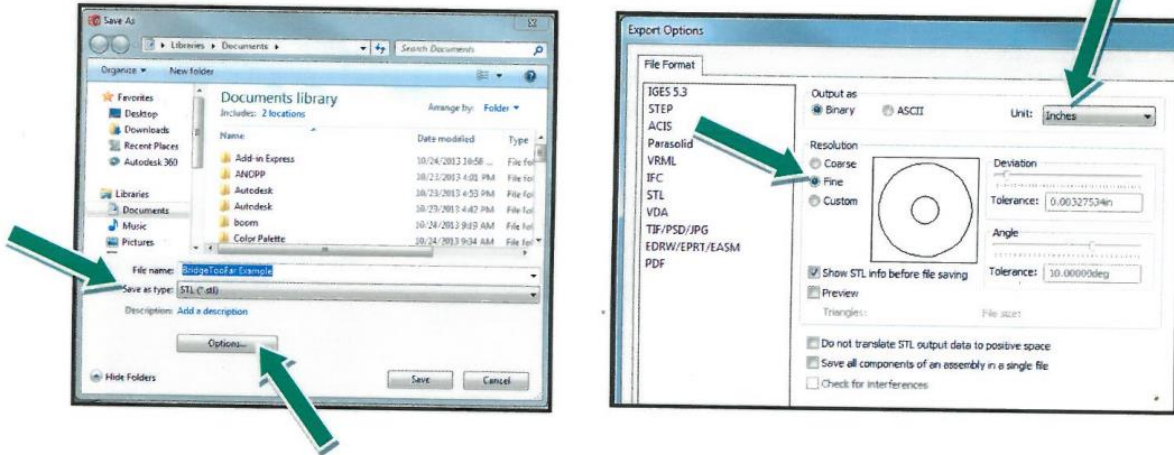


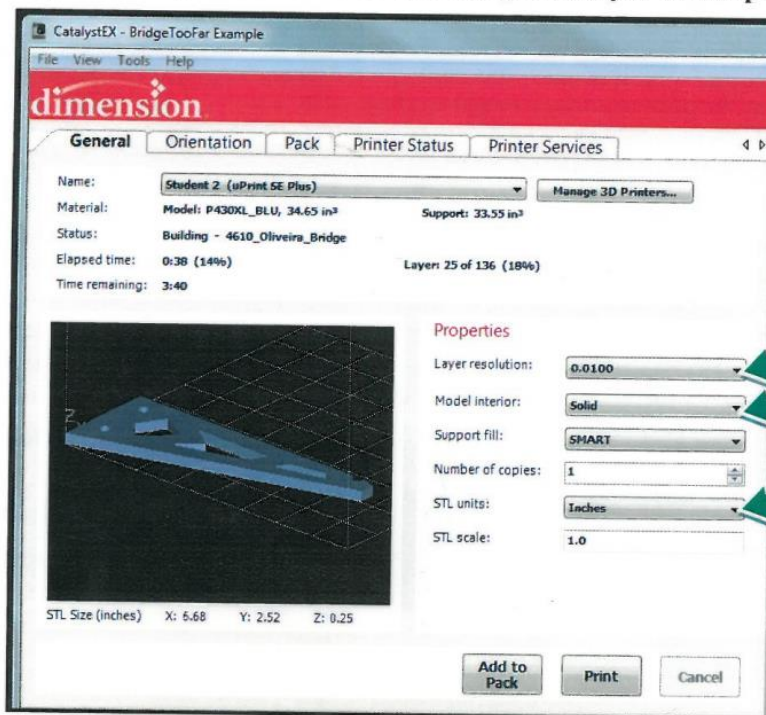
How to 3D Print Parts on the Student uPrints in the Rapid Prototyping Lab

1. You will first need to save your 3D CAD part or assembly file as a stereo-lithography STL file. To do this in SolidWorks, simply go into the File menu and select "Save As..." then select "STL (*.stl)" as the file type. Then click on the "Options..." button and choose the "Fine" resolution and make sure that the units are the same as the ones used in your part or assembly. Click "Okay" and "Save".



