

ISC 226 Ultimaker 2+ User Guide

Be familiar with this guide before using the 3D printer in ISC 226

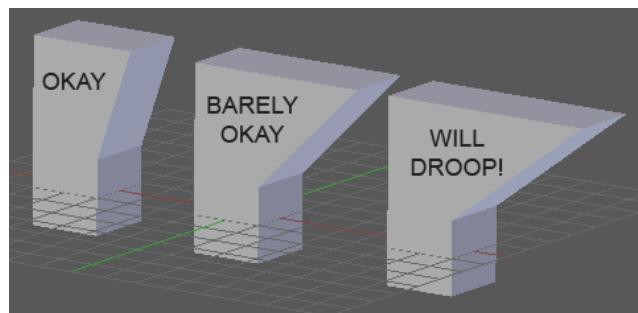
1. Conceiving an Idea

The printer provided in ISC 226 for “academic physics” purposes. You cannot print toy spaceships with it. Your print jobs should be related to coursework (specifically, *Instrumentation (LabVIEW)*, *Intermediate (Junior) Lab*, or *Digital Electronics*), or to a research project with any Physics Professor. You should have instructor approval from your professor before using the printer. Nobody may use the printer until they are certified by Dr. Pogo. Certification will require you to pass two tests (similar to how you get a driver’s license). The tests will not be trivial.

Reading this guide should give you a good understanding of what the 3D printer can and cannot do, and you must account for these capabilities while *initially imagining* your part - not when trying to print it. Before modeling your 3D Model with any software, conceptualize your model on paper with a pencil. Sketch something. Since it’s probably flawed in many ways, get another sheet of paper and sketch it again, but better this time. The first version is never any good. Before you even reach the software stage, you should have the major shapes and dimensions already figured out.

Most 3D printed parts will need to connect with other parts, whether those others are 3D printed or not. You must determine how they will all connect before printing any part, and you must be sure that your connections are properly spaced and sized. When interfacing with pre-existing parts, you can often determine the actual dimensions from the manufacturer. In rarer cases, you can use a caliper to determine the needed dimensions. It is more difficult than you think to determine the spacing between two holes, because the center of the hole has no “edge” for you to align your caliper. When designing interfaces, keep in mind that the accuracy of a 3D printed dimension is typically ± 0.2 mm. So, if you need to make a peg fit into a hole, you usually need to deliberately make the hole a little bigger or the peg a little smaller.

As you conceptualize your design, you should have the capabilities of the 3D printer in mind. Avoid large overhangs of over 45 degrees and unsupported inclines (which will droop), extremely fine features (which won’t be printed at all), parts with thin segments (which will snap), and parts without large enough bases to support them while printing (which will fail to even print in a variety of spectacular ways). If you’re paying attention while reading this, you should be thinking about whether it’s possible to print a “T” shape. If your design requires a “T” shape, then you *can* print it... but it will need to be printed either upside down or laying down. In other words, while printing, many parts will be in a *different orientation* than how they will be used in the final design. You should be mentally planning the print orientation *while* conceptualizing your ideas. If no good orientation exists, you should print multiple pieces that you’ll assemble later. Superglue works pretty well on these parts.



From <http://www.3dprintingera.com>

Holes with horizontal axes have a similar problem, but not to the same extent. For larger holes that *really* need to be circular, you can add “support structure” (see page 5) to temporarily hold the upper edge of the hole while printing.

Each printed part should have the minimum possible mass while still being able to do its job. Usually, this means making your design smaller *in every way*. Once the outer dimensions are minimized, you can often still remove even more mass by placing various holes in your part. You may not print any object requiring more than 6 hours without Dr. Pogo’s prior approval.

Another design issue to watch our for is “likelihood to break”. Some portions of your object will be subjected to higher forces, and will need to be larger. Also, try to avoid strict rectangular holes, which are made from four “sharp” *internal* corners. These magnify likelihood of failure by a factor of 2 or 3. Instead, redesign the part so that you can have *rounded* internal corners.

