UltiMaker S7/UltiMaker Cura

This document is a guide to the basics of working with the UltiMaker S7 3D printers in the DCMME Engagement Center. Details are provided on how to navigate the printers' interface, take proper care of UltiMaker filament spools and print cores, load/unload filament into the printer, and work with UltiMaker Cura to use a 3D model to create a physical 3D object. Common troubleshooting procedures are included as well.

S7 Interface

On the interface on the printers, you can access 3 menu options:

Status (the circle icon): This menu is typically the one displayed on the interface, here the printer's status is shown, indicating if a printer is ready to print or in the middle of a print, as well as if a print job succeeded or failed.

Configuration (the 3D printer outline icon): This menu displays what printing filament is installed (if any) and what print cores are installed.

Preferences (the gear icon): This menu consists of 3 submenus:

      Settings: General interface settings

      Maintenance: Maintenance and calibration procedures, as well as diagnostic logs

      Network: Network and Wi-Fi settings
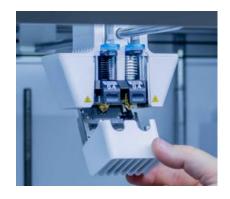
Filament Spools & Print Cores

The image above is a filament spool; you can locate the filament spools under the right side of the table in an open cardboard box. There are multiple types of filament material in the lab: **Tough PLA**, **ABS**, **PVA Natural**, and **17-4 PH**.

Tough PLA (Polylactic Acid) and **ABS** (Acrylonitrile Butadiene Styrene) are thermoplastic materials, meaning they soften at high temperatures then harden when cooling. These two materials are the most used in 3D printing. ABS can be a more durable material than PLA, but PLA is generally more recommended for beginners.

PVA Natural (Polyvinyl Alcohol) is a water-soluble material. Unlike tough PLA and ABS, it is not meant for building and instead is used as a supporting material. This material requires extra care despite its advantages, see "Notes about PVA Natural".

17-4 PH is a metal-polymer composite filament, typically used to produce stainless steel components (although debinding and sintering is required to create a truly finished metal part). This filament is more expensive to use than the other materials discussed in this section since prints using this material require the part to be completely solid. **Only lab managers can use this material.**

Before loading a spool into a printer, you should know what print cores are and the differences in print cores to know when a certain material can or cannot be used.

The picture above on the right is a print core: a compact hot end that contains a nozzle, a heater and a sensor. When the printer is active, this part heats up the filament and extrudes it from the nozzle. Each printer has two print cores in the print head.

There are three types of print cores: AA, BB, and CC.

AA: Compatible with any building material (such as **tough PLA** and **ABS**), as well as breakaway support material. Clean the print core if you're swapping between ABS and tough PLA (see "Cleaning the Print Core")

BB: Used for printing with **PVA** support material

CC: Used for printing with composite materials (such as **17-4 PH**, so only lab managers should be using this core)

Typically, the left nozzle is an AA core, and the right nozzle is a BB core, but you can go to the configuration tab to verify what print cores are installed in the printer.

Print cores are rather simple to maintain and switch out. Before removing a print core, ensure that no material is loaded into the print core (See "Unloading Filament"). Select which print core you want to remove in the configuration tab, then click "Unload". The interface will prompt you to open the print head fan bracket, then squeeze the print core lever upward and slide the print core out of the print head. To put in a new core, squeeze the new core's lever upward and slide it into the print head slot until you hear a click. Do not forget to close the fan bracket when you're done. The configuration tab should update and display the new print core installed.

Loading Filament

After choosing a filament, mount it on the holder located on the back of the printer. Ensure it's in the correct direction so it can enter the feeder from the bottom (the box mounted on the printer with the tube attached to the top)

Note: if you're loading two materials at once, the filament for the right core should be loaded first, then the filament for the left core.

Go to the configuration tab and select which side you want to load filament for. Click "Load", then click "confirm" until it brings up the menu where the interface is detecting the material mounted. Wait around 15-20 seconds. If the detected filament is correct, select it and proceed. If the detected filament is incorrect or no filament has been detected still, manually select the type of material and the color of the filament.