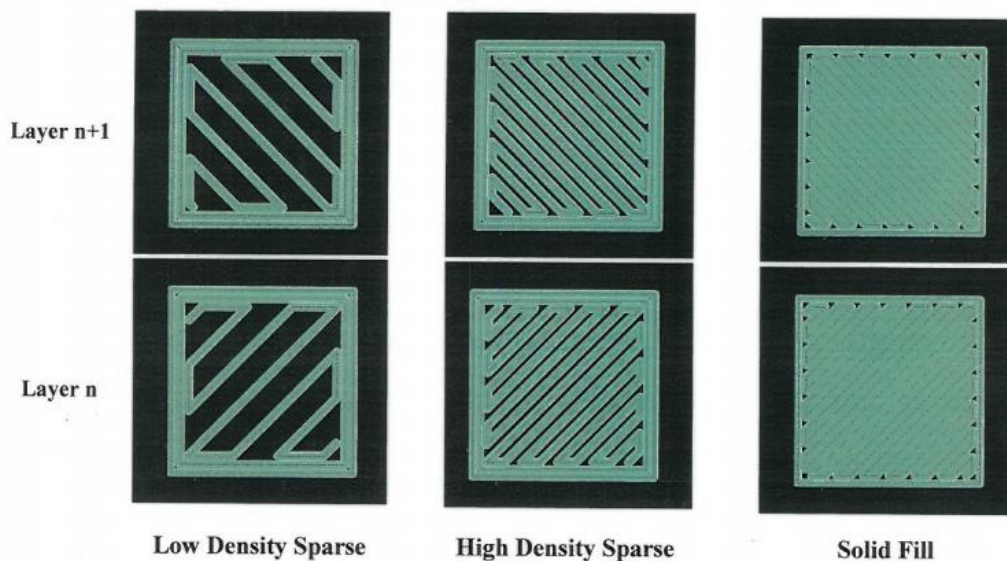
Note: In Autodesk Inventor, you can save a part file or an assembly file as an STL file by going into the Inventor PRO menu in the upper left corner of the screen and selecting Export → CAD Format. Select STL from the "Save as Type" menu. It is very important that you then go into the options menu and make sure that the correct units of measurement are selected (e.g. inches if your part was created in inches). Once the file type is set to STL and the units are correct, select "Save" to create your STL part file.

2. Now you will need to open this STL file in the CatalystEX software program. Note: You may need to search for CatalystEX in the Windows Start menu the first time that you log in. In CatalystEX, go into the File menu, select "Open STL...", and find the STL file of the part that you wish to print.



3. You will now be in the “General” tab (along the top) and have the following options available to you on the right side of the screen...

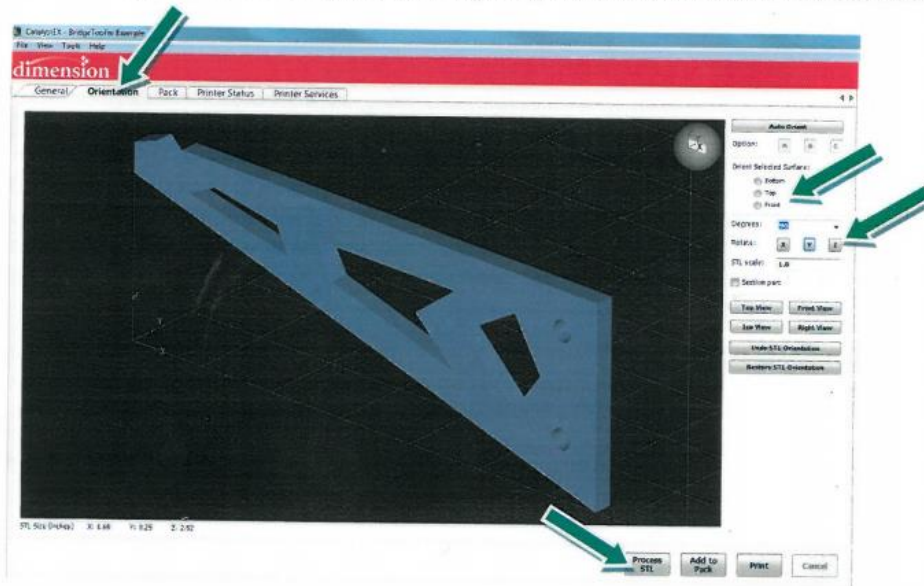
- **Layer resolution:** 0.0100 in or 0.0130in – These specify the height of each layer in the Z-direction (up and down) as your part is printed out one layer at a time. If you want the features of your part to have better resolution (i.e. finer detail in the up and down direction e.g. if you were printing text vertically in a part), you should choose 0.010in layers (the smallest layers). If your part does not have small features in the vertical direction and you want it to be printed more quickly and for it to be slightly stronger, then select 0.013in layers. Note that if you want to print multiple parts out at the same time (which usually shaves around 10-25% off of the total printing time) all of the parts must have the same layer resolution. In other words, you cannot print out a 0.010in part and a 0.013in part at the same time in the same pack.
- **Model interior:** Sparse – low density, Sparse – high density, or Solid – These specify the hatch pattern density with which the interior of your part will be filled. The uPrints always print a layer (sometimes two) around the outer surface/perimeter then automatically fill in the interior space with 45° diagonal lines, which alternate their direction after each layer. Below is an example of different fill densities and their alternating 45° patterns when printing a small cube.