2. Modeling in Software

Once you have a fairly complete concept on paper, you can start to model your part in software. The three main programs used by students for 3D printing are *Fusion 360*, *Autodesk Inventor*, and *SketchUp*. All are free for students. All have substantial online tools to help you learn to use them.

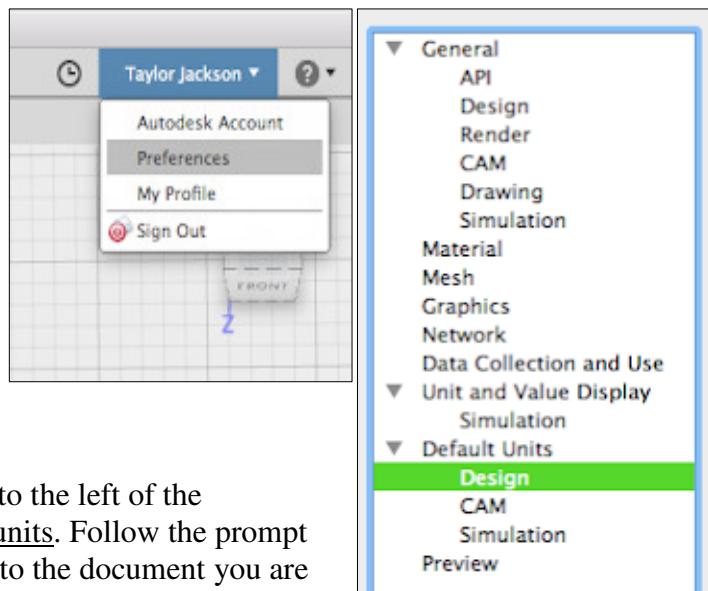
You will generally make *three different* software versions of your model. The first is the actual design, which is saved in one of dozens of formats depending on the software you have chosen. The second is an exported version that can be opened in Cura (a 3D printing processor called a “slicer”). This version is usually an .STL file. The final version is saved by Cura as a “G-code” file that will actually be given to the printer.

Units

The printer expects that your dimensions are all in millimeters.

To permanently change the default unit setting in **Fusion 360**: Click on your name in the upper right hand corner and select preferences. When the drop down bar appears, click on Default units and check or change the units in all three categories to mm.

For “quick” unit changes click on the triangle to the left of the “document settings” and locate the tab called units. Follow the prompt to change your units. Note this change is only to the document you are **currently** working on, if you mistakenly haven’t already set the default units to millimeters.



To permanently change the unit setting in **SketchUp** (<https://app.sketchup.com/app?hl=en>):

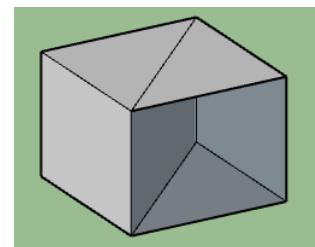
Use the side panel on the right. Near the bottom, select the “model info” option ⓘ. Expand “length units”. Set the “format” to mm, set the display precision to 0.000 mm, and turn the length snapping “off” (i.e., slide it to the right).

To permanently change the unit setting in **Autodesk Inventor**: In Inventor, the procedure for changing the units is part of the exporting process, rather than the design process. See the directions for exporting below.

Comment on Using Sketchup

Sketchup is very user friendly, but it has some quirks that you need to understand at the beginning of the design process. In Sketchup, the parts you design are all hollow... there is no “stuff” inside the skin. You draw the skin, and CURA will make it “substantial” during the next step of the design. However, for each piece of skin (called a “face”), Sketchup will assign an “inside” and an “outside”. Every “outside” face must actually be facing outwards from the body, and similarly every “inside” face must face inwards. “Outside” faces are shown as shades ranging from white to gray. “Inside” faces are shades ranging from light blue to dark blue. Depending on the lighting, the dark gray and dark blue can be hard to distinguish. If even a single face is incorrectly oriented, your 3D print could fail. Therefore, as a corollary, if you have an *extraneous* face on the interior of the part, it will not print correctly, because you can’t set both faces to be “inside” faces.