Insert the filament into the bottom of the feeder, maintaining pressure until the feeder grabs the material. If the feeder starts spinning but hasn't grabbed the filament, try realigning the filament until the feeder grabs it. You can also flip the lever on the side of the feeder upward to manually insert the filament and push it in until you see it in the tube. **Remember to flip it down when you're done or the feeder will not move the filament.**

Wait until the feeder finishes spinning, and the filament reaches the core and begins extruding from the nozzle. If the filament doesn't reach the core, redo the loading procedure on the interface.

Unloading filament

Select the filament you want to unload on the configuration menu, click "unload", and follow the procedure shown on the screen. When the filament is fully unloaded from the feeder, use pliers or scissors to cut off the stringy end of the filament. Place it back in a plastic bag along with a silica gel packet.

Notes about PVA Natural

PVA Natural is a useful supporting filament that dissolves in water, saving you the hassle of chipping off plastic when touching up your finished print and removing supports. However, because it is highly soluble in water, it requires extra care in humid environments. If improperly stored and handled, PVA starts to become brittle and can start to snap easily, which **will** cause issues in both feeding and printing. (Note for lab managers: if the lab doesn't have one already, maybe consider getting a filament dehydrator)

In the meantime, **please do not leave PVA Natural out of the bag for extended periods of time if not in use.** If a PVA spool is installed on a printer that isn't in use, unload it and

put it back in a plastic bag with at least one silica gel packet immediately. If a spool is in bad condition (such as you can barely bend the filament without snapping it), UltiMaker suggests placing the spool on the plate at 55°C for at least 24 hours.

Printing

Alright, now that you've read the basics of working with the printer components, let's get into printing something.

We want to turn a virtual 3D model into a physical 3D model, but we can't simply just give the printer a .stl file. We need to give it instructions on how to actually build the model (i.e., how the print head moves when printing each layer). For that, we use UltiMaker Cura. There are three menus in Cura for each step of the printing process.

Prepare Menu

With Cura, we can upload a 3D model file (stl, obj, x3d, etc..), and configure how we want the printer to build the model. First, select the printer you want to send a print job to with the configuration panel (the rectangle at the top with the printer's name, materials installed, and settings panel). The printers in the lab should be connected to Cura, and typically have the name "Engage", then a number. To find a certain printer's name, it should be on the top part of its interface.

On the left side of the screen, you have tools allowing you to position, scale, and rotate the model. Furthermore, you can access the print settings by clicking the downward arrow on the right side of the configuration tab, allowing you to adjust an incredibly large selection of settings for how the printer will build the model.

Next, click "slice" in the bottom right of the screen. This will compile the instructions for the printer, displaying an estimation of how long the print will take and how much filament would be used.

Preview Menu

Your model is now ready to print. Cura provides a simulation of how the printer will build the model, layer-by-layer. We recommend using the preview tab to ensure your model will print properly. You can use different color schemes to see different line types, differentiate the infill from the outer shell, and even use X-Ray view to check for gaps in the model.

Play around with the various print settings to see how the printer would change its process!

When you're ready, there are two methods of starting the print. You can save the .ufp/.gcode file to a USB flash drive and plug the flash drive into the printer, but we recommend printing over Wi-Fi.

Monitor Stage

This menu allows you to monitor your print progress directly from Cura (you can also see the print job progress on the Status menu on the printer interface)

Starting the print will take some time; the plate and the print heads need to heat up, and the plate must calibrate its leveling before beginning the print.

Removing

When your print successfully completes, you can remove the plate from the printer. Wait for the plate to cool down first; it **will** be hot when the print is finished. When it is cool to touch, hold the plate by the tabs, lift it up, and slide it out of the printer. Bend the plate to detach the print from the plate.

If the print is not detaching from the plate, use a plastic scraper to help remove the print and any brims. **Do not scrape the plate with a metal tool.**

Once removed, place the build plate back by aligning the plate at an angle, then lowering the plate (be aware that the magnets that attach the plate are strong)

Support material – Plastic

If you're using a building material to also support your print, printing the support layers with a zig-zag pattern is the easiest to remove.

