

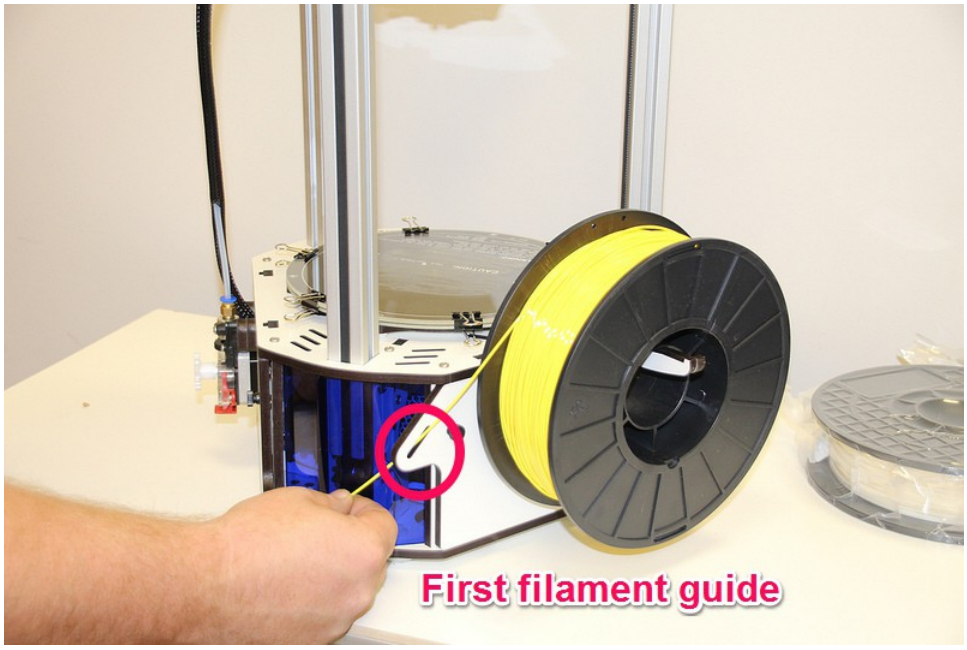


Press down firmly once the hooks on the spool holder are fully inserted into the mounting holes. This will lock the spool holder into place.

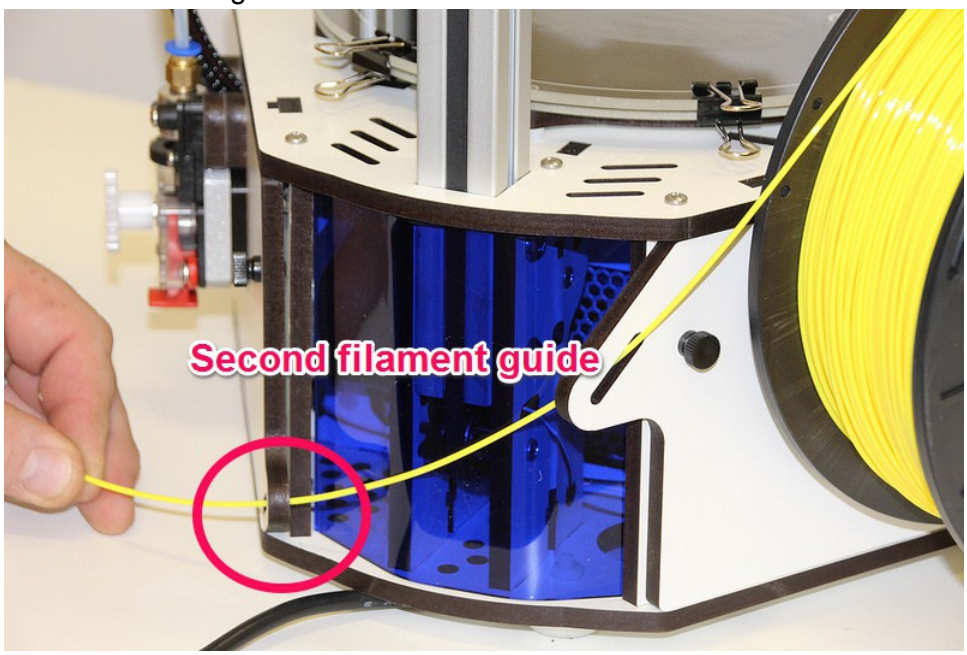


Loading Filament

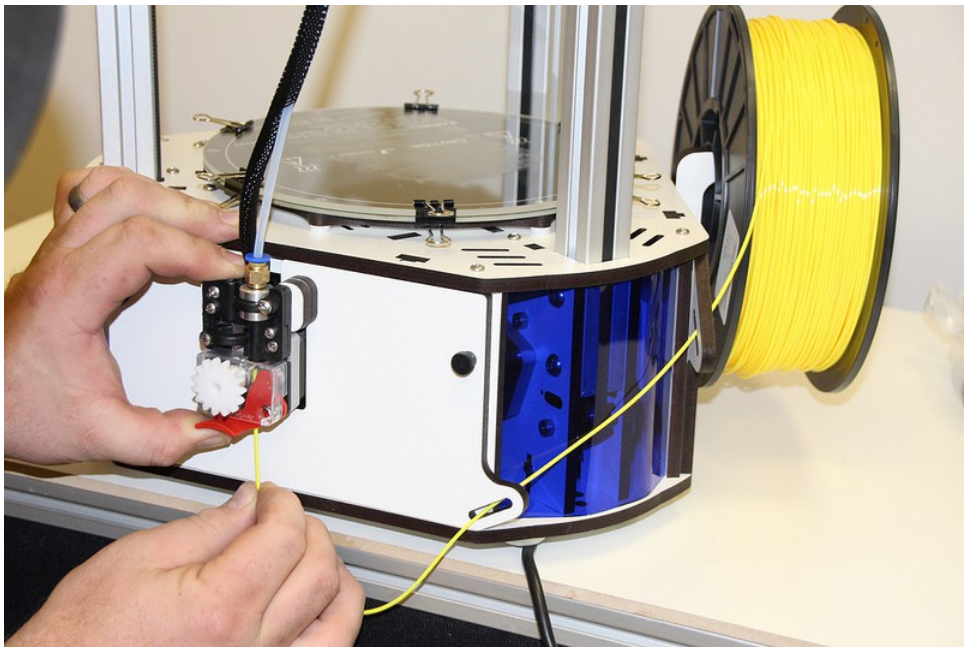
Hang your filament spool on the hanger as shown – you want to make sure that the filament is oriented such that the filament comes off the top of the spool, not the bottom.



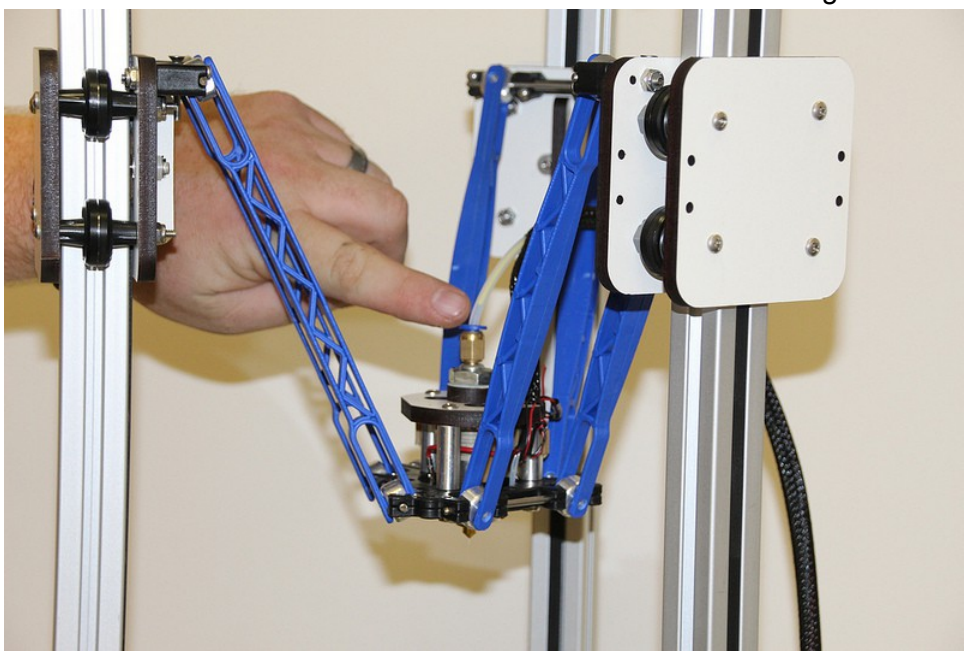
Route the filament through the first filament guide as shown and then route the filament through the second filament guide that's located on the other side of the rear tower.



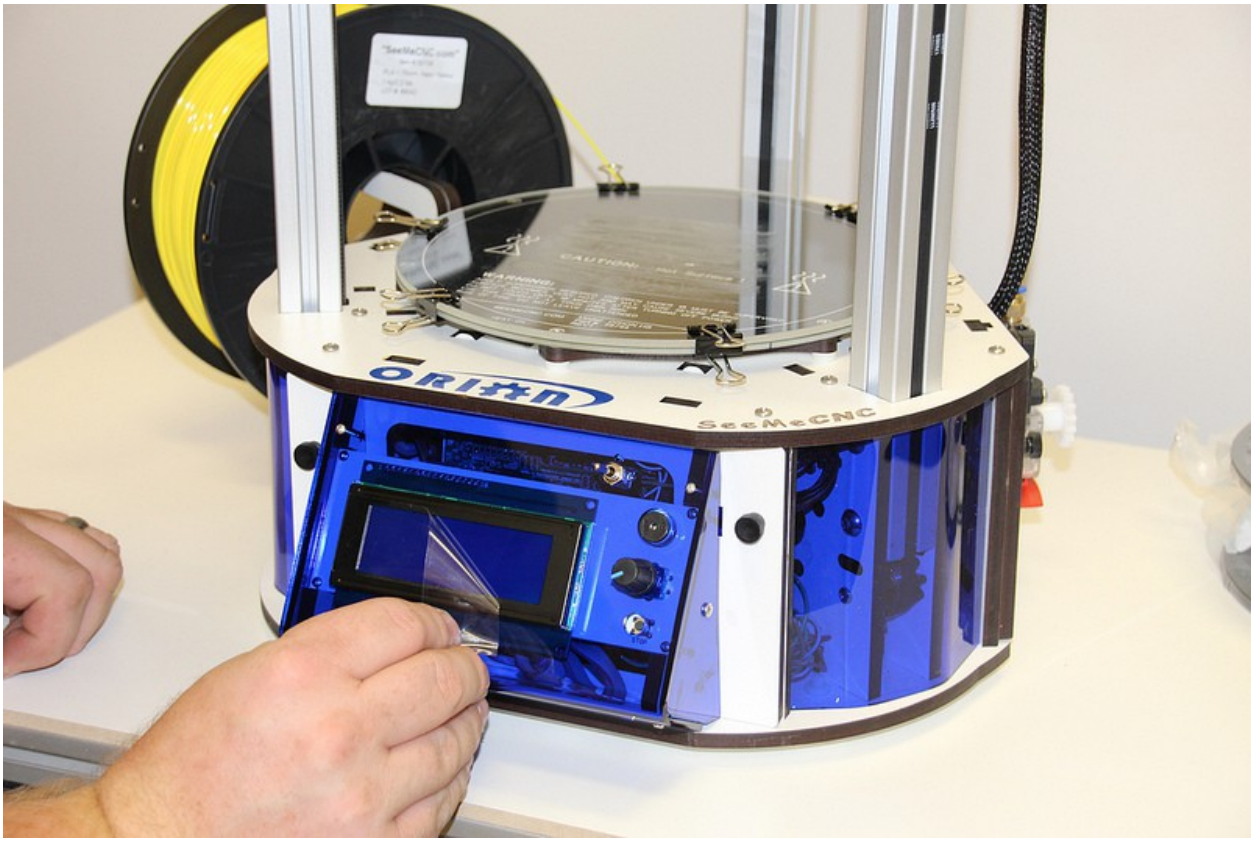
To load the filament through the EZStruder, you'll need to depress the red lever with your thumb (press up) and thread the filament in from the bottom as shown.



Continue to feed filament until the filament enters the hot end through the bowden tube.



Finally, you'll want to remove the protective plastic sheet that covers the LCD.

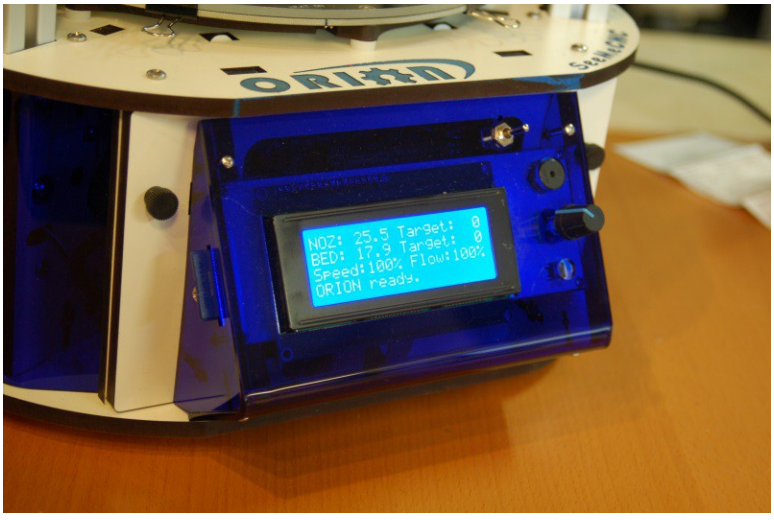


Powering Up your new Orion Delta™ 3D Printer for the First Time

Plug the power cord into a grounded, three prong outlet. Orient the Orion Delta™ to face you and flip the power switch to the right.

You should be briefly greeted by a power on message similar to the one shown below.

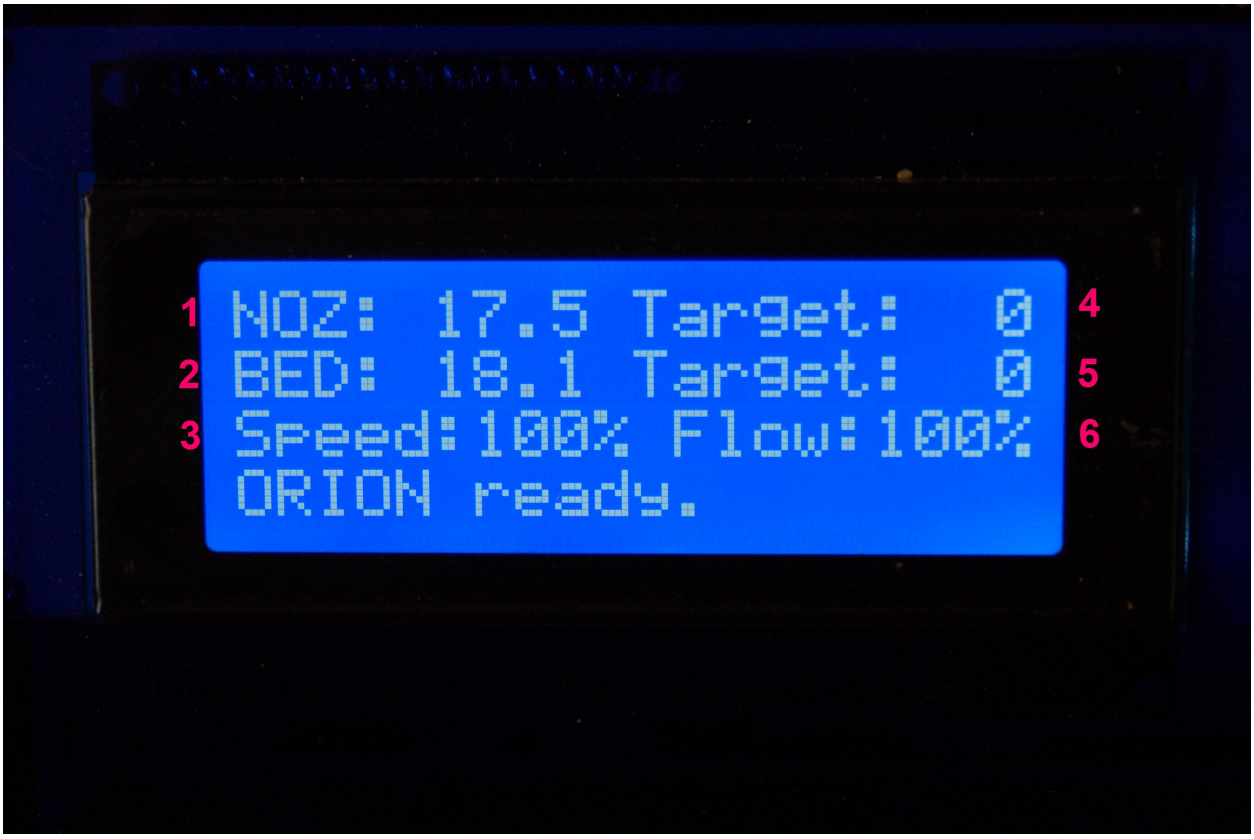
After a short delay, the “front page” of the LCD should be displayed.



Now that you've got your Orion powered up, let's learn what it's all about!