For example, consider this *cube* made by joining two triangular prisms. I’ve removed the front face so you can see that the interior has an “extra” diagonal face along the plane joining the original prisms. The mere existence of this plane guarantees a bad print will result, because both faces need to be “internal”, but one of them will be “external”. So, review your design carefully for hidden internal faces before exporting it.



Threaded Holes

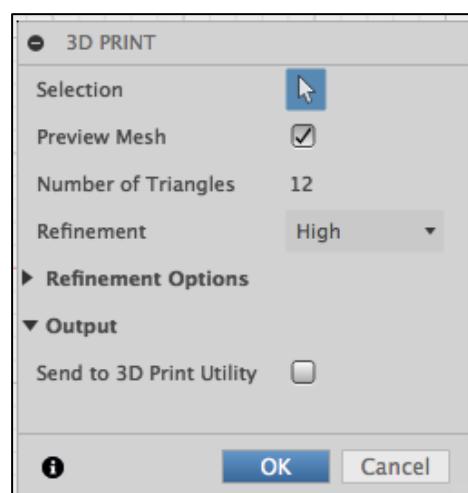
In Sketchup, drawing threads is very tedious. It is also hard in Fusion. In many cases, if your thread is “small”, you can just make a hole and drive a metal bolt through the hole, which will dig out some of the skin, effectively threading it. This is great for set screws, but not larger screws.

3. Exporting Your Design

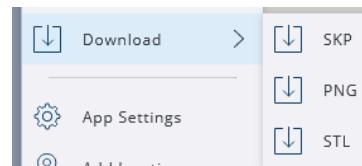
You must “export” your design to an STL file once it is finalized.

Fusion 360: Click on “Make” from the ribbon. Choosing “3D Print” creates a small dialog on the right (seen here).

Then click on your object so that it is highlighted in blue and click ok, then save the file as either an STL file.



SketchUp: By default, SketchUp saves everything to the cloud. But, you'll need a copy on your own computer! Click on the "hamburger" on the upper left corner ("≡"), then go to "download", then "STL".



Autodesk Inventor: Choose File/Export. In the resulting menu, choose to "Export as CAD Format". A dialog will appear directing you to save the file. Make sure it is saved as an STL file. As with Sketchup, there is an "Options" button at the bottom that you must use to make sure that the resulting file has units of millimeters. Then click Save to create a file of the selected type.

4. Setting Print Properties in Cura

Cura should force you to pick an actual brand and model of 3D printer the first time you open it. You can change the specific printer using the "Settings/Printer" menu, or with the black bar in the upper right corner. The printer in the LabVIEW room is an "Ultimaker 2+". The "plus" matters.

You can drag your STL file directly into Cura, or use the "File/Open File..." menu. You can even place multiple disconnected objects onto the build plate! Once you have an object (or objects) in Cura, click on "layer view" near the top-center part of the screen. In layer view, you can separately see the various color-coded "slices" that Cura will make regarding your object. The most important layer is "skin", which is the outside part of the object. The next most important is "infill", which refers to structures or small cells placed inside the object to help hold the skin together. Infill should be between 8% and 20%, depending on the desired strength needed for the piece. "Top and Bottom" refers to special parts of the skin that are obviously on the top and bottom of the print. Cura allows you to separately control the thickness of the top and bottom compared to the rest of the skin. "Helpers" refers to extra material that Cura wants to print to make sure your object is properly printed (e.g., without sagging). These helpers must typically be removed from the object by hand after the print is done.

Printing happens in layers, rising upwards from the print bed. In Cura, just to the right of the (mainly yellow) object is a vertical sliders that can let you see how your object will print with a kind of x-ray vision: it shows the material up to the layer height chosen by this slider.