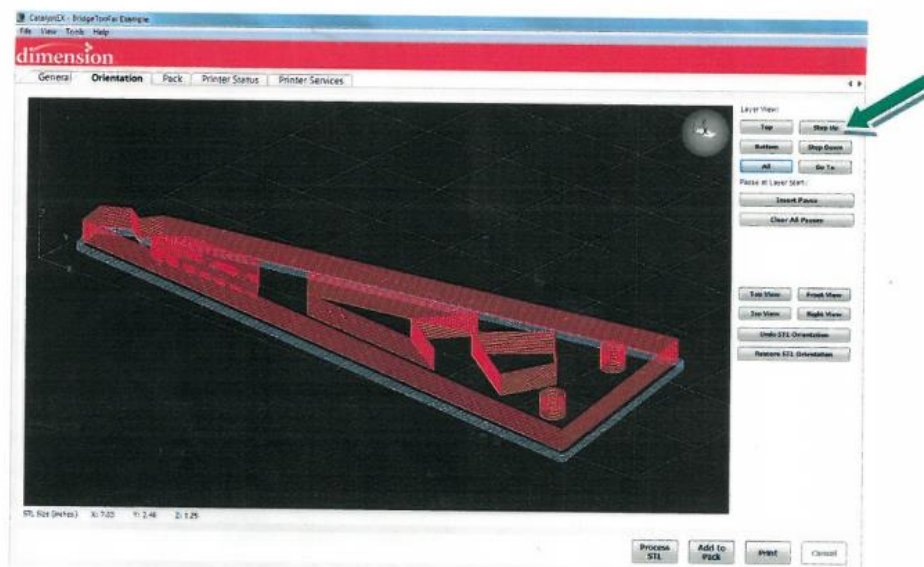
Note: For optimum strength, you should obviously choose to make your part solid. However, if you try to print a very large (e.g. >4in diameter), flat, solid piece, it may warp as it prints and cools and bend up off of the printer tray. In order to reduce weight and material usage, you could choose a low density sparse fill. Note, however, that if you put a sparse filled part in the Sodium Hydroxide bath, it will soak in the solution like a sponge and could take several days to fully dry once it is removed from the bath. When in doubt, you should probably print with a solid fill as your default.

- **Support fill:** Basic, SMART, Surround – You should almost always select the SMART support option. The Basic option is an older version of SMART and not as efficient (i.e. don't use it). In the rare case that you are printing a very delicate part that may need extra support (e.g. if you are printing a pencil shape straight up and down) you could select “Surround” to have a cocoon of support material printed out around the object. Of course, using the Surround option will significantly increase the total amount of support material that your print consumes and therefore the cost of the print.
- **Number of copies:** You should always leave this as 1. If you want to make multiple copies, it is better to do this later on when you get to the “Pack” tab.
- **STL units:** Set this to match the units that you used when you exported your STL file.
- **STL scale:** You can increase or decrease the size of your part as a fraction and/or multiple of its original size.