The LCD Control Panel

Before we get into doing final configuration and printing with your new Orion Delta™ 3D printer, let's take a moment to go over the LCD “home” screen so you'll understand what information is being presented to you.



1. Hot end Temperature. This is the temperature at the nozzle as measured by the thermistor in the hot end. It reads degrees Celsius – you'll quickly find that just about everything to do with 3D printing is done in Metric units of measure. FYI, 17.5C is 63.5F.

2. Bed Temperature. This is the temperature of the heated bed as measured by the thermistor in the center of the bed. Just like the nozzle, it reads in degrees Celsius.

3. Speed Rate. This is the speed multiplier field. Normally it will read 100%, but if you've changed the speed control from the host software, this number will display what that setting is. We'll get into this in more detail later.

4. Target Hot End Temperature. When you're printing a part, this field will show you what temperature you've set the hot end to.

5. Target Bed Temperature. This shows what temperature you've set the heated bed to.

6. Extrusion Flow Multiplier. This shows the flow rate of the extruder.

The current hot end and bed temperatures may differ from what you see above – it's entirely dependent on the temperature of the room that your Orion Delta™ is currently in.

When you specify a hot end and/or bed temperature you'll see those set points reflected in the second portion of each temperature display, like the example below.

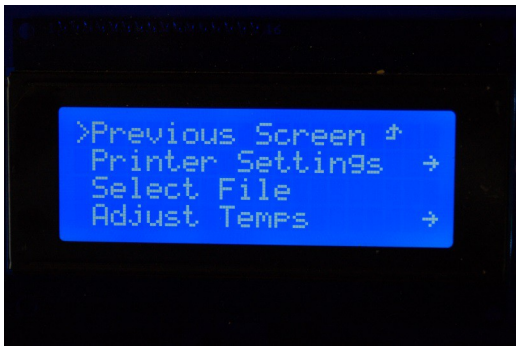
In the image above, the hot end temperature has been set to 200 degrees Celsius and the bed has been set to 90C. Note that just about all aspects of 3D printing is expressed in metric or “SI” units.

You'll notice an additional line at the bottom of the LCD display – this is a “message line” and the firmware in your Orion Delta™ will display information relevant to what it's currently doing on that line.

All control of your Orion Delta™ when not connected to a host computer is done via the LCD panel and the rotary knob to the right of it. The knob will allow you to navigate the various menus and make selections when you press the knob in.

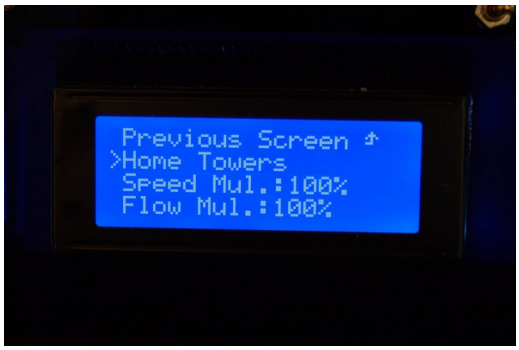
Let's navigate to the “Printer Settings” menu and move the hot end to its home position.

To do this, press the knob in. The interface will beep once and you'll be presented with the following menu:



Rotate the knob to the left and move the selection mark to “Printer Settings” as below:

Press the knob in to select the “Printer Settings” menu. Your LCD should now show the following menu:



Rotate the knob to the left and select the “Home Towers” option. Pressing the button should result in your Orion Delta™ homing all three axes.

Note that if you leave the LCD in “menu mode” for too long, it will automatically revert to the home screen.

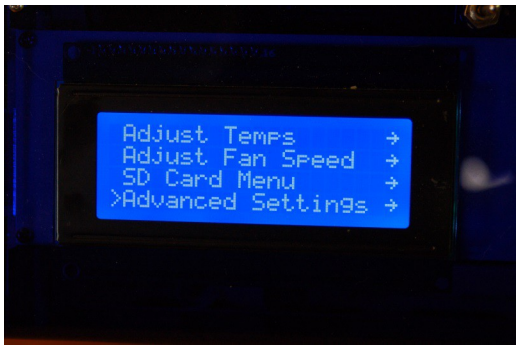
Setting the Z height

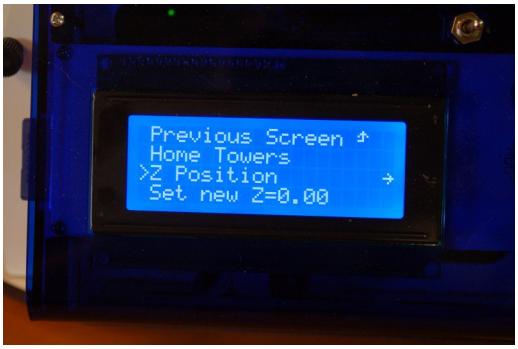
Congratulations! Your new Orion Delta™ 3D printer is alive! But before we get to printing lets take a minute to set the Z height of the machine as it could have been bumped during shipping and it needs to be super accurate to get the best first-layer adhesion of your prints.

Using the control panel on the machine to set the Z height is really easy. You'll find you may need to do this from time to time or after changing to a new build plate or nozzle etc. Pretty much anything that could change the height as measured from the tip of the nozzle to the built plate.

To set the Z height you want to warm the bed and hot end to close to printing temperature to let any heat expansion take place. To do so, click on the knob and scroll to "Printer Settings" then "Preheat ABS" and press the knob. This will set the heated bed to 90c and the hot end to 200c. This is less than the melt temp for ABS, but a good holding temperature that will make sure that any ABS filament doesn't burn as it sits in the hot end waiting for the bed to heat up. It may take up to 15 minutes to heat the bed to 90c depending on the room temperature, but it's important to let it heat up before setting the Z height.

Once the bed and hot end are up to the target temperature, click on the knob to reach the menu and scroll down to "Advanced Settings". Then down to "Calibrate Z Height" and then down to "Home Towers". Press the knob to home the machine – we're giving it a starting point for the next step.





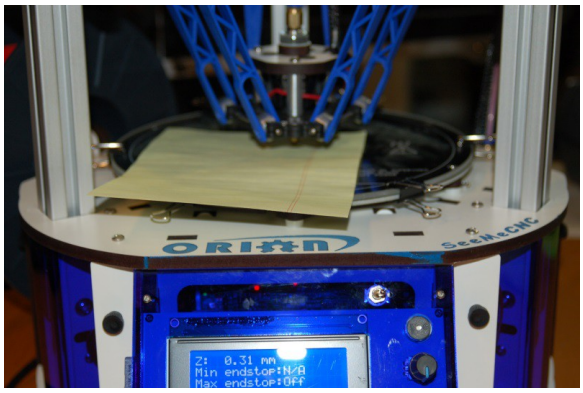
After the machine is homed, scroll to “Z Position” and click.

You'll see the display change to what is similar to the photo above. In order to move the platform down, you'll turn the knob counter-clockwise. If you turn it quickly, you'll get large movements and when you turn it very slowly, you'll get a .01mm per-click change.

Use the knob to lower the platform so that the nozzle tip is about 1/2” above the glass build surface.

Make sure the nozzle is clean and there is no filament hanging from the nozzle (Be careful, it’s hot now!). Take a single sheet of notebook paper and place it under the nozzle.

Turn the knob slowly and jog down until the nozzle just begins to “snag” on the paper. Now press the knob to return to the menu and and scroll down to “Set new Z=0.00” to store the new Z height to memory. That’s it! You’ve now set the Z height to the table and you're now ready to prime the hot end and begin your first print!

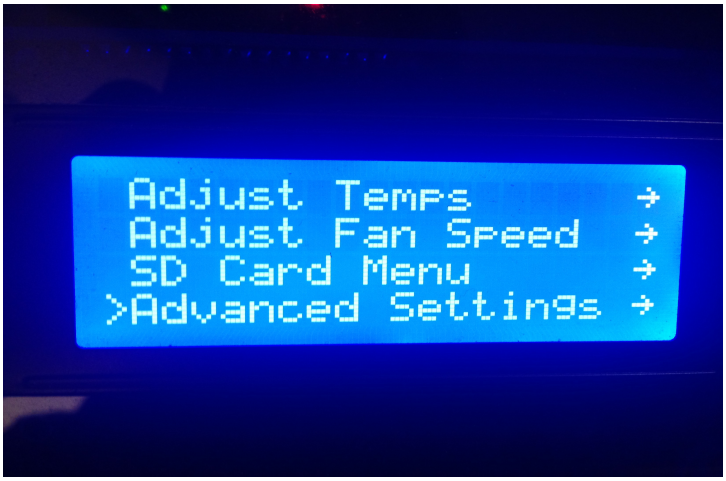


Leveling your new Orion Delta™ 3D Printer (The Easy Way!)

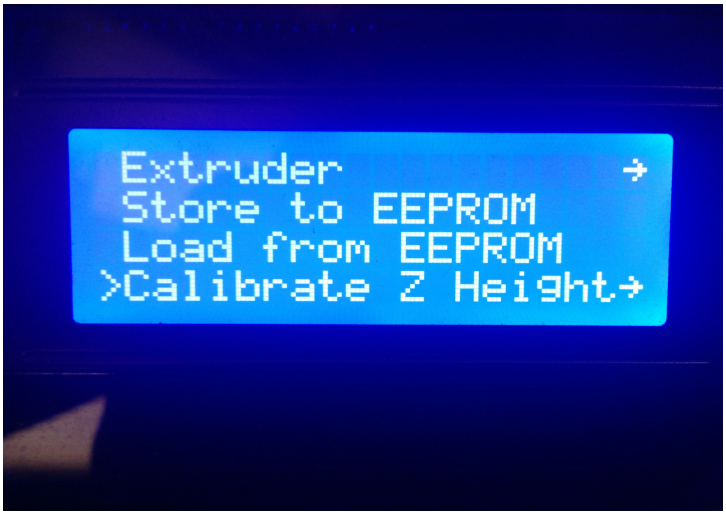
Sometimes the abuse the Orion can experience during shipping can adversely effect the factory calibration. If that's the case with your printer, you can use this easy to follow guide in order to re-calibrate your Orion. This procedure was written by the gentleman that built and calibrated your printer, so there literally is no more authoritative source for the process.

1) Preheat hot-end and bed to the material you use. If you use both, just use the “Preheat ABS” in the printer settings menu on the LCD.

2) Once the hot-end and bed are up to temp, on the LCD, click the knob to bring up the menu, go to Advanced Settings (at the bottom of the list). Click the knob.



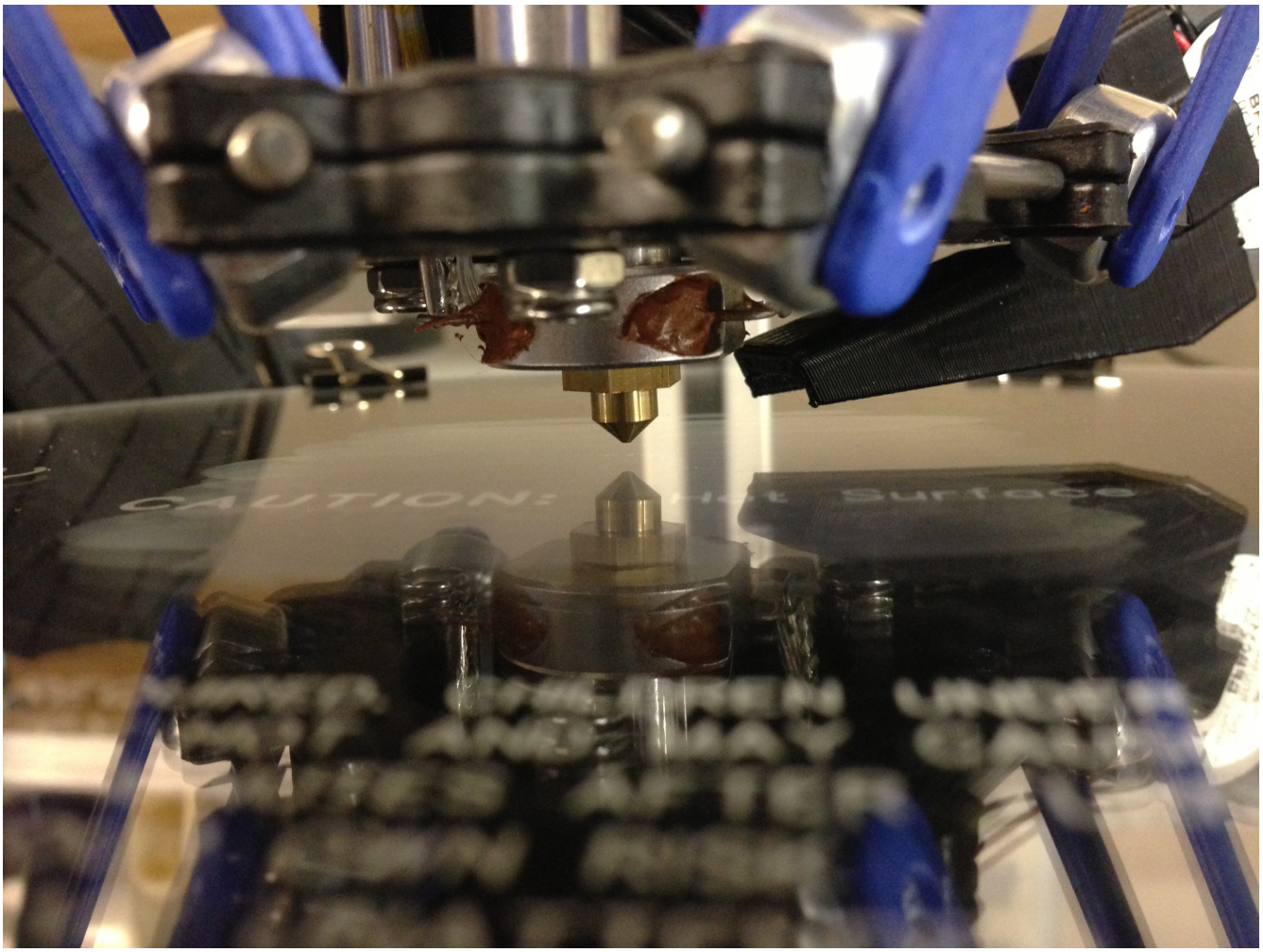
3) Go to the bottom of the next menu to Calibrate Z Height. Click the knob.



4) Now you are in the calibration menu. Click Home Towers. The machine will go up and hit all the end-stops.

5) Go down to Z Position and click the knob.

6) Turn the knob Counter-Clockwise to lower the nozzle. Bring the nozzle down till its a couple millimeters from the glass like the following pic.



7) Turn the knob counterclockwise SLOWLY and look eye level with the bed, and bring the nozzle down till it just barely meets the glass (this is easiest with a white wall in the background so you can watch the reflection of the nozzle – if you don't have a white background to use, you could tape a sheet of paper between the towers to give you enough contrast). When the nozzle touches the glass, click the knob.

8) Go down to Set New Z=0.00 and then click the knob.

9) Go to Home Towers. Click the knob. You have now set the z height, and now it is time to calibrate the towers.
The next steps can be done through a computer or SD card. I find it is easier on a sd card so you can keep close to the machine to watch the movement.

10) Open notepad or a text editor of your choice.

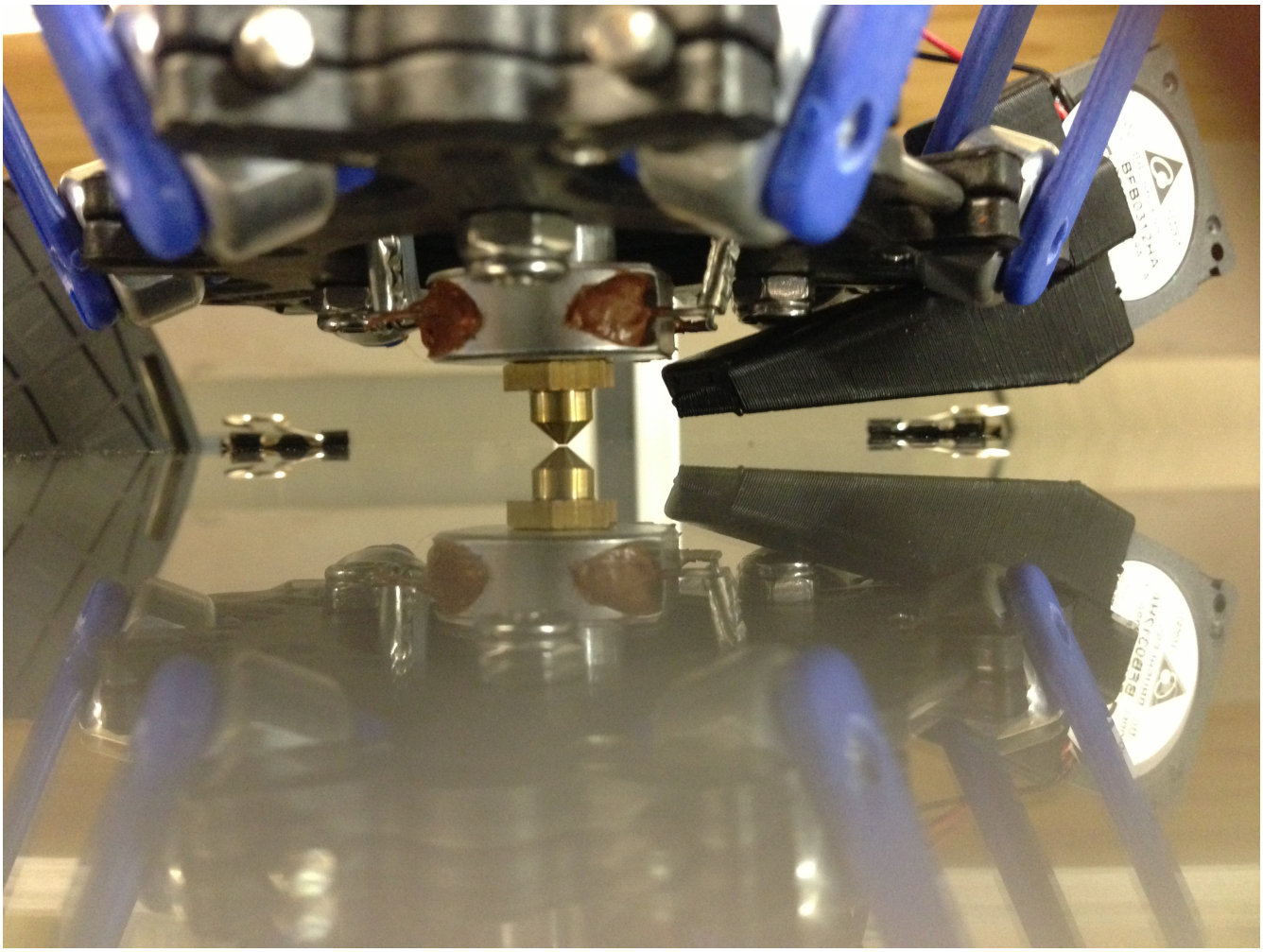
11) Paste the following into the text editor:

```
; Tower endstop calibration script
G28
G1 Z.2 F15000
G4 S1 ; pause for 1 second
G1 X-65 Y-37 F2000
G4 S2 ; pause for 2 seconds
G1 X0 Y0
G1 X65 Y-37
G4 S2 ; pause for 2 seconds
G1 X0 Y0
G1 Y65
G4 S2 ; pause for 2 seconds
G1 X0 Y0
```

12) Save the file as “Towers.gcode” and place it on an SD card, or if you are going to be using a computer, load it into your host software that you use.