In Cura, the right mouse button rotates your view of the object. The mouse wheel controls zoom. If you hold down the mouse wheel while moving the mouse, you can slide your viewpoint (as opposed to rotating your viewpoint).



Click on the (mainly yellow) object, then on the left, choose the "Rotate" icon (slightly highlighted in the image on the right). This will create red, green, and blue handlebars on your object. Rotate the handlebars to place the object into the appropriate print orientation. Once rotated, Cura will automatically re-center the object in the print bed.

If your object will experience high stresses during its use, you should be aware that 3D parts are stronger in the direction that was horizontal during printing, as compared to the vertical direction. So, a support stick of some kind should be printed horizontally, not vertically.

Large holes in a vertical plane (i.e., with a horizontal axis direction) will *probably* print, but might sag somewhat during the printing process. The sagging can probably be remedied after the print using a small hand file, but that's more time-consuming than you'd guess. So, try to place holes such that their axial direction is vertical.

A common error is to attempt to print features that are too fine for the printer. In layer view, slowly raise the slider from the bottom to the top, examining for yourself where the plastic will actually go for each layer. You'll need to have both "Shell" and "Infill" checked, and the extruder itself must also checked. If there are problems, go back to the design phase in *Fusion*, *Inventor*, or *Sketchup*.

Cura will allow you to screw up your design, using the various settings in the "print setup" menu on the right. Until you've made at least 6 prints, just use the "recommended settings" option. At the top left, I'm only providing "PLA" material, and our nozzle size is 0.4 mm. Make sure these are correct in Cura. The nozzle size is more or less the horizontal width of printed lines. The "layer height" is more or less the vertical thickness of printed lines, and is usually 0.10 to 0.15 mm.

Here are some features that you may manually adjust:

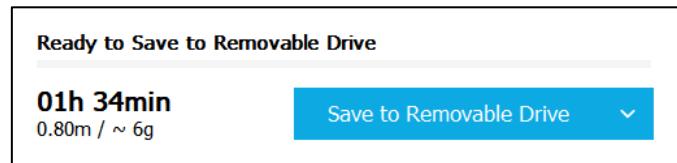
- In Cura, within the "print setup" menu on the right, there is an option for "Support". "Support" refers to extra material that will be printed in addition to your design. Supports are added by cura whenever you have a severe overhang (or similar feature) that would otherwise sag during printing. In most cases, you should create your original design with an intended print orientation, or with multiple parts, such that support is not needed anyway. But, support is sometimes necessary. Set the support angle to about 50°. "Towers" are rarely needed. Support structure must be removed by hand after the print is done. Sometimes it comes off cleanly, but other times it requires quite a bit of filing and scraping to remove.
- Also, within the "print setup" menu, one of the categories is "build plate adhesion". Generally, use a "skirt" (which is a thin line printed before your object just to make sure the print nozzle is clear). If the skirt line comes out poorly, stop the print *before the first layer is even done*. You may use a "brim" (a thin sheet attached to the lowest layer of your object) if your print has a mass of 50 g or more (the total mass is shown on the bottom of Cura, a little to the right). Because of the differential rates of cooling of the printed plastic, the bottom layers of the object may warp or even separate from the print bed before the top layers are done printing. A "brim" is an attempt to minimize these two possibilities. The third choice of adhesion, "raft", will never be used.
- As mentioned above, you may set the "Infill Density" to any value between 8% and 20%. You need special permission for other values. Larger values result in printed objects that are more "solid", while lower values have more air inside of them. You're probably thinking that "more solid is better". You would usually be wrong about that. Higher densities warp more, and smaller values are almost as strong, and print much faster and for a much lower cost. Most of the strength comes from the skin or shell anyway, not from the infill.
- The default "Speed" for printing is 50 mm/s. You may lower it if you are printing very small objects (less than 3 grams) with very fine features.

