THE ULTIMAKER PRINTERS

The Ultimaker is an FDM 3D Printer, which means it melts plastic and extrudes it in a
pattern, building up a 3D shape layer by layer. The pattern is determined by a digital 3D model, which the software “slices” into layers.

This training will include an orientation to both the Ultimaker and to Cura, the software used with it at the Foundry.

**Safety**

For 3D Printers in general...

- **Shop Buddy:** You cannot use the 3D printer alone. For the 3D printer, your shop buddy can be in the room with you or in the main workspace in a position where they can see you. If your shop buddy is not in the room, you must work with the door open.
- **Eye Protection:** You must wear eye protection when working in any shop space, to protect your eyes from anything that comes out of the machines.
- **Close-Toed Shoes:** Tools, bits of material, caustic resin, and even the printers themselves can fall and close-toed shoes will protect your feet from cuts, bruises, burns, and breaks.
- **Long Pants:** Long pants will protect your legs from cuts, bruises, burns, and splinters that might come from handling these tools.
- **No Jewelry:** Rings, bracelets, dangling necklaces, watches, headphones and sweatshirt strings can all get caught in the machine, which can drag the machine off the shelf or drag you towards the machine.
- **Long Hair Secured:** Like jewelry, long hair can also get entangled, and potentially drag you into the machine.
- **No Food or Drink:** Printing material, especially resin, is toxic if consumed, and the residue can contaminate food and drink. Residue from food or drink will also make the machines gross.

For the Ultimaker FDM specifically...

- **Clean Bed:** The “bed” or print area of the machine needs to be clean for it to work properly. Before you print, check to make sure it's clean, and after you print make sure nothing is left behind.

**Printer Anatomy**
Main components Ultimaker 3 or Ultimaker 3 Extended

1. Build plate
2. Dual print head
3. Build plate screws
4. Build plate clamps
5. Push/rotate button
6. Display
7. USB socket
8. Feeder 2
9. Bowden tubes
10. Feeder 1
11. Ethernet port
12. Cable cover
13. Double spool holder with NFC cable
14. Power cable

The model exposed is the Ultimaker 3. The main components of the Ultimaker 3 Extended are the same and on the exact same position. The only difference between the two printers is the height of the housing. See Specifications for the different dimensions.

OPERATING THE MACHINE
1) **Load your part into Cura**

Open Cura on the computer next to the printer, or on your own computer. You should be looking at a window similar to this:

If you are using your own device, make sure your Cura is configured for the “Ultimaker 3 Extended” printer, using the button in the top right corner.

Click the “Open File” icon in the top left corner. Select the STL file you want to print and open it.

Your part will appear in the Cura workspace window. If it fits in the printer workspace, it will appear yellow, otherwise it will appear striped. A striped piece is too big to be printed.

A yellow piece with red patches on the underside is okay: those patches indicate where supports will be placed. Supports are tiny scaffolding designed to support a piece so it doesn’t collapse during printing, but can still break off easily from the finished piece.
Click the yellow model to bring up the placement menu on the left side. Use the top four buttons to adjust your piece:

- **“Move”** will let you reposition your piece. Try to put it in the middle of the print bed.
- **“Scale”** will change the size of your piece. If you are making a functional piece rather than a decorative one, be careful: this will also resize things like bolt holes.
- **“Rotate”** will change the angle of your piece. This is the most important placement setting to get right. Rotate your piece so that it sits securely, and won’t need too many supports, or too tall supports.
- **“Mirror”** will reflect your piece around some axis, so you can create two images that are the reverse of each other.
- The other two options, “Per Model Settings” and “Support Blocker,” should be left alone.

2) **Select print settings**

- **Material:** Select the material you’re using: either PLA or ABS.
- Click “Custom” to see more options. The recommended options are usually good, but not that versatile.
- **Layer height** is how thick the slices are. Smaller layers will mean the print is higher quality, but it will also take longer since there will be more layers. A good layer height is 0.25 mm – if you want to print thinner than that, ask a Scout if that’s okay.
- **The Shell** is the outside of the piece. Printed parts are semi-hollow, but the shell is solid up to a small depth. A good wall thickness is 0.6 mm, strong and thick enough that scratches won’t make it through to the
hollow part. If you are printing something that needs to support a lot of weight, you can turn the wall thickness up as high as 1 mm.

○ The inside of a 3D printed part is filled with a honeycomb pattern. Infill density is how much it is full of plastic instead of air. Although it sounds low, 20% infill is a good number. Even as low as 10%, a piece will still be strong. 30% is very high, and should only be used on pieces that need to support a lot of force. Higher infills will also extend the print time, so don’t make it higher than you have to.

○ The settings under “Material” should be automatically set. Make sure they match the guide on the wall, but don’t change them.

○ Everything under Speed is set automatically as a property of the machine. Leave all of the speeds as-is.

○ There’s only one setting under Cooling: Enable Cooling. Check the box.