13) Again with the bed at eye level and looking close, Run Towers.gcode from either your host software or SD card. Watch closely as the machine will home, then the nozzle will drop to 0.2mm ABOVE the glass.

The gap will look like the following pic.



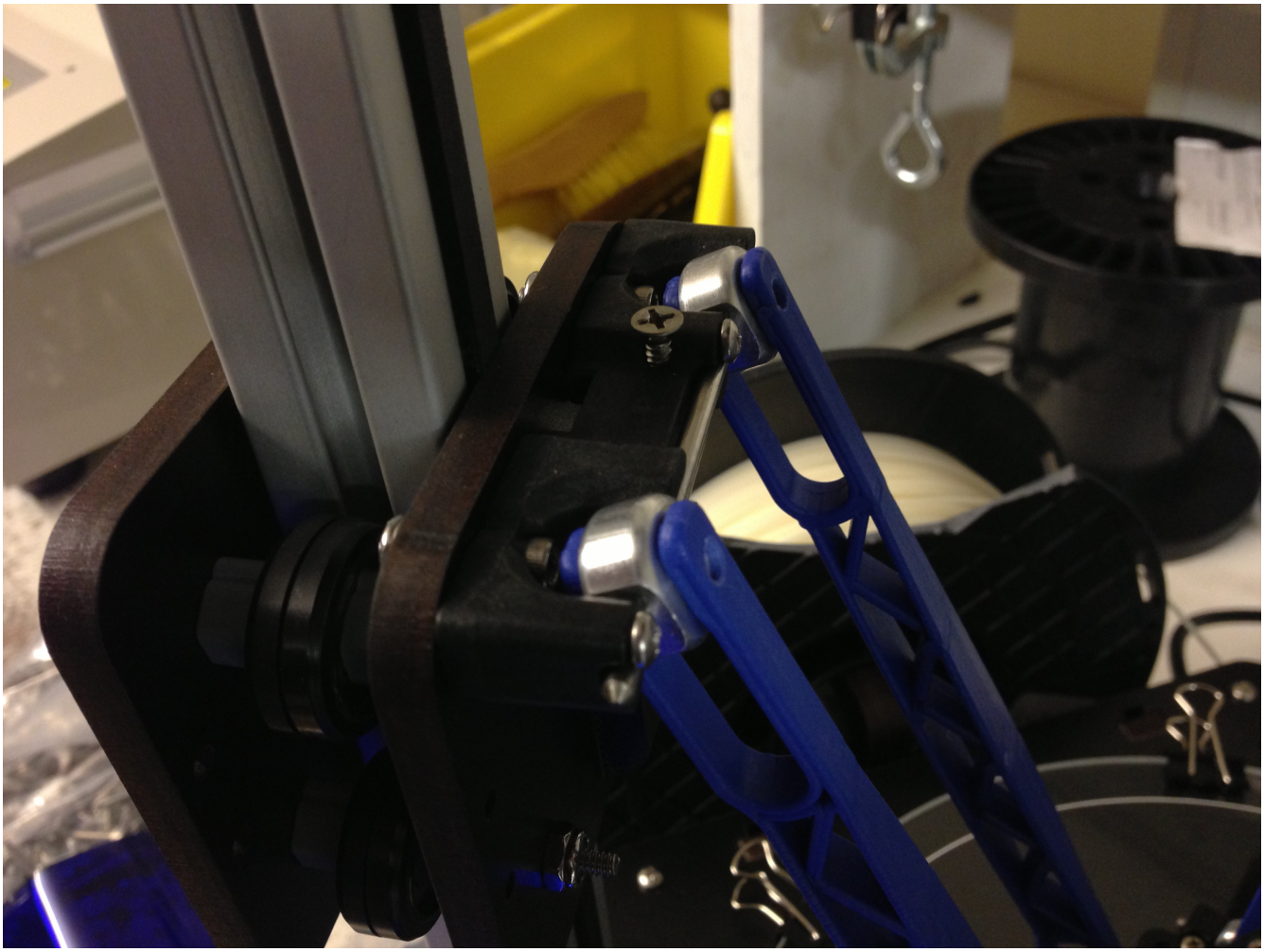
After it drops to the center like that, it will travel to the X tower. DO NOT pay attention to what the nozzle does while traveling, what you must pay attention to is when it pauses.

The nozzle will pause at the X corner, then return to the center, and move to the Y tower and pause. After the pause, it will again move to the center, and move to the Z tower, pause, and return to the center.

14) What you want to remember is the nozzle when it pauses. You want to compare the movement to the gap at the center.

– If the nozzle at the tower RAISES (compared to the CENTER), you want to adjust the screw that hits the end-stop by turning it counterclockwise (moving the screw up).

– If the nozzle at the tower LOWERS (compared to the CENTER), you want to adjust the screw that hits the end-stop by turning it clockwise (moving the screw down).
Pic of screw you will adjust:



*** Make small changes to the screw about 1/8 – ¼ turn adjustments***

15) After you have adjusted the screws, You have changed the Z height, so you want to go back and re-set the Z height the way you did earlier in this guide.

16) Re-run Towers.gcode and watch as you did before, again noting the changes in the nozzle at each pause. You will run through these steps till the nozzle remains the same height at each tower when compared to the center.

If the nozzle goes the same direction on all 3 towers (such as you have the gap in the center, and at every tower the nozzle lowers. Or you have the gap at the center and at all 3 pauses the nozzle raises), you will adjust the radius in the following way.

17a) If from the center gap, the nozzle goes DOWN toward the glass at ALL 3 TOWERS, load your host software and bring up the EEPROM information. You will look for Horizontal Radius. You want to RAISE that number. I suggest raising it by 0.2 and run towers.gcode to see the change, and keep raising the number till the gap evens out (changing this number will not make you need to re-set your z height, it will just raise the outer edges where the nozzle pauses).

17b) If from the center gap, the nozzle goes UP toward the glass at ALL 3 TOWERS, load your host software and bring up the EEPROM information. You will look for Horizontal Radius. You want to LOWER that number. I suggest raising it by 0.2 and run towers.gcode to see the change, and keep raising the number till the gap evens out (changing this number will not make you need to re-set your z height, it will just lower the outer edges where the nozzle pauses).

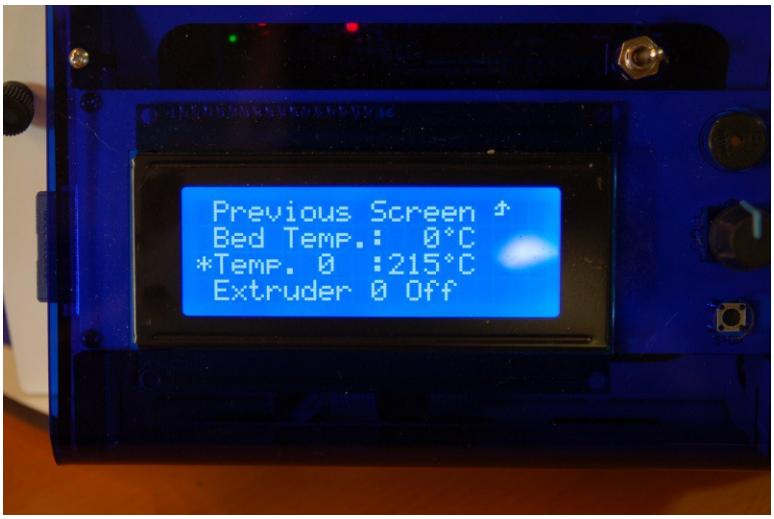
After doing this, you will see any changes where one tower may be higher than another, if this is the case, go back to adjusting the end-stop screws as before.

Typically it can take anywhere from 5-10 or so re-runs of the tweaks to get the gap to remain the same at all 3 pauses compared to the center of the machine. Once the gap is the same at each tower compared to the center, your machine is calibrated and ready to print!

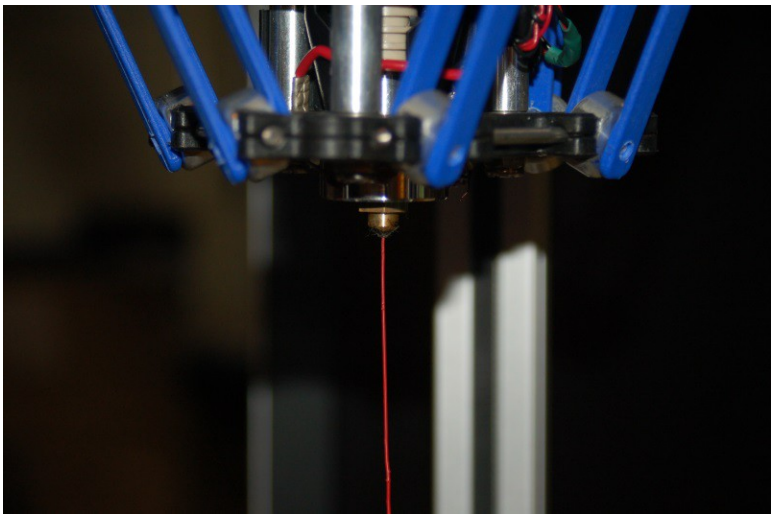
A Simple Guide to Hot End Priming

Before you can print with your new Orion Delta™ for the first time (or any time you load new filament), you'll need to prime the hot end with the new material. Fortunately, this is a very simple task!

Now on the LCD screen, click and go to “Printer Settings” and scroll down to “Disable stepper” and click. This will unlock the motors allowing you to turn the extruder by hand in order purge the hot end. Now go to the menu and select “Adjust Temps” then scroll down to “Temp. 0” and click, then set the temp to 215 and click to set the new hot end temperature. You can wait for the screen to go back automatically to the main screen or scroll up and click “Back” and get to the home screen.



Once the hot end is up to temperature, reach around the right side and rotate the knob on the extruder counter clockwise to feed filament slowly into the hot end and you will see it start to flow out the nozzle. Let it flow out about 4” or so, then stop and remove the hanging filament.



That's all there is to it! You're ready to print!

Printing from the SD Card

The SD card included with your new Orion Delta™ 3D Printer has folders already on it with some sample prints as well as the firmware that was used to calibrate your machine.

The SD card goes in the left side of the control panel, label facing inwards, through a slot in the blue acrylic side panel as shown below.

If you've got an SD card reader on your computer, you can easily save files to the SD card in order to print with your Orion Delta™ in “stand-alone” mode. You don't need to connect the printer to your computer in order to print!

Let's take a look at the demo files that were included on the SD card that was shipped with your Orion Delta™.

Click the control knob to enter the LCD menu and scroll down to the entry marked “Select File” and click.

Scroll to the “Print file” menu item and click. This will get you into the top level directory of the SD card.

The odd little symbol you see to the left of the directory names are actually little folder icons. This simply helps separate the directories from g-code and other files on the SD card.

Click the “GCODE” directory to see a list of the files included.

For this first print, go ahead and click on the file, “BLINKY.GCO”.

When you select the file name, the LCD controller will make a “chirping” sound and the heated bed will begin to heat up. Once the bed has reached its target temperature, the printer will home all three axes and then the hot end will begin its heating process. Once that has completed, the printer should begin the print.

Note – before you run a print job on your Orion Delta™, you need to apply a <u>thin</u> layer of adhesive using the included glue stick. This will allow the ABS plastic to stick to the glass. Apply the glue stick in a thin layer of parallel lines. Let it dry and repeat the process, using lines perpendicular to the first layer. Once it dries you can print.

Do NOT apply the glue stick to a hot bed!

If you need to cancel a print job for any reason, simply turn the machine off and use the control panel to re-home the towers.

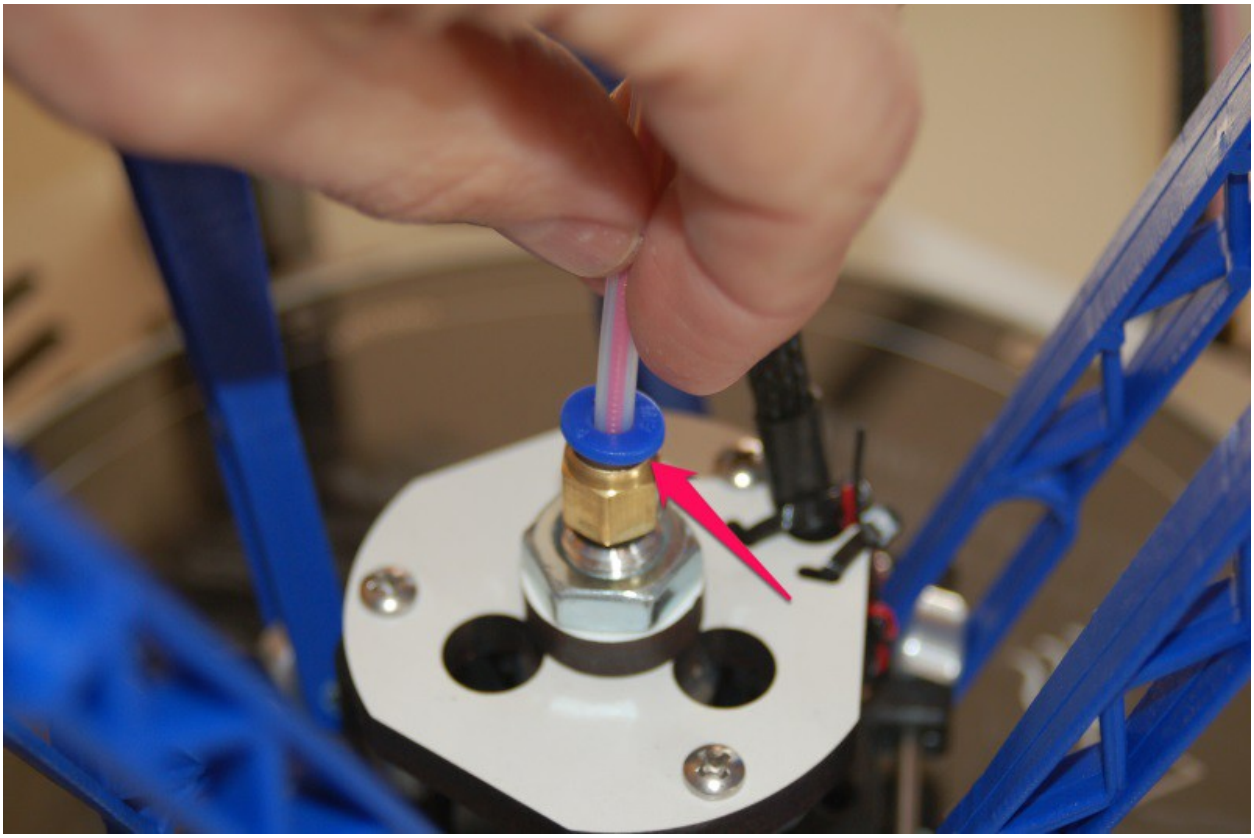
Changing Filament

Changing the filament on your Orion is a very simple process.