► Within the “Shell” submenu, the default wall thickness describes how thick the skin of your object will be. Although you (sort of) specify this value in mm, the resulting thickness will actually be an integer number of “lines” of printing, rounded up. For example, if your nozzle is 0.4 mm thick, and you ask for a 1 mm skin, then you would expect 2.5 lines. That’s not possible. So, you would actually expect the skin to be three lines thick, which would be nearer to 1.2 mm of skin rather than the 1 mm you requested. Similarly, all other “fine” features will be rounded to an integer number of lines.

Once you’ve made your choices for your settings, review the layers carefully using your own magnificent brain before using Cura to save the G-code.

Saving in CURA to “G-code”

Your computer will not attach to the printer directly. You save your printer file using an SD card on your laptop. You should find an SD card-to-USB adapter with the printer, because otherwise, most computers don’t have SD readers but most have USB ports. Once you place an SD card into your laptop, CURA will automatically detect it, and prompt you to save your work immediately (bottom right). Next to this, you’ll see an estimate of the time required to print your object. You may not print any object requiring more than 6 hours without Dr. Pogo’s prior approval.



“Saving” places a G-code file onto the SD card. In the most real sense, *this* file is what will be printed. Because of the choices you made about skin thickness, etc., your original design is not truly recoverable from the G-code file. The G-code file is an approximation of the design.

4. Physically Printing

The interface for the printer is a little hard to get comfortable with. It’s a menu driven system, where the only input is a single wheel on the bottom front panel of the printer. Rotate the wheel to go to a new choice (as shown on the nearby LED display), and press the wheel inwards briefly to actually make the selection. It is easy to accidentally rotate the wheel one click while pressing in.

Safety

There are a few physical dangers when 3D printing. The most obvious is burning yourself. The nozzle temperature is more than 200 degrees Celsius, or around 400 degrees Fahrenheit. So don’t touch it! While printing, the build plate itself is also pretty hot at around 60 Celsius (about 150 degrees Fahrenheit). It’s possible to touch it without harming yourself, but only for a second or two.

The next issue is a little less dangerous. Don’t stick your hands (etc) into the print volume while the printer is running. The print head moves around and changes direction quickly, and you can trap or crush your hand between some of the printer guides. Most likely, this will break the printer, but it can also damage your hand.

Finally, as discussed below, removing the object after it has been printed usually requires a razor blade. Naturally, it would be very easy to cut yourself with the razor. Also, if you get frustrated and

jam the razor in too hard, you can snap off a portion of the blade so that it flies across the room, into your eye or your (former) friend's eye. Wear safety glasses when using the razor, and go slowly! Work the razor around the edge, one millimeter at a time.

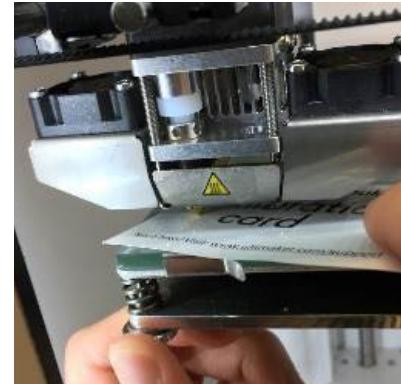
Build Plate Leveling

A common physical problem that occurs with the printer itself is that the glass print bed becomes out-of-level. Usually, this happens when the prior user, a foolish and impatient student, tries to remove the finished object from the printer while the print bed is still hot. While you *can* physically remove the object in many cases, it requires an excess of force. If I find out you're doing it, you'll lose your certification to use my printer. So, wait until it's about room temperature before removing the printed object from the printer.

You'll know that there is a problem with the levelness of the build plate by observing the thickness of the "skirt" that prints before your object is printed. The "skirt" will be a thin outline, separated by a centimeter or so from your object. If the width of the skirt is not uniform for its entire length, the print bed is not level. In this case, *Abort* the print immediately (using the wheel on the front panel), and re-level the plate before trying again. If you don't see "Abort" as an option on the LED, then first choose "Tune".