When you remove your print, try to remove the supports by hand or with pliers. You can use something like a flathead screwdriver to help, but it might damage your print if you're not careful.

Support material – PVA Natural

When you remove your print, find a container to fill with water and submerge your print in to let the PVA dissolve. Dissolving PVA takes a while, even up to several hours. When the PVA is almost completely dissolved, rinse the print to remove any excess, then let the print dry completely. Dispose of the water.

Troubleshooting

Material Errors

The print head can detect if filament is being properly extruded from the nozzle(s), so the interface will display the error above if no filament is being extruded.

If you see this error, check the spool to see if it has any filament. If it's empty, you can click "Ok" then navigate to configuration. Click on the material that you need to swap, then click "change". Follow the directions, swapping the empty spool with another spool of ideally the same material and color. If you can't find a similar spool, you can cancel the print, or you can at least use a spool of the same material and a different color.

If the spool **has** filament, then it most likely is a feeding issue. Ensure that the filament isn't crossing itself or that the spool isn't stuck. A good way to check is to tug on the filament a little bit. If the spool starts to spin a little, then the feeder should work properly. Otherwise, attempt to fix the "tangle", then resume the print job.

If the error persists, check if the filament is brittle or if it snaps easily. If so, remove the spool, place it in a plastic bag with a silica gel packet, and replace the spool like above. If the filament seems to be in good condition, you might have to clean the print core.

Cleaning the Print Core

Cleaning the print core typically isn't necessary to do often, but it's helpful to do when the filament isn't extruding from the core nozzle. It's also recommended to do when swapping filaments of different materials. For example, PLA has a lower heating temperature than ABS, so switching to PLA after using ABS might result in clogging in the nozzle.
To start, navigate to preferences -> maintenance -> print head -> print core cleaning. Select which print core to clean and UltiMaker Cleaning Filament as the material. Wait for the print core to heat up.

The interface will prompt you to disconnect the Bowden tube (the clear tube connecting the feeder to the print head). To do so, remove the clamp clip holding the tube to the print

head. **Be careful when removing the clip;** it's a small piece of plastic and can be easily lost. Hold down the coupling collet (the ring around the tube on top of the print head) and pull the tube upwards.

Hot pull: when the core is heated, grab some pliers and a cleaning filament rod, insert the rod into the core, hold the top of the rod with pliers, and apply pressure; the filament should be extruding from the nozzle. Keep applying pressure until the nozzle extrudes clear filament, then pull out the rod quickly. If the rod is mostly clean, then the core should be good to go. Otherwise, cut off the tip of the filament and repeat.

Cold pull (Optional): After performing the hot pull, the interface will prompt you to do the cold pull, but typically you can skip this step and cancel the cleaning procedure unless you want to ensure the core is thoroughly clean. Continue to the cold pull procedure, and insert the filament again into the print head, just like in the hot pull. Confirm to continue and maintain pressure until you are prompted to release the filament. Wait until the core cools down. Grab the filament with pliers and pull it out with a quick, firm tug. Repeat the cold pull if the filament is not completely clean.

When done, reinsert the tube and insert the clip under the collet.

Manually setting core temperatures

Cleaning nozzle exterior

In some cases, extrusion can go poorly, and filament starts burning against the nozzle, and can even clog up the area around it. To remove burnt filament from both the nozzle and the area around the nozzle, you should manually heat up the print core to soften the filament stuck on the print core and scrape it off. **Wear gloves to protect your hands from the heat and be very careful not to touch the hot print core. The print core can have a temperature of up to 200°C.**

To set the print core temperature manually, go into the configuration tab, select the print core, then click on the icon with the 3 dots and click "set temperature".

Start at 150°C, open the fan bracket, and scrape the burnt filament off the nozzle area with a flathead screwdriver or a metal scraper. Increment if needed but be very careful and avoid touching the print core.

When done, set the temperature back to around 50°C and close the fan bracket.