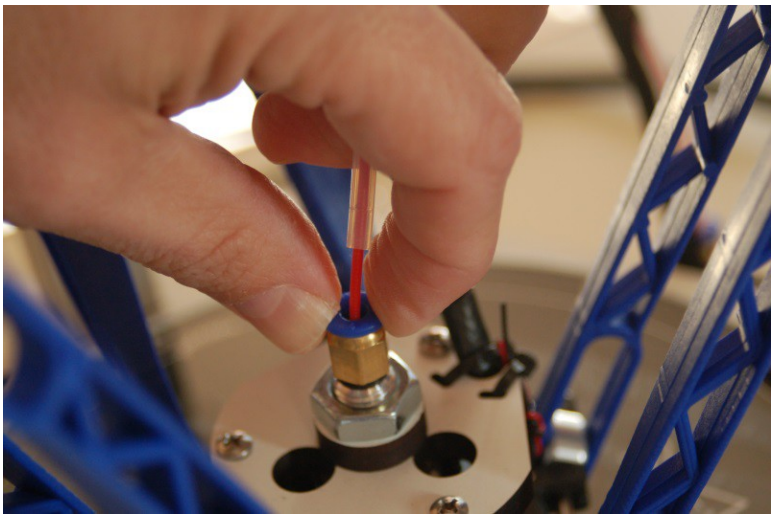
First, you'll want to bring the hot end up to the temperature you normally set it at when you're printing.

Once the hot end is at operating temperature, pop the bowden tube off the hot end as shown in the following steps.

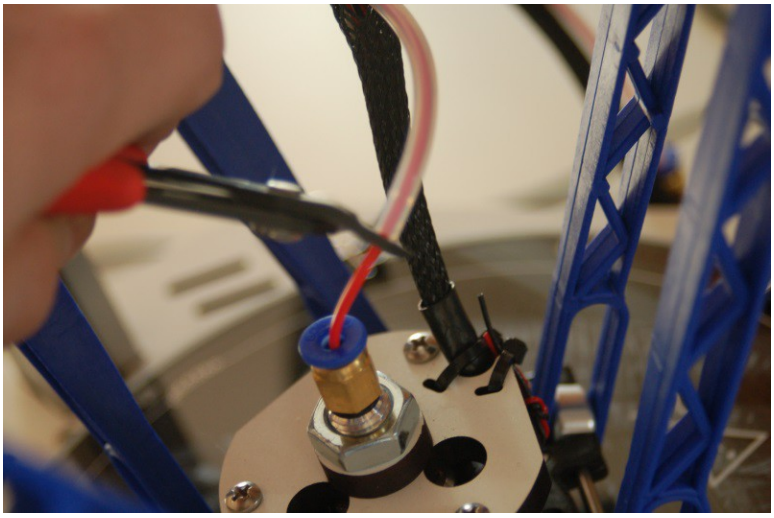
1. Grip the bowden tube with one hand and press down the blue ring and pull up on the bowden tube. You may need to depress the red release lever on the extruder in order to get enough slack to pull the bowden tube free of the hotend fitting.



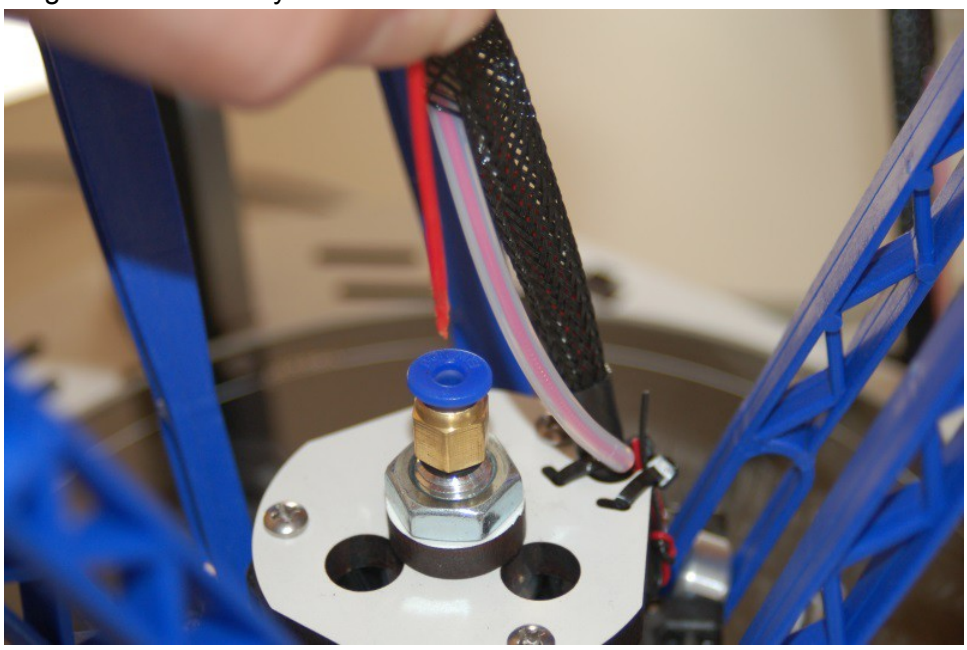
When the filament pulls free, it should look something like the photo below.



2. Cut off the filament flush with the end of the bowden tube.



3. Pull the filament stub from the hotend and then re-insert the bowden tube into the hotend, making sure it seats fully.



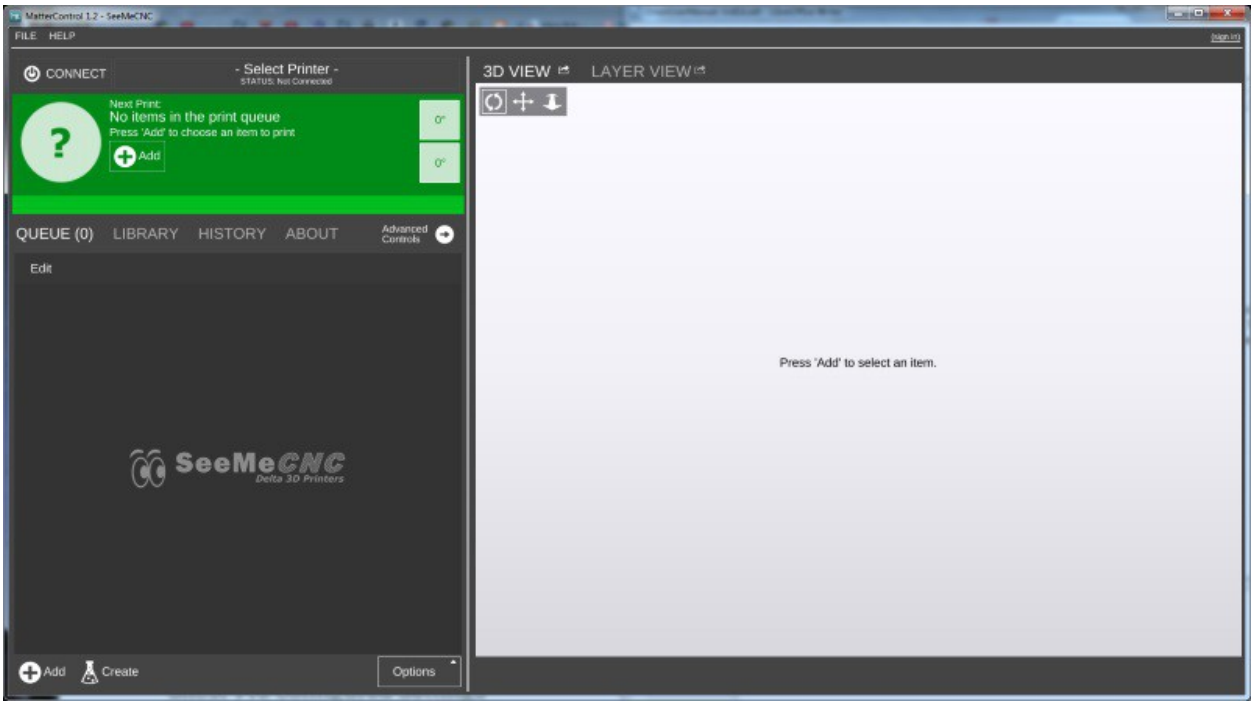
Printing From Your Computer

In order to print from your computer, you'll first need to get the right software installed. Since the machine I use with the Orion is a Windows based PC, these instructions will focus on that platform. If you're using MacOS, the broad strokes of this section will apply to you.

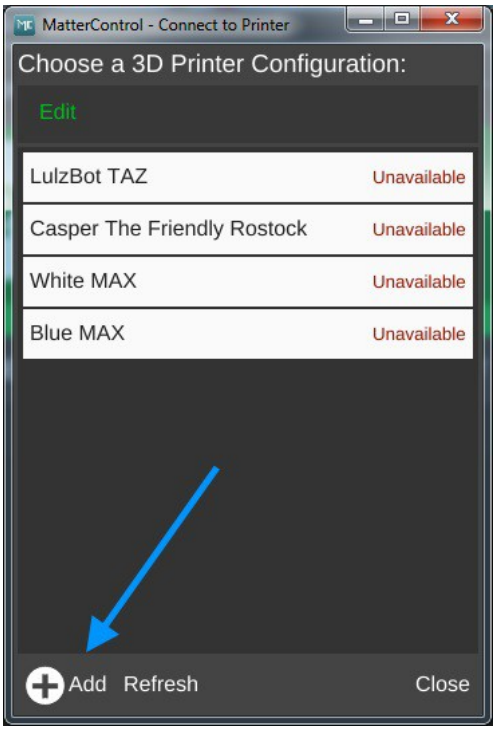
The software used to send jobs to the printer (among other things!) is called MatterControl. From now on, I'll refer to it simply as “the host software”, or some cryptic variant of that.

You can download MatterControl by going to the SeeMeCNC website – http://seemecnc.com/pages/downloads. Scroll down to the “Software” heading and you'll see the links for MatterControl. Download the version for your platform and install it.

Start MatterControl. Below you'll see what the basic MatterControl interface looks like. Your colors may be different – I'll show you how you can customize that later on.



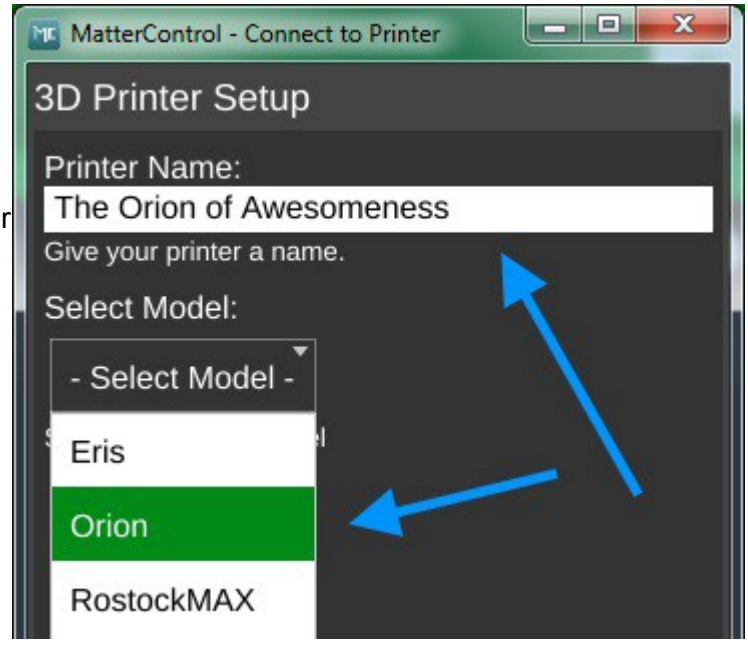
Now that you've got MatterControl open, you need to tell it about the Orion so you can start printing! Click on “File” and then “Add Printer”. The following dialog will open:



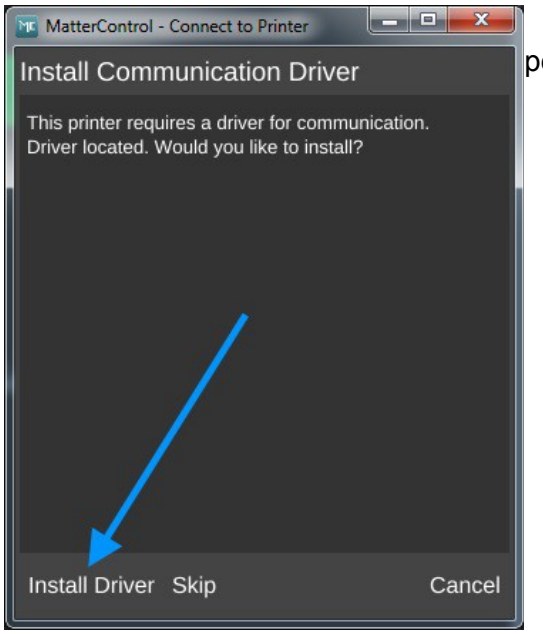
This is where you can manage all your different printer configurations. Click the “Add” icon as indicated by the blue arrow.

Clicking on the **Add** icon will open up the 3D Printer Setup dialog. Here is where you'll name the new printer configuration and where you select your printer model.

So go ahead and enter a printer name and then chose “Orion” from the **Select Model** list. Once you've done this, the **Save & Continue** button will appear at the bottom left corner of the window. Click on it to save your new printer configuration.



After you've saved, MatterControl will prompt you to install the driver needed to communicate with the Orion.



Click on **Install Driver**. Once you've given permission to run the installer, the following dialog will appear:

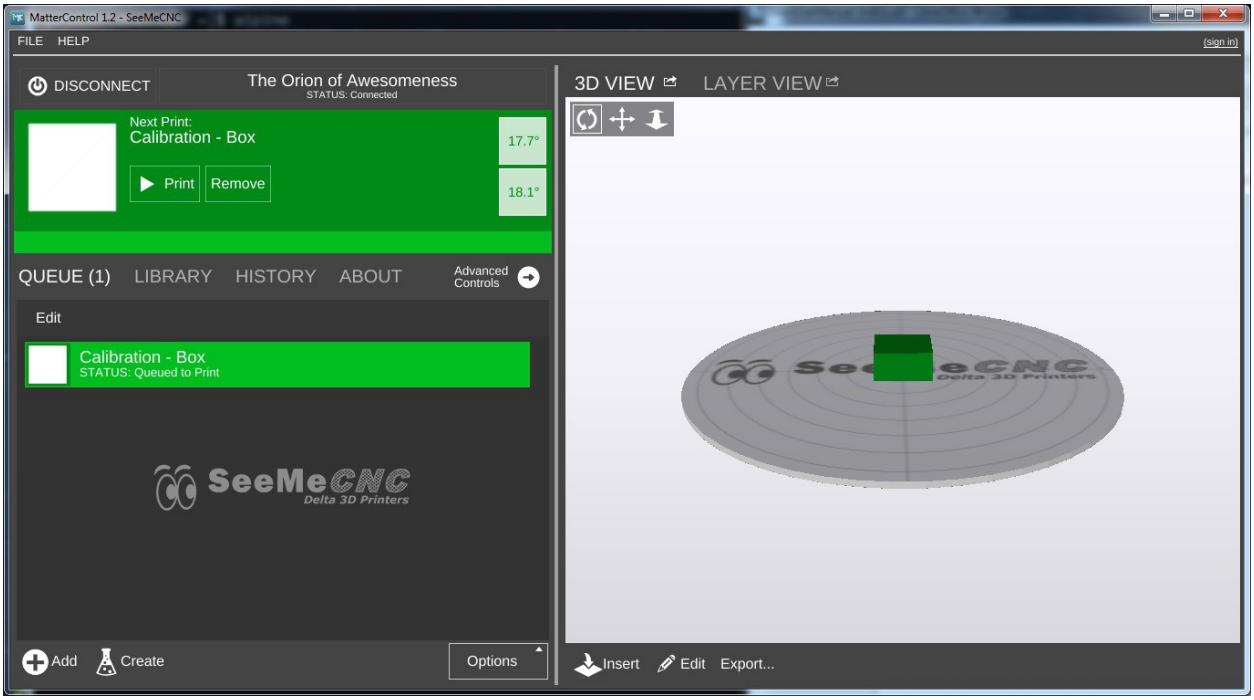
Click on the **Install** button to install the driver. When the installer is finished, MatterControl will continue on to auto-detect the printer.

Make sure that you've got your Orion disconnected and then click **Continue**.

Once you click on **Connect**, the following dialog should appear:

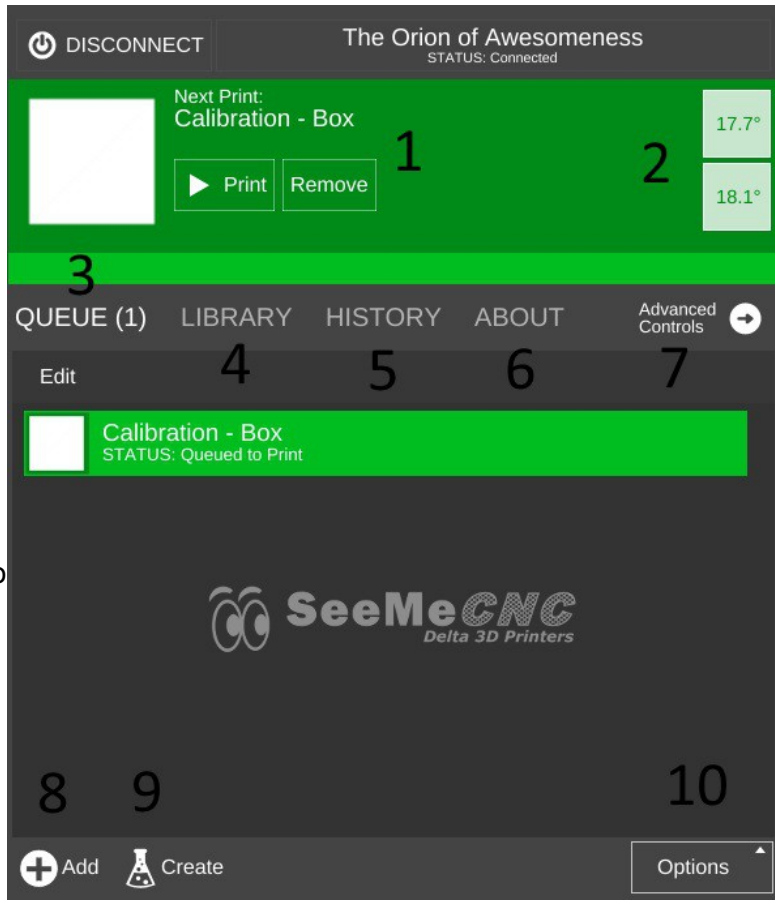
Congrats! You're all done. Click on the **Done** button to continue.