To level the build plate, rotate the knob to "maintenance", then "build plate", and then follow the directions there (the next paragraph is just a clarification of those instructions). You will require a small calibration card for this purpose; there is a stack of yellow calibration cards attached to the left side of the printer. Return them to their holder when you are done.

Using the calibration card requires that you first separate the nozzle from the build plate (using the front-panel dial for the back screw, or using your fingers for the two front screws). Then, slowly adjust the build plate, raising it up towards the nozzle while repeatedly passing the card between them. At first, the card will not touch the nozzle at all while passing below it. The card should remain flat with the build plate for each pass. But, when the build plate becomes close enough to the nozzle, you will feel that the card is brushing against the nozzle as you slide it along the build plate. As soon as you can just barely feel the card brushing the nozzle, you are done. Do not continue to raise the build plate so far that the card either feels a lot of resistance, or worse, gets jammed stuck between the build plate and the nozzle. You can seriously damage the nozzle by pressing the build plate too hard against it!



Adhesion

Another issue that arises for many prints is a lack of build plate adhesion. Basically, the 3D part becomes unstuck from the glass build plate while it is in the middle of printing. Once this happens, the printed object will become permanently stuck to the nozzle rather than the build plate, and all the new plastic will be improperly deposited. Fortunately, with our usual material (PLA), this problem happens a lot less often than it does with other materials (e.g., ABS). So, before you print, you should use a glue stick to add a *thin* layer of glue to the glass build plate over the regions where the print will happen. In many cases, the leftover glue from the previous print will be *more* than

enough to ensure proper adhesion. The added glue should be uniform, thin, and not clumpy. Don't overlap layers of glue. Similarly, if you add too much glue (such that the layer of glue itself becomes a tenth of millimeter thick or more), your bottom layer will print poorly, and (surprisingly) make adhesion worse than if you had no glue!

Changing Material

You should only use PLA in my machine. However, you'll sometimes need to change material to get a different color, or because there is not enough material left on a spool to completely finish your next part. You should note that CURA tells you the length, in meters, needed for each part in the bottom right corner.

The plastic is stored in a spool on the back of the printer. The filament passes from the spool through a small motor, then through a tube called a "Bowden", and then into the print nozzle itself. The spool itself is oriented such that the filament spirals counter clockwise as seen from behind, so that the free end of the filament goes *up* into the feeder motor housing. To change material, use the wheel on the front panel to select "*Material/Change*", and follow the directions on the LED display. This process will actually cause material to flow from the nozzle as a final check. Once you see that 6 inches of smooth material has been made, you can end the process.

Starting a Print

Insert the SD card into the slot in the front and rotate the button to navigate to "print", and push the button to select it. Then select the file you would like to print by rotating the wheel, and pressing the wheel inwards on the correct file.

You must stay and *personally watch* the first four layers print before you leave the printer to finish the job. The first thing that happens is the nozzle prints a small blob of material in the front left corner of the build plate. Next, it will print the skirt, outlining your part as discussed above. While watching the blob, the skirt, and the first 4 layers, you should make sure that there is no peeling (loss of adhesion), that the line widths stay uniform, and that plastic is flowing smoothly from the nozzle without jamming or sticking due to surface tension. If you see any of these problems, *Abort* the print immediately!

Modifying Settings During a Print

You can change a few things "live" while the print is in progress. This is done using the "Tune" menu on the front of the printer. The default build plate temperature is 60 degrees Celsius, and the default nozzle temperature is 210 degrees Celsius. It takes longer for the build plate to heat up than for the nozzle. If your part is showing some signs of peeling from the build plate, you might try raising the temperature of the build plate, but to no higher than 65 degrees. Similarly, if the nozzle is sputtering a little bit, you might try to raise the temperature of the nozzle a little (but to no higher than 215 degrees). Similarly, you can raise or lower the print speed a little bit, too. If the print is too stringy, maybe slow it down to 90% of the default speed.