○ Supports are printed scaffolding that support any overhanging parts of your piece to prevent it from collapsing. Supports are made to break away easily, so you can remove them after you print. Check the box to make Cura automatically generate supports.

○ Build Plate Adhesion determines what prints where your piece sits on the build plate. There are four options: None, Skirt, Brim, and Raft. We recommend a brim for all PLA parts and most ABS parts, or a raft for ABS parts that seem more likely to warp.

  i) None means the printer will start right off with your piece, without making a base. This is risky, as prints are more reliable with a base.

  ii) A Skirt is a thin line of filiment that outlines your piece, but doesn’t actually touch it. This is almost the same as None, but it gives the extruder a chance to warm up first, so it is slightly less likely to jam.

  iii) A Brim is a thin sheet printed beneath your part. A brim helps your piece adhere to the build plate, it holds the bottom flat during the early stages of the print, when the piece as a whole is thin, and it gives the piece some protection against scratches when you remove the piece from the printer. Brims are usually the best option.

  iv) A Raft is much thicker version of a brim. It serves the same benefits, but is much better at preventing warping. It has the downside of increasing the print time and using extra filiment, so it should only be used when warping is a concern.

Once your settings are all ready, click “slice” in the bottom right corner. This will generate a “gcode” of your part – instructions for the printer. It will also tell you how long your print will take, how many grams of material it will use, and how many meters of filiment it will use up.
If Cura’s time estimate is longer than 8 hours, check in with a Scout. Long prints can be run overnight, so we may ask you to wait so that shorter prints can happen during the day.

3) **Physical Checkup on Printer**

Choose one of the Ultimaker printers and do a checkup. There are four things you want to check for.

Check if the bed is clean. If there are any old prints left there, remove them and put them on the shelf. If there are any scrap bits of filament on the bed, gently wipe them off with your hand or a paper towel.

Make sure the nozzle is clean. If there is any plastic solidified on it, gently pluck it off by pulling downwards. Be careful not to push any material back into the nozzle.

Last, check the roll of filament on the back. Make sure it is the right type of material (it should match what you selected in Cura), make sure the color is one you are okay with, and make sure there is enough filament left on the role. Cura tells you how much length is required, so you won’t start a print that will run out before it finishes.

4) **Load part into Ultimaker itself**

One the file is set up in Cura, and the printer is ready to go, you can start the print.

Since you’ve already sliced the part, you will now see a button that says “Save to Removable Drive.” With a USB drive in the computer, click this to save your gcode to the drive. Safely eject the drive and plug it into the Ultimaker.

Using the dial on the Ultimaker, select “Print.” Then scroll through to find your piece and select it. Your part will start to print automatically.

5) **While your part is printing**

Part of the beauty of 3D printing is that after setup, the machines are completely automated. You don’t need to stick around for the whole process, but you need to watch the first few minutes.

The printer will heat the bed and the nozzle, which will take a minute or so. Then it will move to the front left corner and extrude a small blob of filament. This is to make sure the machine extrudes properly – if you don’t see a blob, pause the print and ask a Scout to help you fix the filament.

Then the print will start for real. The vast majority of 3D printing failures happen during the first few layers. Watch the printer make the brim or raft, and let it print enough that you can be sure it is working. If something goes wrong, stop the print and ask a Scout to help you fix it.
6) **Now that your part is done...**

Make sure you know how long your print will take, and when it will be done. Try to come back and get it soon after it is ready. This saves the next person the trouble of having to remove your part from the bed, and it avoids the risk that someone will mishandle your part.

Use common sense here. If a part is going to be done at 3am, you don’t have to get it then – some time the next day is fine.

When you do retrieve your part, remove it from the bed using one of the scrapers that hang on the wall. It’s better to use the scraper like a chisel, removing the piece with several small pushes rather than hacking it off all at once. More forceful pushes can scratch and damage the bed.

---

**Maintenance for Ultimakers**

Maintenance numbers
TECEDO.com (603) 224-8324
Company who sold us the machines (800) 338-2238

Material is stuck in the 3d printer head/extruder)

1) **Feed material out through the back of the feeder**
2) **Recommended open a window**
3) **Remove Boden Tubes (note, there is a small white moon shaped piece of plastic connecting the tubes and printer head, remove that first)**
4) **Take material and put it directly in the top of the print head, mark with a marker where the first exposed material is when you insert the material. This will make it easy to see how much the material has moved**
5) **Turn the printcore heat up above the melting point of the melting point of material inside that is stuck (if you are not sure, try 200)**
6) **Gently push the material through the print head**

The printhead supports should flex only slightly.

7) **Gradually, the material will go through the print cores. It can take up to two hours of gently pushing the material through the print cores for the print material to be all the way through the machine**
8) **As material builds up on the end of the print core, dab the material away with a paper towel (but remember it is hot)**
9) **Clip off the material that is covered in the material that is stuck in the machine**
10) Repeat steps 1-8 until the material is stuck is completely removed