At this point, your MatterControl screen should look something like this.



The configuration process automatically loads a test object that you can print. Let's take a few minutes to go over the various features of the MatterControl interface.

The MatterControl interface is split into two halves. The left half is where you can load objects or G-Code to print, start prints, and manually control the printer.



1. This is the print queue display. If nothing is queued up, you'll see a message indicating that there no items in the queue to print.

2. This is the temperature display. The top figure displays the current nozzle temperature and the bottom figure displays the current heated bed temperature. Both values are shown in degrees Celsius.

3. Queue Count. This shows the number of objects currently in the print queue.

4. This is your object library. We'll go over this in detail later.

5. Print History. This will show you what you've printed in the past along with statistics about each print.

6. About. This gives you information about the fine folks that created MatterControl. It also includes a button to send them feedback as well as a button to check to see if any updates are available.

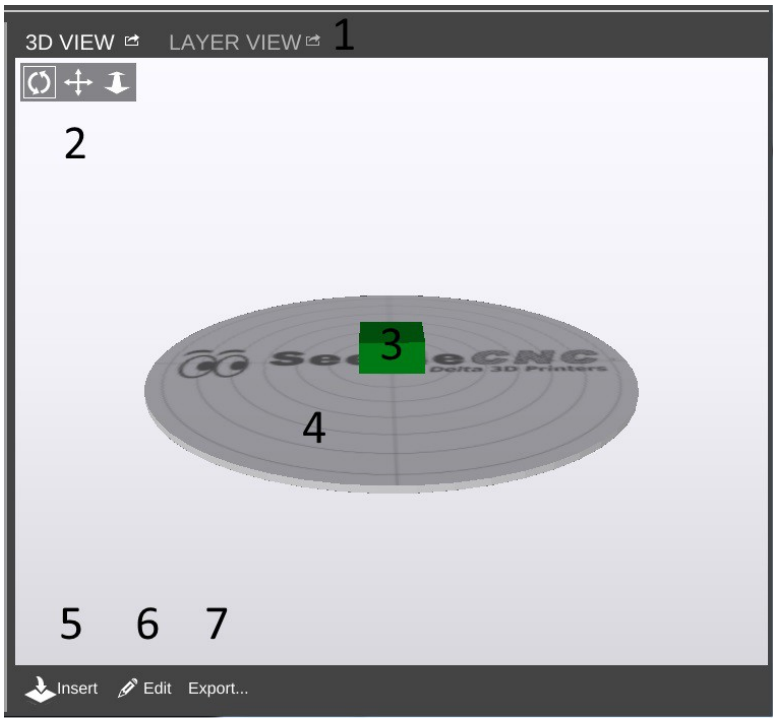
7. Advanced Controls – this is where you can manually control the printer. We'll cover this later as well.

8. The **Add** button will allow you to add objects or G-Code files to the print queue.

9. The **Create** button displays a list of available plug-ins that are used to create printable objects right inside of MatterControl. We'll cover this one later.

10. The **Options** button opens a menu list that will allow you to export the current file and perform other operations on the print queue. You guessed it! We'll cover this one later.

The right half of the MatterControl interface is taken up by a 3D view of your build platform and what objects are currently loaded and ready to print.



1. The 3D and Layer view controls allow you to switch between the 3D view (shown) and the Layer view. The layer view shows you the path the print head will take as your part is printed. The layer view won't display anything until the part you want to print has been “sliced”.

2. View manipulation controls – Rotate, Pan, and Zoom. By default, the 3D view will show a rotating display of the part. You can stop the rotation by clicking anywhere in the 3D view window.

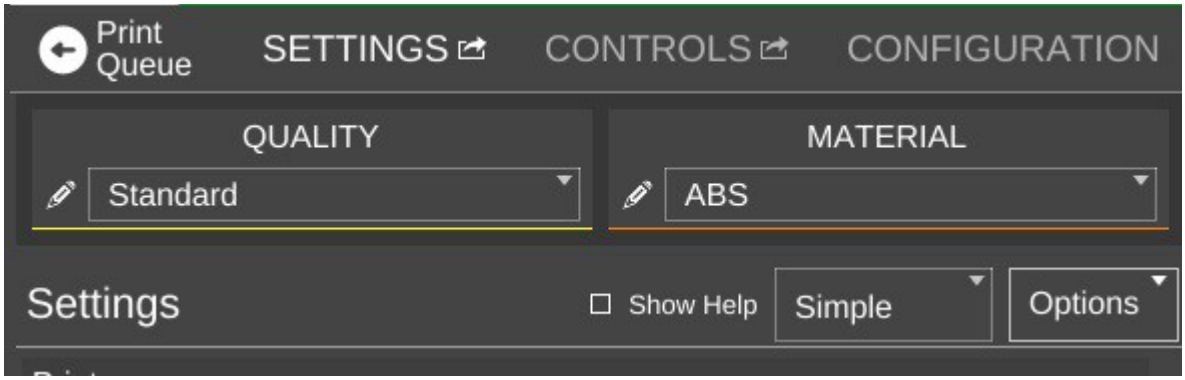
3. The object currently ready to print.

4. This is a representation of your Orion's print bed. As long as your object fits within the circle, you can print it!

5 & 6 The **Insert** and **Edit** buttons allow you to add and manipulate multiple objects to the current print job. We'll cover this in depth later as well.

7. The **Export** button allows you to save the current state of the build plate in a few different file formats.

Since MatterControl was nice enough to provide us with a test object, what say we print it? Click on the **Advanced Controls** button on the left pane.



For right now we're only going to worry about the **QUALITY** and **MATERIAL** settings. Click on the **QUALITY** drop down and select **Standard**. Click on the **MATERIAL** drop down and choose the filament that you've got loaded in your Orion. Since I've got ABS loaded, that's what I chose. **Make sure you've applied glue stick to the bed as mentioned earlier in this guide!** Now click **Print** button at the top of the display to start the print job!

You'll notice right off that the display changes to show you what's going on. The status window will tell you what MatterControl thinks the print time will be as well as a progress bar along the bottom that gives you the actual percentage completed. (0.7% in the example below.)

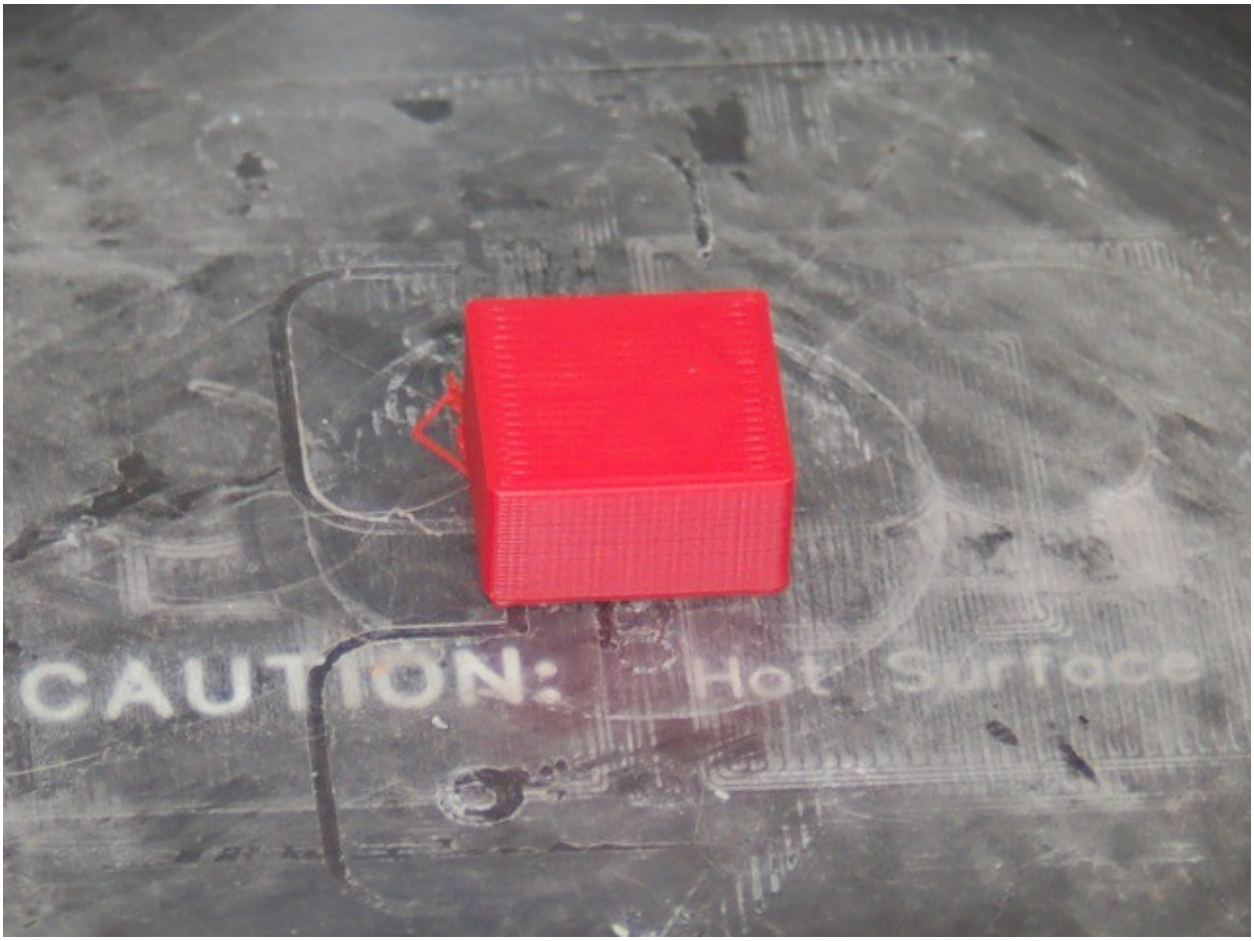
Your Orion LCD will show you that the hot end and heated bed are coming up to temperature.

If you're printing with PLA, then your temperatures will be different.

Kick back and relax! Once the Orion has it the temperature targets shown, the print will start!

I should note that the print time estimation is very, VERY optimistic. :)

After the print job is done, you'll have a little plastic cube that looks something like the photo below. I suspect your bed glass is going to be a lot cleaner than mine!



Congratulations on your first successful print! Let the bed cool to room temperature and you should be able to pop the cube right off the bed without any problems.

This has been a warp speed introduction to MatterControl. In the next section, I'll cover MatterControl in some detail and we'll print one of the files included on the SD card that was shipped with your new Orion.

Using MatterControl

MatterControl is a very complete 3D printing package and it's got a LOT of options. Some people can find this intimidating, but I assure you – there's nothing to be worried about!

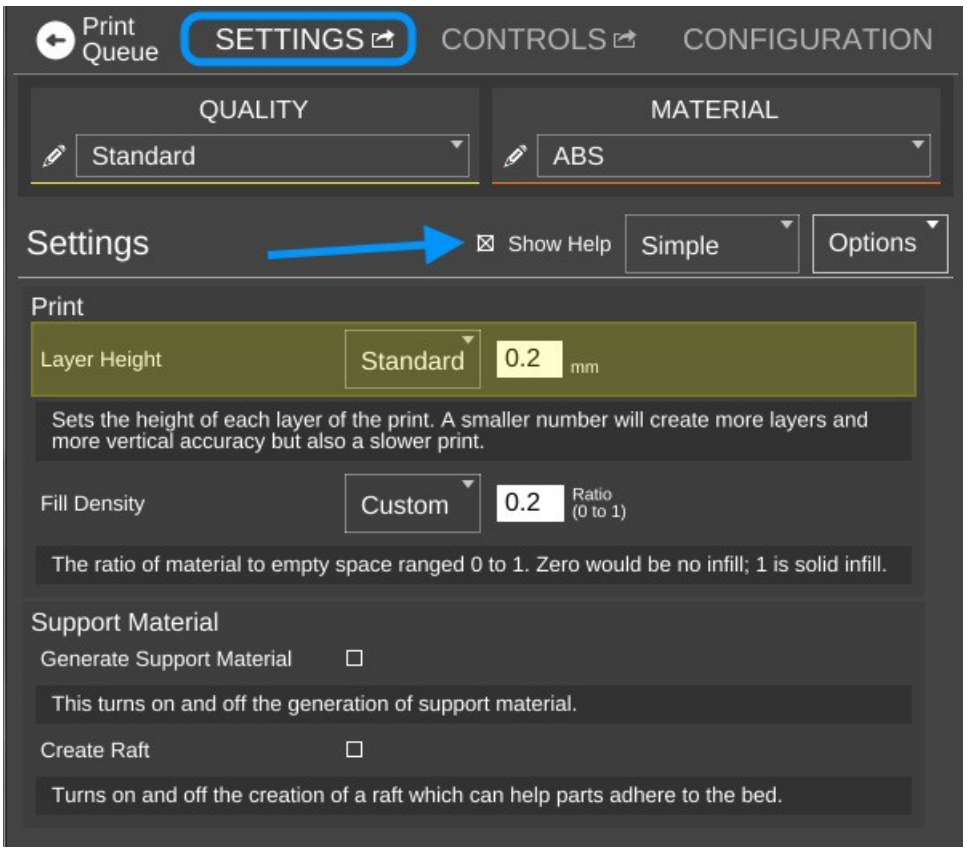
MatterControl is an integrated host application. This means that it provides everything needed to control the Orion and to prepare models for printing. The task of preparing a model for printing is called “slicing”. It's a very descriptive term for what is actually happening. In order to print a 3D model, it needs to be converted from a solid object into a series of very thin layers that are in turn converted into G-Code (more on this later). For example, if your print layer height is 0.2mm, the slicing tool is going to “slice” your model into a number of layers – basically the model height divided by 0.2mm. For a tall part, this can mean a LOT of layers!

MatterControl provides three slicers for your use. MatterSlice, CuraEngine, and Slic3r. This guide will only cover the specifics of MatterSlice, but don't let that stop you from experimenting with and using the other slicers! I'll show you how to change the slicing engine later on in this guide.

The final task of the slicer is to translate the sliced layers of model into something called G-code. G-code is a simple control language that's used to position the print head and tell the extruder how much plastic to deliver and at what rate. Going into the details of G-code is beyond the scope of this guide, but if you'd like to learn more you can check out the following resources: http://en.wikipidia.org/wiki/G-code and http://reprap.org/wiki/G-code.

For the most part, you'll never directly interact with G-code, but it's nice to know what's going on behind the curtain!

I want you to click on the **Advanced Controls** button to bring up the pane shown below. Click on the **SETTINGS** link to make sure your display follows (by and large) what you see below.

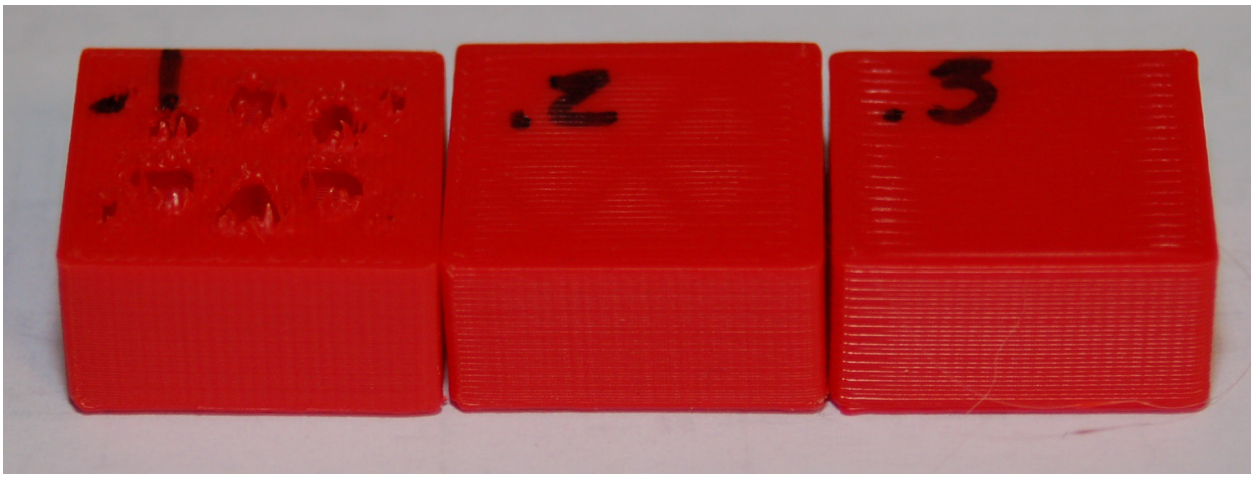


The first thing I want you to do is click on the **Show Help** check box that's highlighted by the arrow in the image above. This will turn on verbose descriptions of each one of the parameters available in the Settings page.