Finishing Your Print

Every time you start any print, even if you abort it during the “skirt”, you have to log it. Logging should happen when the printer stops (whether due to an “Abort”, or because the job finished), because I will be asking you for the total minutes that the printer was operating.

To log your printer activity online, use this Google form:

<https://goo.gl/forms/y4WgV1t5KauZqrE43>

There is a quick link to this page from the LabVIEW course homepage. You *won't* get in trouble for aborting a bunch of prints, but you *will* get in trouble for not properly logging your activity.

Once you have logged your use of the printer, you can begin the process of *waiting* to remove your object from the build plate. It will take at least 15 minutes for the build plate to cool down enough (to room temperature) for you to remove the printed object. This is not exactly a safety issue: the hotter the plate, the more completely the object will be stuck to it. For larger objects, you might have to wait more than 45 minutes to begin the process of removing the part. The LED panel on the front of the printer will tell you when you might start to remove the smallest parts.

Wear safety glasses to remove the object. Use a razor blade and work your way around the edge of the part slowly, digging the razor no further under the part than 1 or 2 millimeters. Once you have a millimeter separated around the entire edge, start another circuit, sliding the blade another millimeter more as you go. Eventually, the entire piece will just pop off suddenly. This is when you are most likely to cut yourself with the razor, which is why you’re going slowly. A second important issue is that when you get impatient, you’ll push too hard, and you’ll damage the mechanism holding the build plate. At best, this means re-leveling by the next student (which might be you!). At worst, it means I have to buy and install replacement parts. So, be gentle!

Instead of working a razor around the object, you can also use dental floss. Place the middle of the floss under one corner, and gently saw back and forth while pulling the floss towards yourself until the object pops off of the build plate.

In most cases, there will be some small strings of plastic dangling from your part. Similarly, there may be other glitches due to “rounding” errors as discussed on page 6. You should use the small file set provided to clean up the edges. Don’t push too hard when filing; the files are very easy to break!

Cleaning Up

If you used your own SD card, then return *my* SD card to the printer slot when you’re done. Use the razor to gently scrape off any glue *clumps* from the build plate, but do not otherwise clean the buildplate or try to remove *all* the glue residue. Return the other tools (leveling card, glue stick, goggles, razor, etc.) to their proper place.

5. Troubleshooting on Your Own

Ultimaker provides an immense amount of advice for troubleshooting their printers. I'll discuss the most common issues below, but here is a good place to start hunting for other issues:

<https://ultimaker.com/en/resources/18785-print-quality-issues>

Unjamming the nozzle (material is not extruding)

This is a pretty common problem.

Step 1: While it is cool, clean the outside of the nozzle, removing any excess material that is stuck. Then, using the wheel on the front panel, select *Maintenance/Advanced* and heat the nozzle to 150 degree Celsius. Then use a razor, tweezers, or pliers to gently scrape off any burned-on plastic. Don't burn yourself!

Step 2: Select *Advanced/Move* material, and wait for the nozzle to heat up a little more from 150 degrees (which is the temperature during step 1) to 220 degrees. Slowly rotate the wheel clockwise to push more material through the tube (rotating counter-clockwise will attempt to back material out of the nozzle). The motor that pushes the filament through is inside the small white square on the back of the machine. If you hear a clicking sound, then the motor is not successfully pushing anything through, and is probably damaging the filament. If you hear this, then grab the filament below the motor with your left hand, and push upward with about 5 pounds of force while you advance the wheel with your right hand. Don't push too hard... you're just giving a small "assist" to the motor. You'll know you're successful if you succeed in pushing through about 6 inches of smooth plastic from the nozzle. Stop "moving" the material, remove and loose strands of material, and re-start your print as quickly as you can (so neither the build plate nor the nozzle has a chance to cool off).