I'm going to only cover the “Simple” configuration settings for right now. There's a LOT that goes on to configuring your slicer and the Simple configuration setting allows many of those to be hidden until you're more comfortable with how your printer works.

Layer Height – This parameter tells the slicing engine how thin to make the layers when it slices up the model as I described earlier. A good default layer height is 0.2 or 0.25mm. The lowest practical layer height with a 0.5mm nozzle is 0.1mm. You can go lower than that, but it requires a smaller nozzle diameter. You can also got a lot thicker, but that requires a larger nozzle. If you change the Quality from **Standard** to **Coarse** or **Fine**, you'll notice how the layer height changes.

Here's what the **Coarse**, **Standard**, and **Fine** layer heights look like when printing the little test cube.



Starting from left to right, the layer heights are 0.1mm, 0.2mm and 0.3mm. You'll notice that the top layer on the 0.1mm print is kind of ratty and torn up. This is because the number of top layers is set to 3. This is perfectly ok with thicker layer heights, but it should have been set to at least 5 for the 0.1mm layer height that the **Fine** setting uses. You'll learn how to tweak that in a little bit.

You can see how the smoothness of the sides decrease as the layer thickness increases. If you want to print something really quick, you could go up to a 0.35mm layer height. I wouldn't recommend anything over 0.40mm if you're using a 0.5mm nozzle however.

Fill Density – This parameter controls how solid your printed part is. The number is a percentage, from 0 (totally hollow) to 1 (totally solid). The default fill density (also known as “infill”) is 0.2 or 20%. The image below shows what that looks like inside our little test cube.

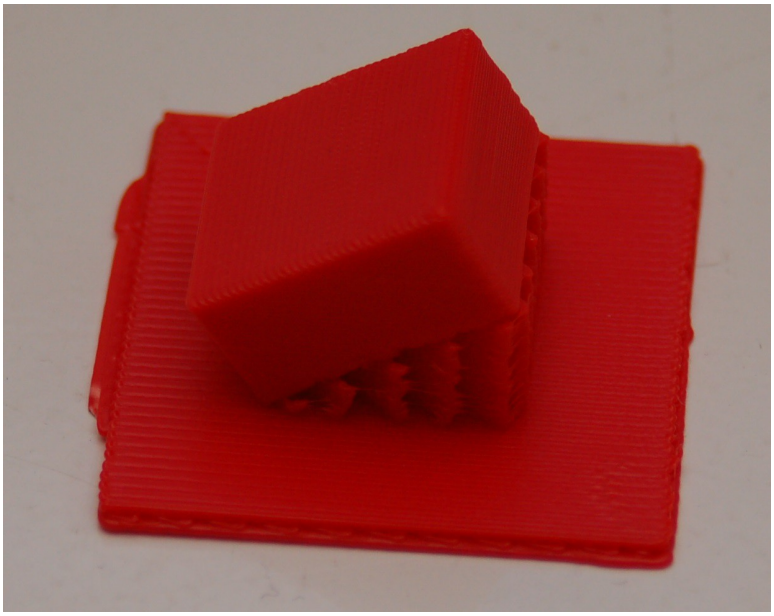
You can tweak the infill to get a more robust or a lighter part. For most prints, 20% is a good infill value.

Later on I'll show you how to change to a different infill pattern. The one shown on the right is the Triangle pattern.

Support Material – Support material is used when the part you're printing has free-standing features (like the chin on a bust) or another feature that requires it to be physically supported during the printing process. When you check the **Support Material** box, the slicer will automatically design support for the part that's currently (or will be) loaded.

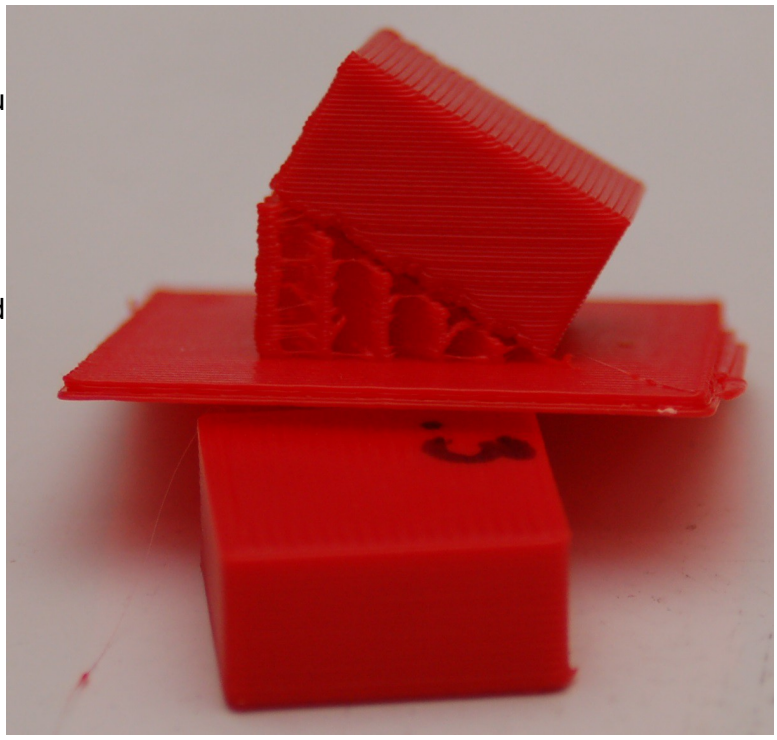
Create Raft – A raft is essentially what it says, a “raft” of material that your part will print on top of. Rafts are most often used when printing a part that is having bed adhesion problems due to its geometry. For example, if you're printing a part that sits on small feet, a raft would come in handy if the initial layers of the feet don't stick very well.

In the image below, you'll see an example of both support material and a raft.

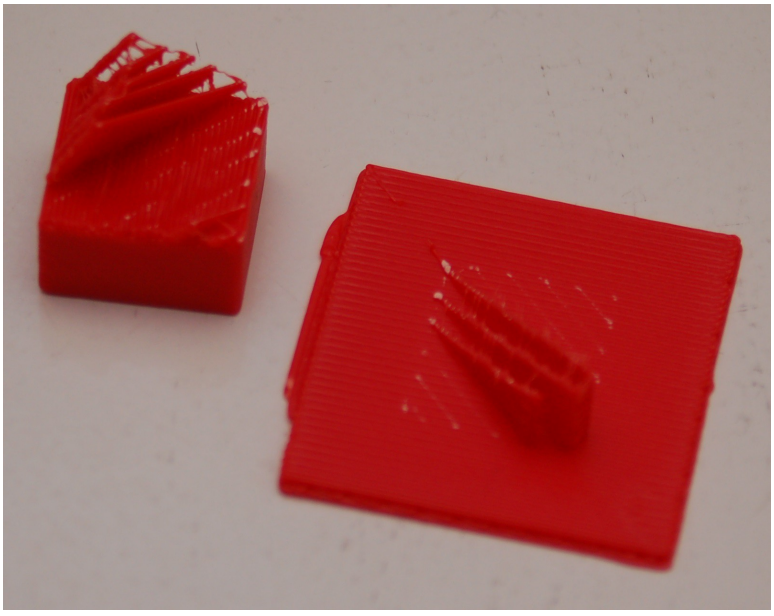


I took the little test cube and through a little manipulation, printed it tilted at 30 degrees. I took this opportunity to also demonstrate what a raft looks like. This one is exaggerated in its size, but gives a great example of what a typical raft will look like.

The image to the right should give you a pretty good idea of what the part looks like from the side. You can easily see the support material as well as the layer lines that will be at a 30 degree angle when the little cube is laid flat.

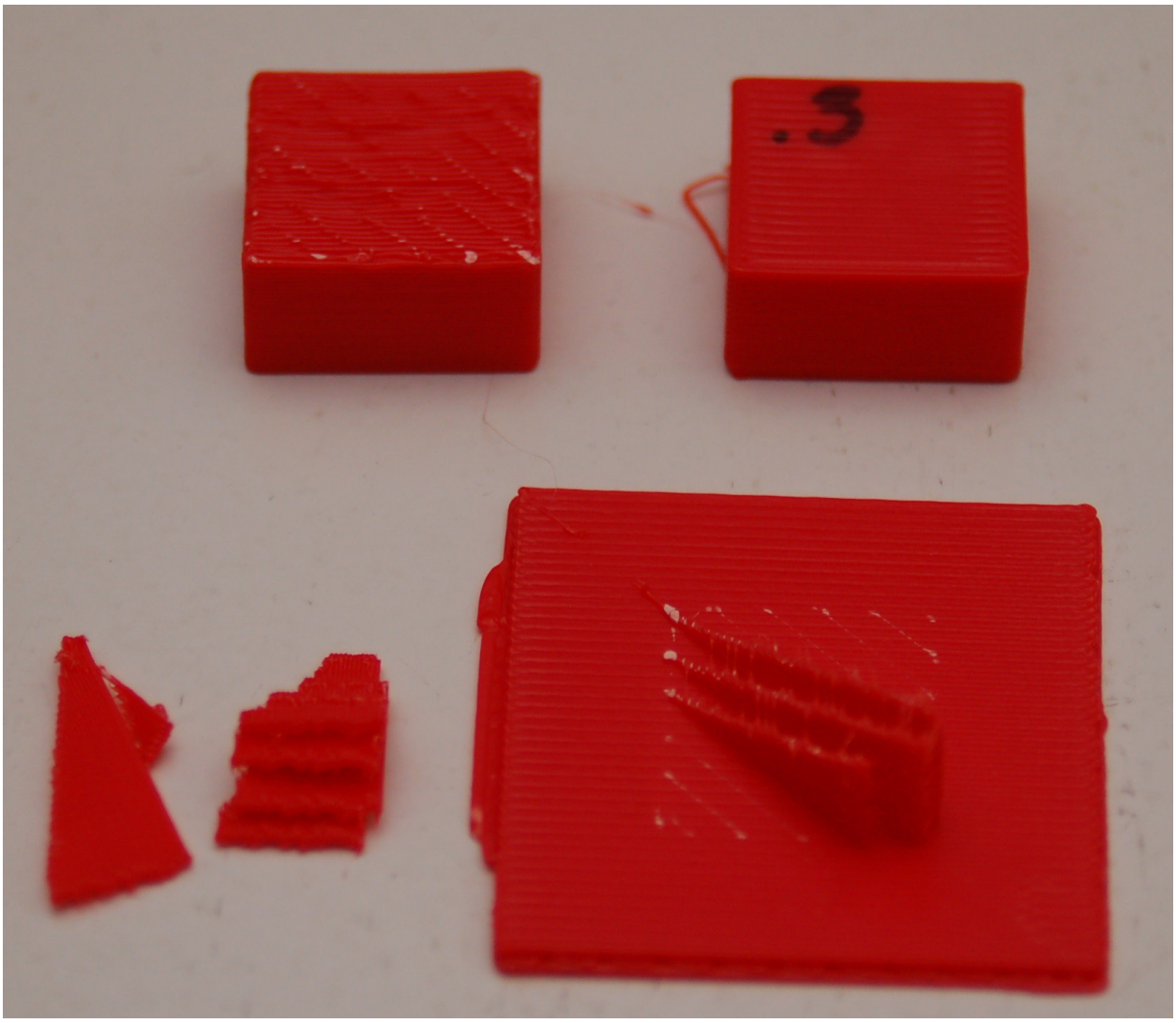


Support material is generated in such a way that there is just enough of it there to handle the actual print layer that it will be supporting. In my example, the support material has a 2.5mm spacing between the walls of support material and it uses an infill angle of 45 degrees.



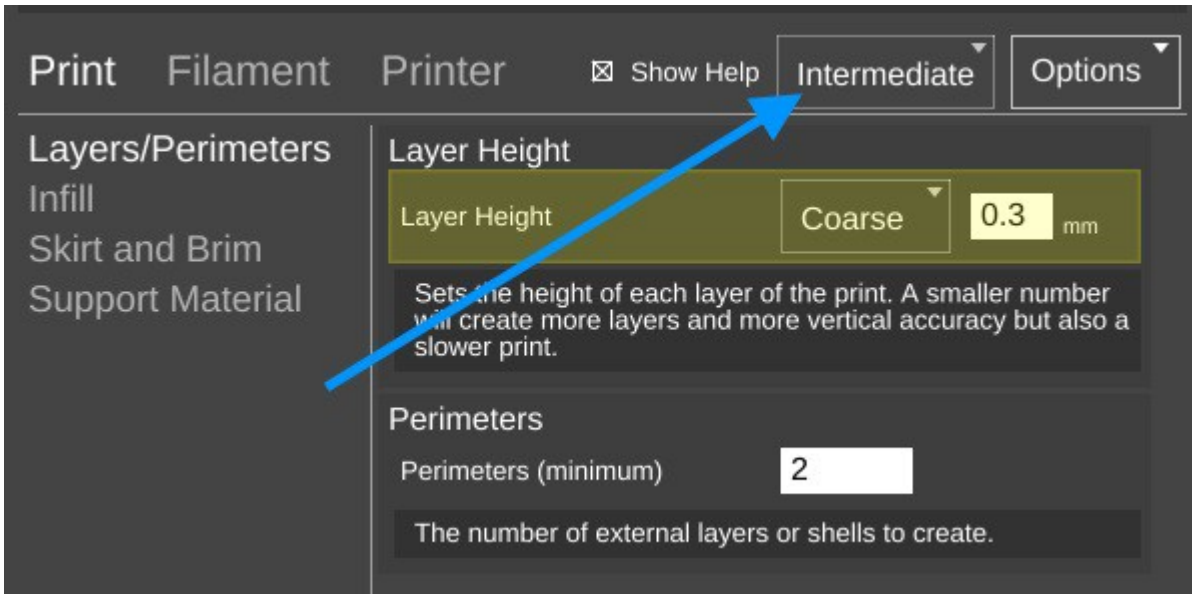
The printed part will usually separate from the support material fairly easily – however, some material will be left behind if you're using a raft. Cleaning up the left over support material is a simple and straightforward task.

Here's what you end up with after removing the support material from your part. As you can see, there's still a little clean up to to be done to the printed cube (on the left). A quick hit with some 220 grit sandpaper will knock the rough edges of the support material down.



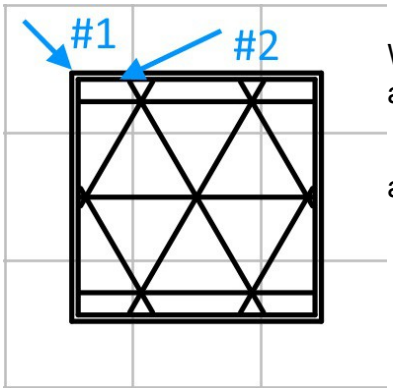
That's pretty much all there is to the **Simple** settings level. Next, let's dig into the **Intermediate** setting!

Click on the **Simple** setting drop down and pick **Intermediate**. Your MatterControl settings screen should change to something resembling the image below.



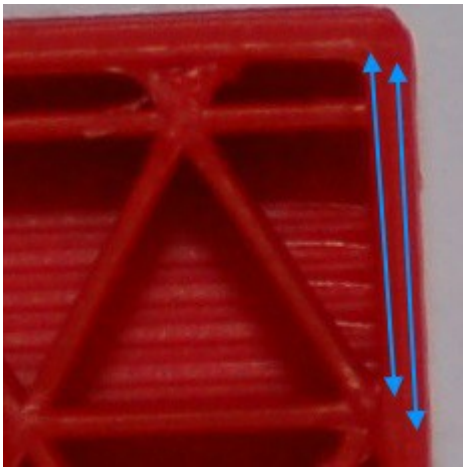
Quite a number of new configuration options are accessible under the **Intermediate** settings level. The first set of options we'll tackle live under the **Print** heading.

Layers/Perimeters – This setting page allows you tweak the layer height just as before when in **Simple** mode, but now adds the **Perimeters** setting. This setting is often called “shells” because it controls how thick the “skin” of your printed object is going to be. The default is 2 perimeters.

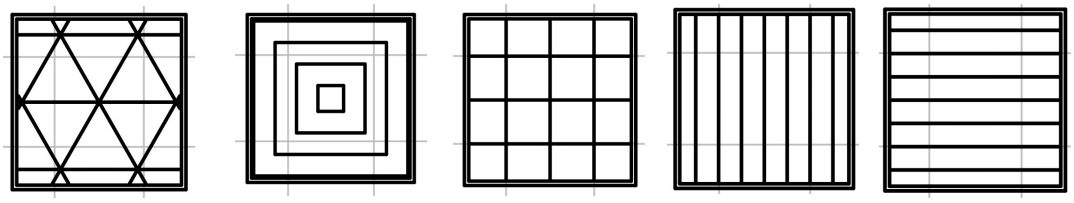


On the left is an example of what two perimeters look like. When the cube is being printed, the #2 perimeter is printed first and then the #1 perimeter. (The order is configurable.)

In the image below you can see what those perimeters actually look like on the cube we printed earlier.



Infill - The **Intermediate** level adds **Infill Type** in addition to the **Fill Density** figure we covered before. The infill types available are **Triangles**, **Concentric**, **Grid**, and **Lines**. Examples are shown below.



Triangles *Concentric* *Grid* *Lines #1* *Lines #2*

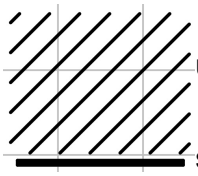
The **Lines** infill pattern differ from the others in that the line orientation is alternated every other layer. All the examples show a 20% infill density.

You're probably wondering which infill pattern is “best”. I wish I could go into that, but I've been unable to locate any studies that cover the topic in any depth. If I were asked to provide a recommendation for a good structural pattern I would probably pick the **Triangles** option. It offers a good internal structure for most infill densities that I've used it with.

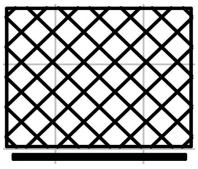
Skirt and Brim – This is a new option that appears with the **Intermediate** and **Advanced** setting levels. The **Skirt** option is used one of two ways. First, it can be used to “prime” the hot end with filament before the actual part itself begins to print. You may notice that your hot end may “drool” filament while the bed is heating up and the hot end has already reached temperature. This is perfectly normal. However, without some kind of priming action, early features of your part may not print properly. The **Skirt** solves this. Secondly, the **Skirt** can become a **Brim** if the **Distance from Object** setting is set to zero. What this does is make sure that the skirt is physically connects to the part, becoming a brim. This can be handy when you're printing a small part and you're having bed adhesion issues and you don't want to have to use a raft. Later on in the **Advanced** settings section, you'll see more options on how you can tweak the **Skirt and Brim** settings.

Support Material – With the **Intermediate** and **Advanced** setting levels, you get more control over how the support material for your part is generated.

The new option here is called **Support Type** and allows you to choose a **LINES** or **GRID** pattern for the support material. When I printed the support example, I used the **LINES** mode as that is the default for the **Simple** settings mode. Looking down on it from above, this is what the **LINES** support material pattern looked like for that print.



This shows the second layer of the print. The lines will continue to stack upon one another for the entirety of the support structure.



The **GRID** pattern (on the right) uses the same spacing as the **LINES** option, but is designed to provide more support where it may help to provide a better end result.

The next new category exposed by the **Intermediate** setting is called **Filament**. Under the **Intermediate** setting, the only option is **Filament**. This allows you to set three parameters that deal directly with the material you're currently using to print with.

Diameter – This is the diameter of the filament you're using. The more accurate this figure is, the better the quality of your prints. This is because the slicer uses the filament diameter to help calculate the optimum flow rate for the extruder during the print.

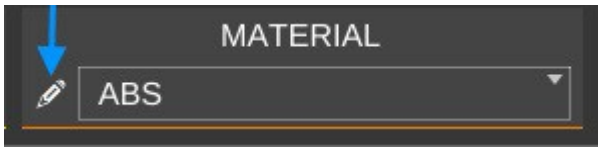
In order to get an accurate filament diameter, spool off a meter or so of filament and check it in 10 spots along the length of the material. Record the measurements using a digital micrometer and average the results. That average should go into the **Diameter** field.

Extruder Temperature – This figure determines the target temperature of the hot end for the material you're printing with. A typical heat range for ABS is 220 to 240C and 190 to 220C for PLA. Other materials will have their own recommended temperature ranges. ***NEVER, EVER, EXCEED 245C WITH THE STOCK HOT END THAT'S SHIPPED WITH THE ORION!***

The reason for this is because of how the stock hot end is designed. It uses a PEEK section (a high-temperature plastic) as the “cold end” of the hot end. This material will begin to fail at 247C. If you need to print with a high-temperature filament such as Nylon, it's highly recommended that you purchase an all-metal hot end.

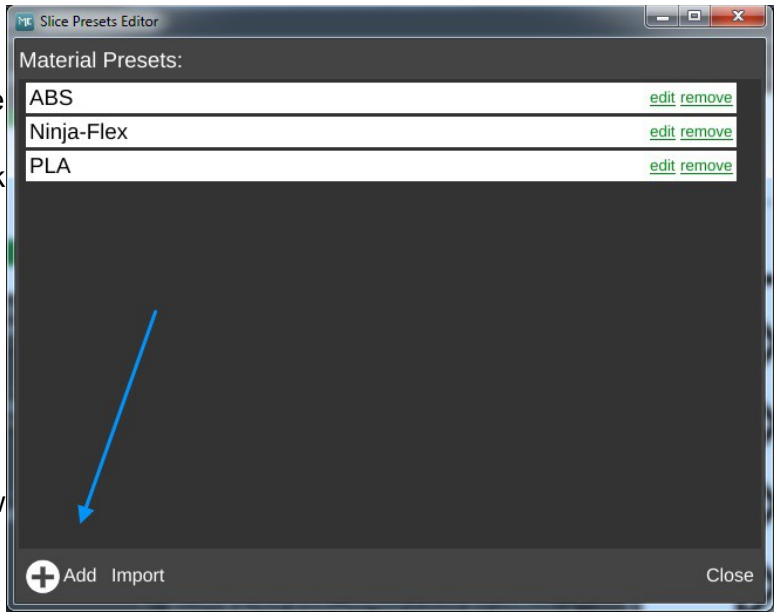
Bed Temperature – Like the **Extruder Temperature** the bed temperature is material-dependent. For ABS, a typical heated bed temperature range is between 80 and 100C. For PLA the range is typically 55 to 65C.

Each filament can have different heating requirements, even within the same type and color! For example, it's not unusual to have two rolls of identical material require different hot end settings. Bed temperatures tend to be less variable. When you're working with a new roll of filament, I recommend printing a test object or two in order to find out what the best temperature setting works best with that material. Note the settings on the spool label, or add your own. This presents a nice opportunity to explain another feature of MatterControl – material profiles! You'll notice a little pencil icon next to the material drop down

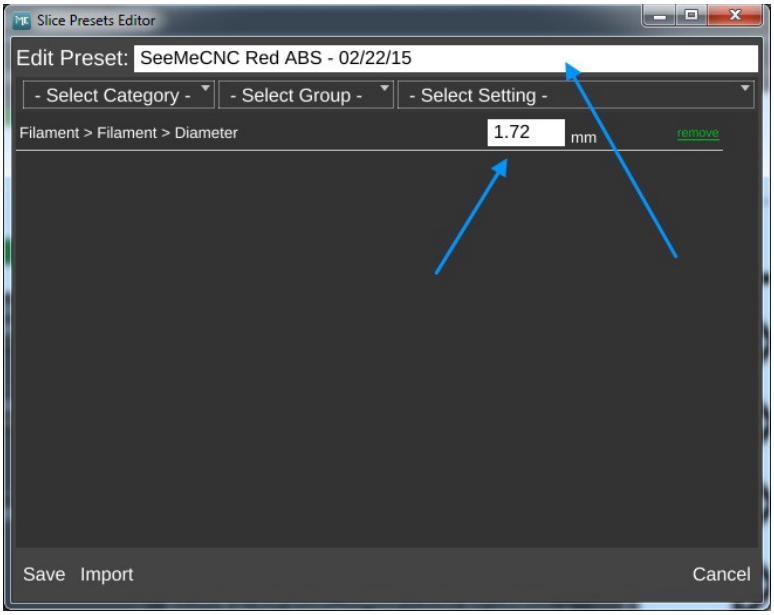


Click on that to print up the preset manager.

The **Material Presets** list shows you what pre-configured material settings you've got available to you. When you get a new roll of filament in, you can easily keep track of its settings by using this system. Click on the **Add** button and we'll create your first custom material!



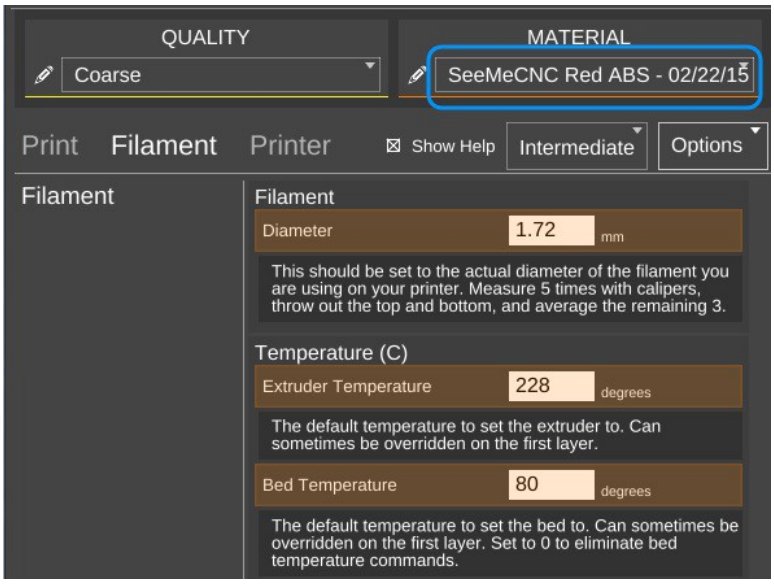
When you click on the **Add** button, you'll be presented with a screen that looks something like the one shown below. The **Edit Preset:** field is where you can name this new material configuration. In this example, I've named it after the vendor, the material color and the date I purchased the material. Since this is the first time I've used this material, the only thing I know for sure is what the average filament diameter is. To enter that, I picked **Filament** from the **Select Category** drop down, then from the **Select Group** drop down, **Filament** was chosen again and finally **Diameter** was selected from the **Select Setting** drop down.



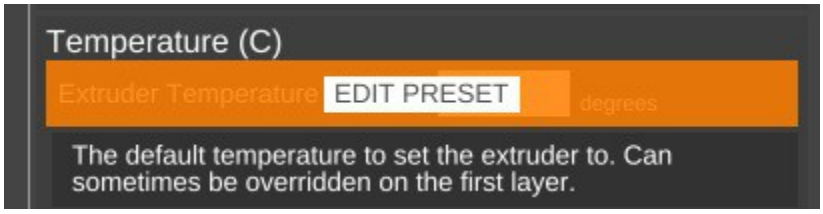
I then entered the filament diameter that I calculated using the process I outlined to you earlier.

Click on the **Save** button to commit your changes. Congrats, you've added your first custom material profile!

After you've saved your new profile, it will appear as the currently selected material as shown to the right.

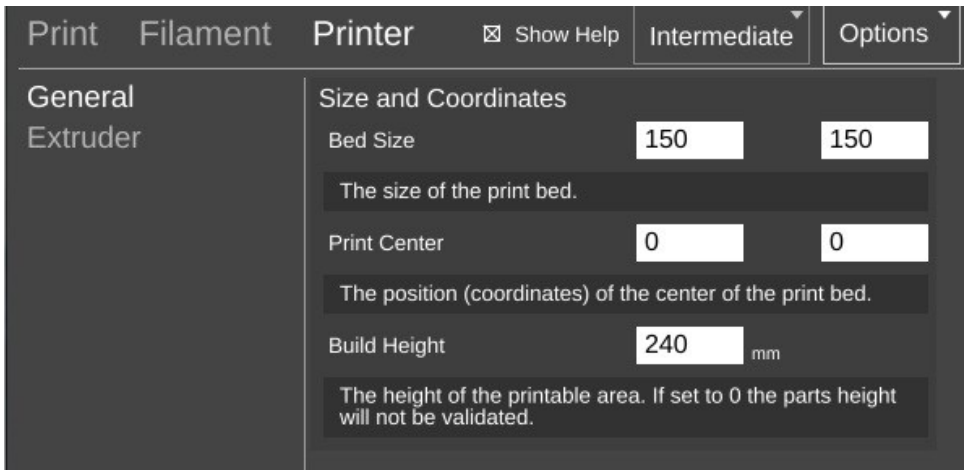


Now say you've printed a test cube and have decided that the print might look better if you bumped up the temperature 2 degrees. This is a simple change to make. If you hover your mouse over the **Extruder Temperature** option, you'll see that the foreground is covered by an **EDIT PRESET** button as shown below.



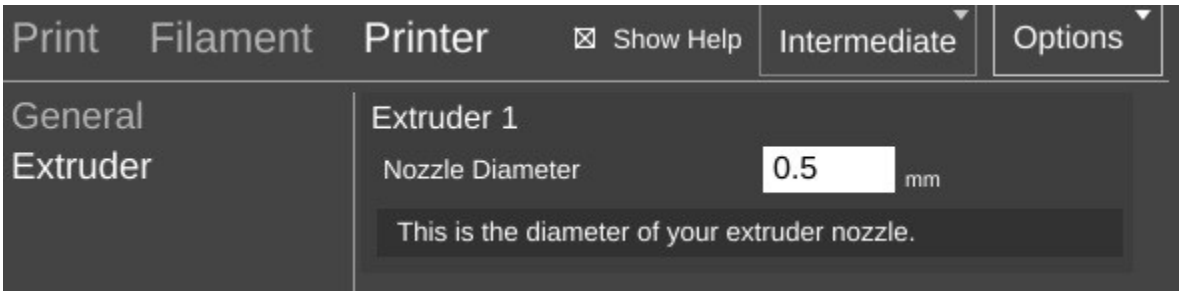
Clicking on the button will open up the materials editor and allow you to change the temperature. It's as simple as that!

The last new category exposed by the **Intermediate** mode is called **Printer**. This category deals with the physical properties of the printer itself. Because you chose the specific printer model you have when you first set up MatterControl, all the basic parameters have been filled in for you.



The figures here are pretty self-explanatory. You're unlikely to need to change the parameters on this option page, so let's go to **Extruder** and see what that's all about!

One of the nice features of the hot end that the Orion uses is the ability to change the nozzle that you're using for a print. The stock hot end size is 0.5mm, but if you're going to do really detailed work, you may want to change to a 0.35mm nozzle. If you're doing coarser work and want to print really quickly, you could use a 0.7mm or larger nozzle. In order for the slicer to know how to properly calculate the material flow rate, it needs to know what the diameter of the nozzle is. It goes hand in hand with the material diameter setting that we covered earlier.



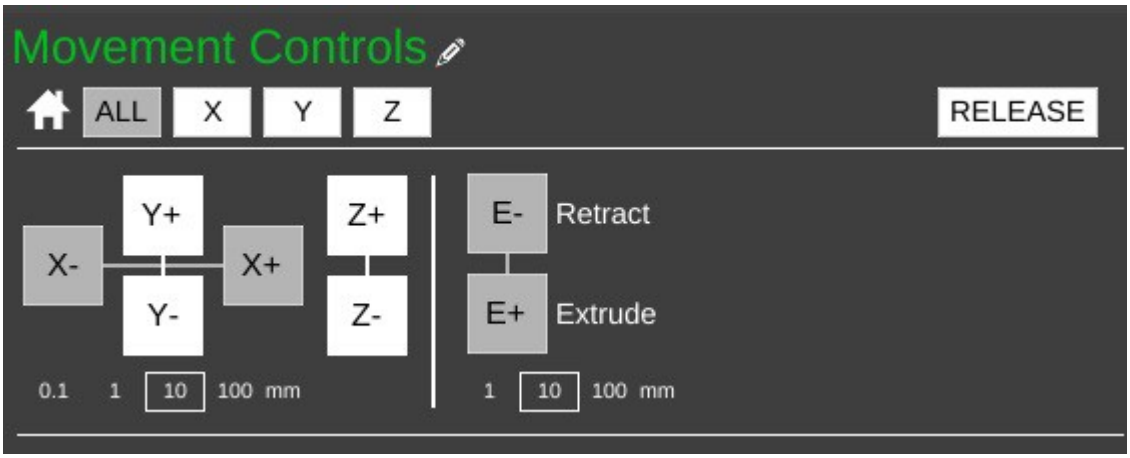
Before we move on to the next topic, I wanted to cover one more item – the **QUALITY** presets. I'm not going to go in depth on this one except to note that both the **QUALITY** and **MATERIAL** presets can store identical settings. They're split into two categories in order to make it easier for you to group common settings together. You will typically store things that directly affect the print quality of your model in the **QUALITY** presets. Things like layer height, print speed, etc. **MATERIAL** presets should only contain those settings that directly deal with the material you're using to print the current object with. You'll find that both the **QUALITY** and **MATERIAL** presets contain more options than I've covered here. Those new options are exposed by choosing the **Advanced** settings level. I'll be reviewing those with you later on.

We've covered both the **Simple** and **Intermediate** settings for the slicing engine. **Advanced** is something I'll cover later, so let's move on to the **CONTROLS** section.

The **CONTROLS** page is where you can manually control your Orion 3D printer. You can heat the hot end or bed, as well as manually position the effector platform and extrude plastic.

The **Temperature** pane contains everything you need to manually control the temperatures for both the hot end and the heated bed. MatterControl provides PLA and ABS presets. You can edit them by clicking the pencil icons. You can also enter in a temperature and heat to that value by clicking on the **SET** button that will appear as soon as you begin typing.

The **Movement Controls** pane contains controls that will allow you to manually position the Orion's effector platform.