Step 3: ["Atomic" method: use this only if the first two steps above have failed]

Original article with video:

<https://ultimaker.com/en/resources/19510-how-to-apply-atomic-method>

Select *Material/Change*. This will pull the filament all the way out of the printer. Then select *Cancel* to get rid of pesky questions about the new material it thinks you're going to add. Then remove the blue clip at the end of the Bowden on top of the nozzle casing. Place the blue clip on the table, and don't lose it! With the blue clip gone, you can push down on the white cylinder that holds the Bowden tube, and while the white cylinder is down, you can pull the Bowden up and out of the nozzle. Select *Maintenance/Advanced/Heat up Nozzle* and raise the nozzle temperature to 225 degrees. While it's heating, find some leftover PLA that is too short to print with anymore, and cut a 20 cm (8 inch) long segment of it. Straighten it as best as you can by gently bending. You might use some cutters to give it a little pencil point. Once the nozzle is hot, gently push the point of your new PLA stick into the nozzle, right through the white Bowden tube holder. Push hard, but not too hard. If you're lucky, some material will begin to drip from the nozzle. Whether lucky or unlucky, the intention is to "weld" your new stick to any clump of old plastic that is melted inside the nozzle. So next, lower the temperature to 90 degrees (which is still pretty hot!). When the temperature reaches 90, grab the top of your pencil-like stick of filament, and remove the material with a sharp tug!

Hopefully, all the clogging material will come out. You might have to repeat this entire process again. You'll be certain there is no more clogged plastic if your pencil-like stick is removed without any new excess material stuck to it. To reassemble the Bowden tube, press the white cylinder down and insert the tube pretty far into it. Then release the cylinder and tube, and slide the blue clip back into place. Then follow the directions on a previous page to connect a spool of filament.

Fixing a jammed feeder (the material is not being moved forward)

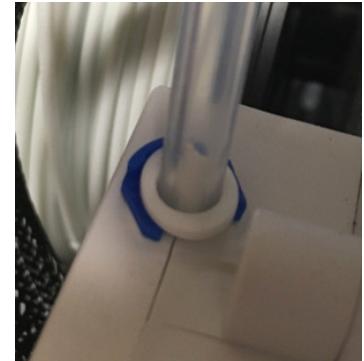
This is usually because the feeder motor has chipped or roughened some of the filament. Select *Material/Change* and follow the directions to attempt to remove the material. The nozzle should first heat up, and the printer should then attempt to reverse the material. You may gently pull down on the filament *below* the motor housing in the back to help it move. You may cancel and attempt to reverse material again if it does not work the first time. After the material has been removed, cut off the damaged length of material that was extracted and reload the material as prompted by the printer. Save any pieces that are longer than 8 inches for use with “atomic pulls” later. Throw away anything shorter than that.

Leveling the build plate

This was already discussed on page 7.

Filament is stuck in the Bowden while trying to change materials

If the filament is kinked, it can jam inside the clear plastic Bowden. While removing it, you can try to assist the feeder motor by pulling down on the filament below the motor housing, as discussed above. If this doesn't work, you'll need to disconnect the Bowden completely. First, disconnect it from the nozzle end (as discussed in the “Atomic method” section above) by removing the blue clip. When the nozzle end is disconnected, also remove the blue clip and Bowden from the feeder end. Grab the entire tube, remove it, and set it aside. Then, cut the damaged filament just above the feeder motor, and remove the last bit of material using “*Move Material*”, as discussed above. Then re-assemble the Bowden, and re-insert new (undamaged) filament.



6. Contacting Dr. Pogo

You should have access to the following supplies. Dr. Pogo can help if they need to be replaced:

• Filament (PLA)	• Goggles	• Wire Cutters
• Glue stick	• Razor	• Calipers
• Build Plate leveling cards	• SD Card	• Flat or round file
• String with handles.		

Abort your printing and contact Dr. Pogo if

- The build plate is wobbly.
- The cooling fans besides the nozzle are not running (you should first check to see whether you accidentally turned them off using a CURA setting).
- The stepping motors make loud clunky noises instead of quiet buzzing noises.
- The feeder motor is losing traction, causing a clicking sound on the feeder and resulting in missing material in the middle of your object.