The row of buttons to the right of the little house icon control “homing” of the movement axes in the Orion. Because the Orion is a delta configuration printer, the only buttons active are the **ALL** and **Z** buttons – they perform the same action. Connect your printer if you haven't already and click on one of them to see what I mean. The printer will home itself and await further instructions. (Good robot! Have a Scooby Snack!) The **RELEASE** button will tell the Orion to power down the stepper motors so that the axes can be moved by hand. This is handy when you want to loadnew filament into the printer without having to turn it off first.

The axis motion is controlled by the X, Y and Z labeled + and – buttons shown above. Below those four buttons are selectors indicating the step distance from 0.1mm to 100mm. The selected axis will move the selected step distance with each mouse click. For this reason, please take special care when you've got 100mm set for the step distance. The Orion is smart, but not THAT smart. It relies upon you to not put the poor thing in an unlikely position. :)

The last set of buttons control the extruder motor – they're marked **E-** and **E+** and can be used to manually extrude filament. Note that they will only work if the hot end is up to operating temperature! The amount of filament extruded (or retracted) is set using the step selector below the control buttons for the extruder.

The **Fan Controls** allow you to manually control the layer fan on your Orion *if you have one installed.* The fan control will not control the PEEK fan as that is required to be on for the duration of the print job. You can turn the fan on 100% by clicking the control, your you can enter in a percentage value to set it to a speed lower than full-on. I'll go over the use of the layer fan when the **Advanced** settings are covered.

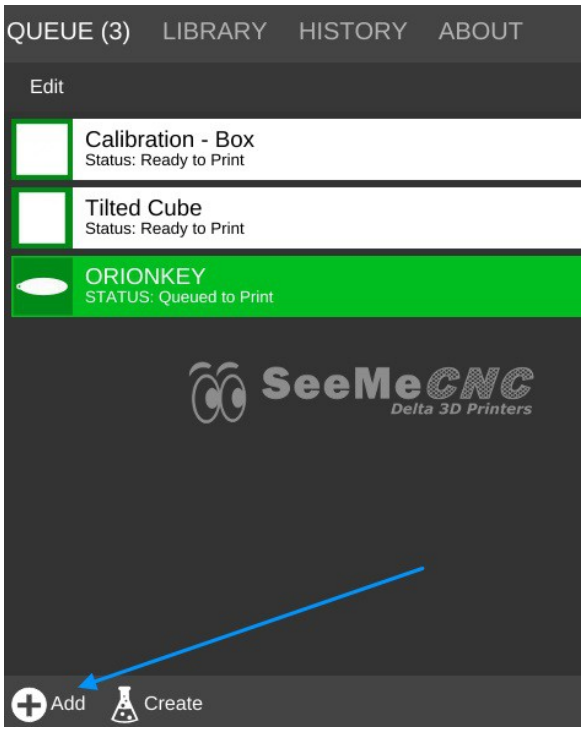
Loading and Printing an Object

We've previously worked with the small cube that MatterControl provided as an example. Now we're going to cover loading and slicing an object from start to finish.

For this section, I recommend you head over to http://www.repables.com and find something you'd like to print. I'm going to chose the Orion Key Chain (http://www.repables.com/r/151/) for my example print. You don't have to make the same choice, but pick something geometrically “simple” in order to make the learning process a bit easier.

Most (if not all) 3D printers can read a file format called “STL”. (You can learn more about this file format, including its origins, here: http://en.wikipedia.org/wiki/STL_%28file_format%29) When you download a file from Repables or one of the other free, online object repositories, you'll often get the file as a zip file. One nice feature of MatterControl is the ability to select a zip file and MatterControl will transparently extract all the files it knows how to read and load them up into your print queue.

To load a file into MatterControl, make sure you're on the **Print Queue** page and click the **Add** button at the bottom right hand corner of the window.

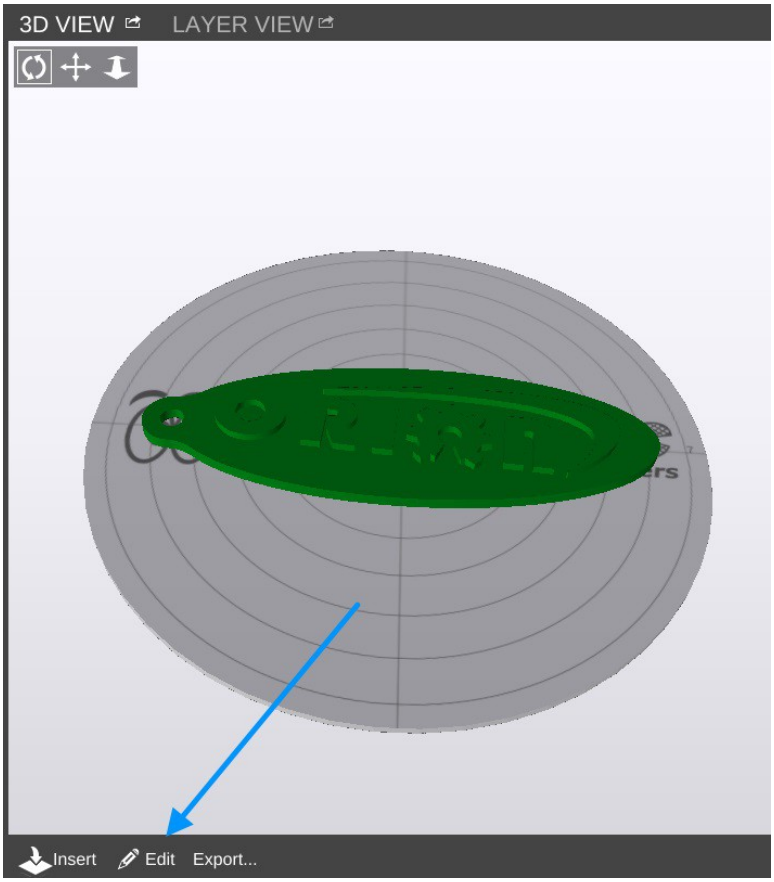


Navigate to where you've stored the STL or ZIP file and open it using the Open File dialog that will appear.

Once you have the object loaded, click on the **Advanced Controls** button so we can make sure your print settings are the way you want them.

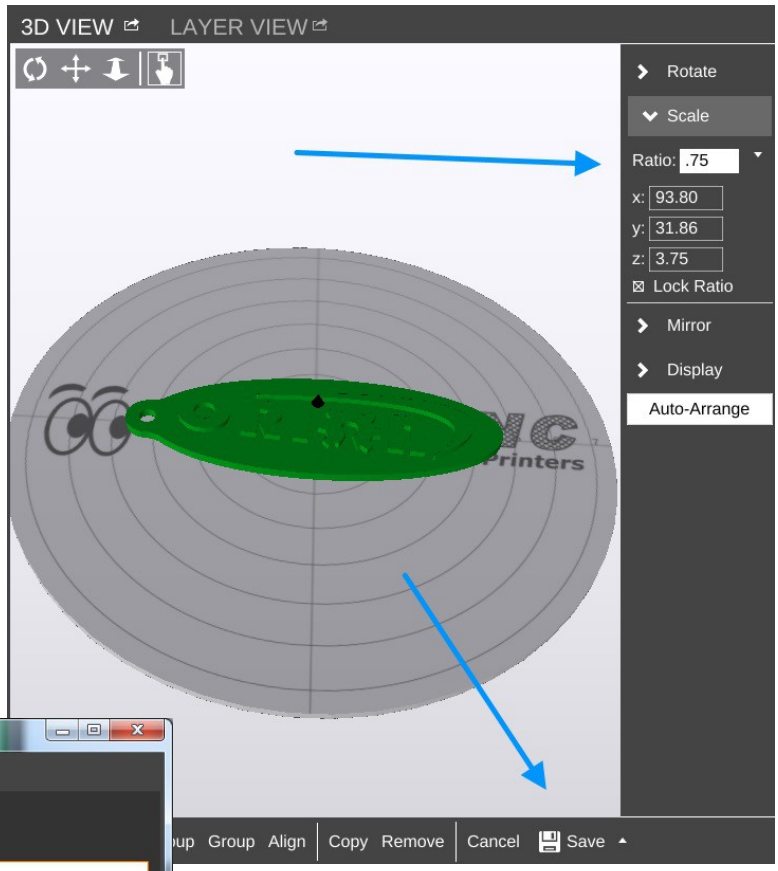
For my print, I've decided to leave the **QUALITY** setting at **Coarse** and I'm using the tweaked material values that I set up earlier.

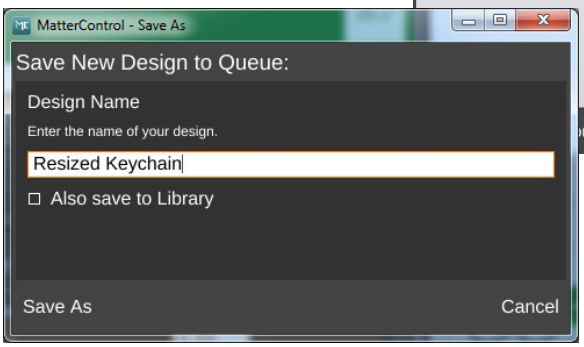
Because the key chain is pretty large, I'm going to scale it down to 75% of it's original size. This is an easy task in MatterControl.



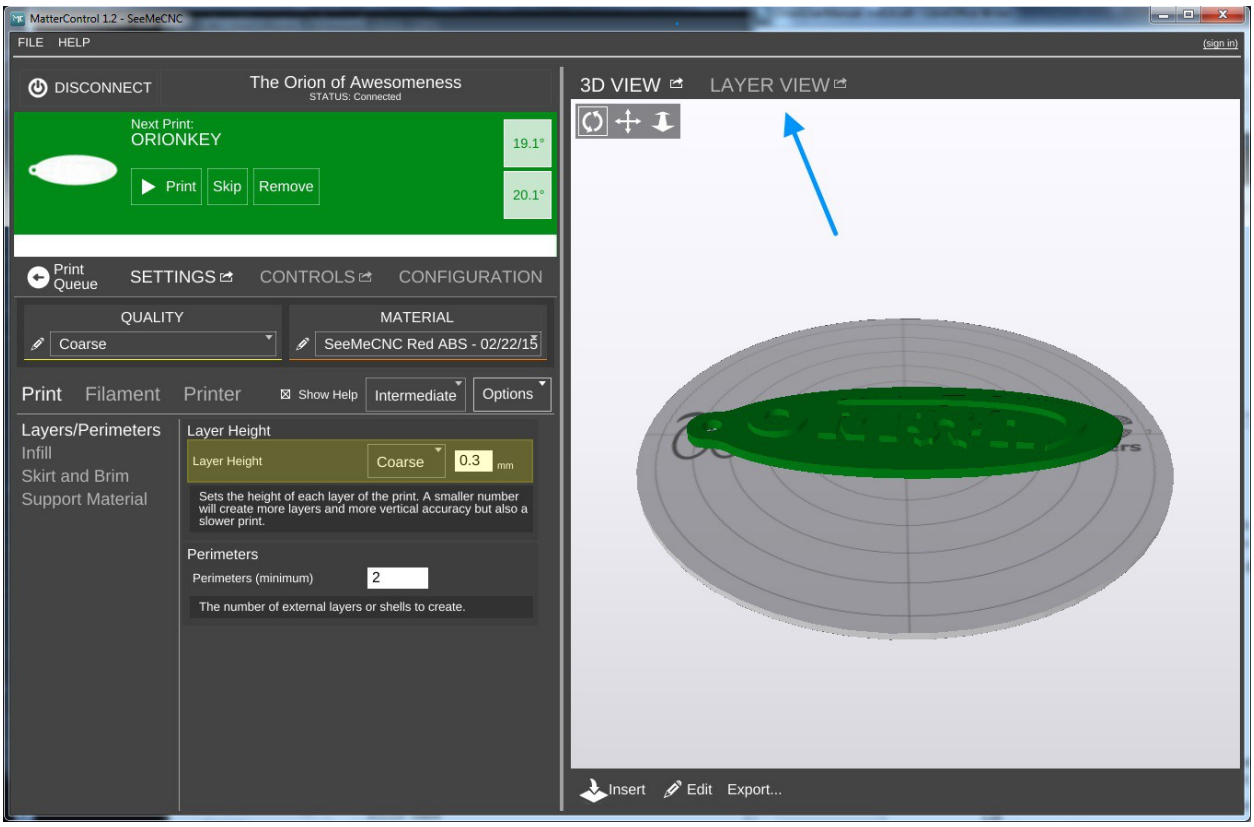
Click on the **Edit** button as shown above. This will modify the 3D View so the edit controls are visible.

Click on the **Scale** button to access the scaling controls. As you can see, I changed the value to .75 or 75% of its original size. Click on the small up arrow next to the **Save** button at the bottom of the window. This will allow you to save the object under a different name if you don't want to over-write the original.

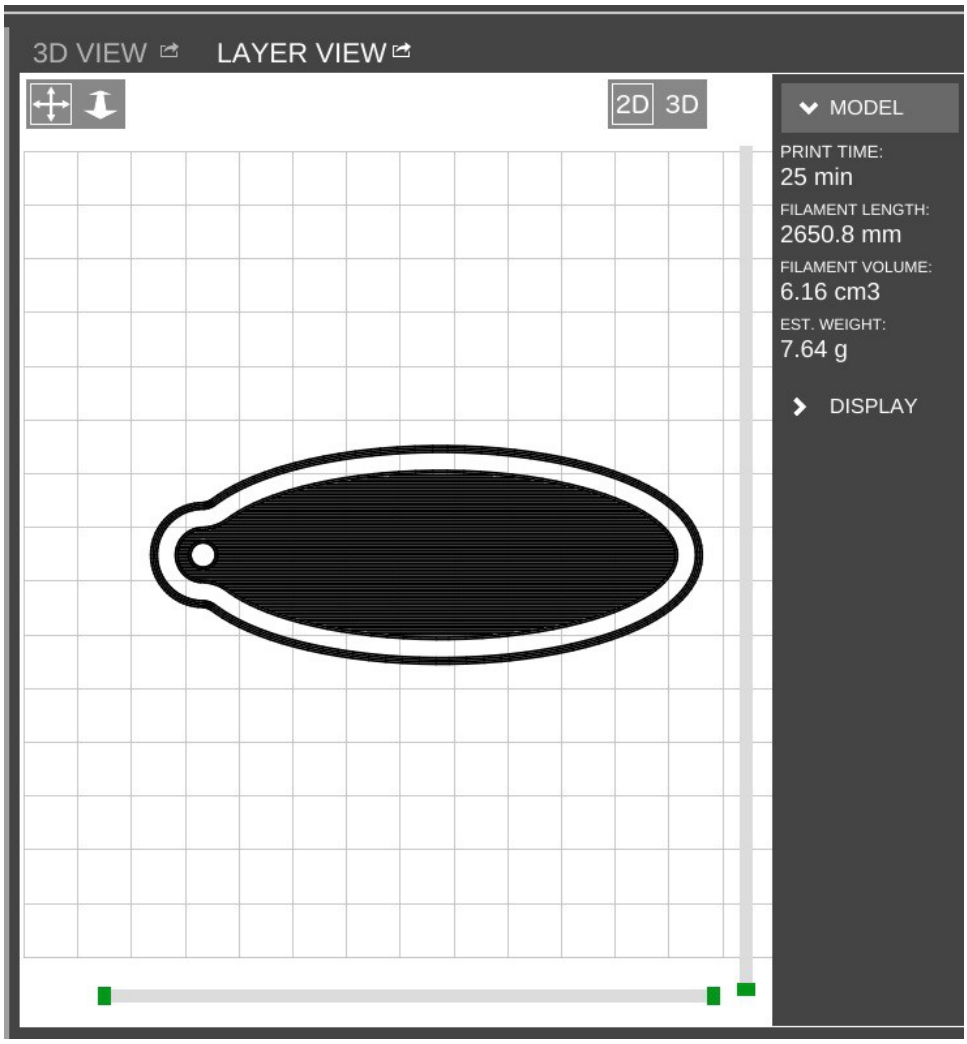




Before we start this print, let's take a second to examine a feature of MatterControl – the **Layer View**. If you've not sliced this object yet, you'll like see the text “Press 'generate' to view layers”. Go ahead and do that now.



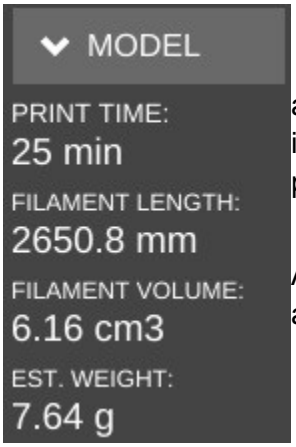
When it finishes, the layer view will display the first layer of your print job.



You'll notice right off the skirt that I covered previously. It's important to make sure that the hot end is primed by the time it begins to print your part!

At the bottom of the window you'll see some controls that will allow you to either re-slice the object (**Generate**) or view the individual print layers.

The controls will show you how many layers are on this object as well as what the layer number is that you're currently viewing. You can navigate forward and backward through the layers by using the **>>** and **<<** buttons. If you want to jump to a specific layer, you can enter it in to the box and click **Go**.



After slicing the object, MatterControl will display a few statistics about the current print in the Layer View window. This can be handy information if you're selling your services and need to know how much a particular part is going to consume in both time and materials.

Go ahead and click the **Print** button and get your object printing! After it's done we can cover moving objects around on the build surface and printing more than one object at a time.