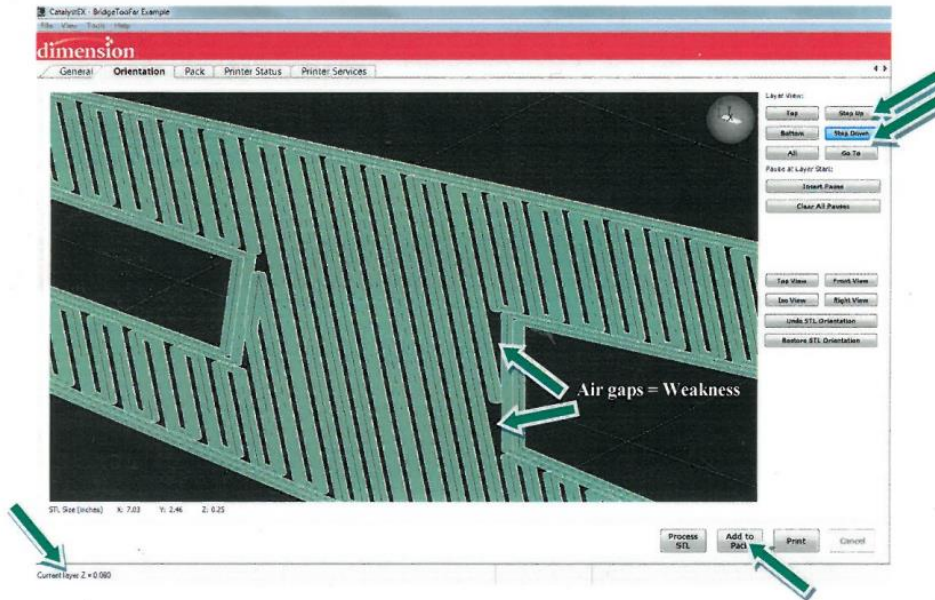
4. Now you will need to select the proper orientation of your part. Click on the Orientation tab along the top of the screen. The “Auto Orient” button at the top positions the part in such a way as to minimize the amount of support material that needs to be used. If you are creating a functional part (i.e. something that needs to withstand stress and strain and not just look pretty), *you should not use the Auto Orient option.* Fused Deposition Modeling inherently creates an anisotropic part (i.e. FDM-printed parts do not have the same material properties in all directions). Under load, parts typically tend to fail along their layers (i.e. where the layers are fused together). Thus, you should orient your part in such a way that a load will be distributed along or across long lengths of continuous plastic strings in the horizontal X and Y directions and not across vertical layers in the Z direction (this will become more obvious when you visualize the filament string patterns in the next step). You can use the “Orient Selected Surface” option to click on the face of your part that you want to set to be the bottom, top, or front. You can also rotate your part by an exact angle around the X, Y, or Z axis. When you think you have optimally oriented your part, click on the “Process STL” button. This generates the 3D printer’s toolpaths, which determine the internal structure of your part.



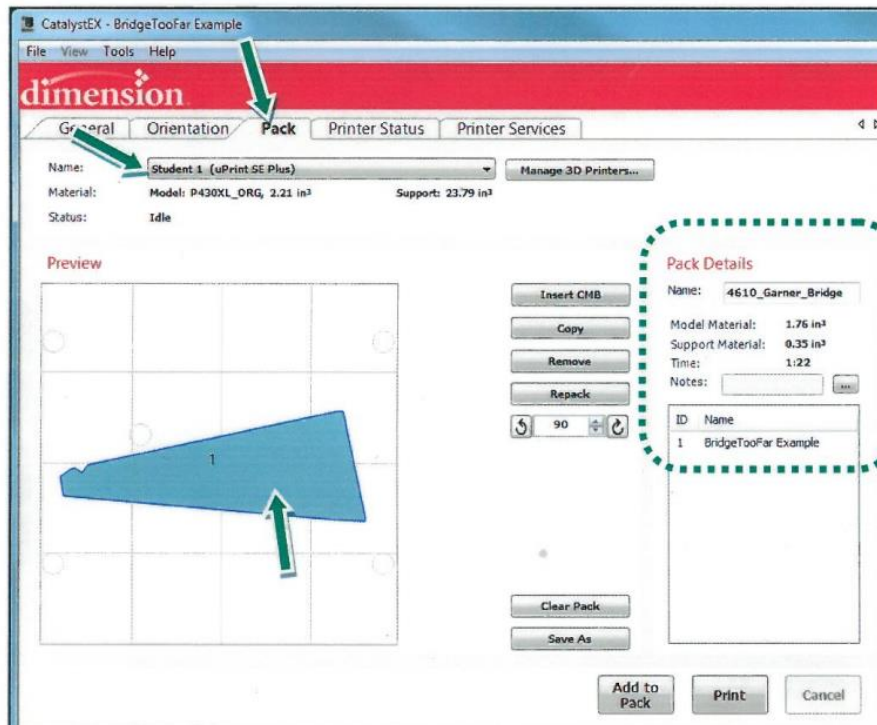
5. After processing the STL, notice that the orientation tools on the right have now been replaced with Layer View options. Click on the “Step Up” button.



- You can now see exactly how the ABS plastic (green) and support material (blue) will be extruded to make each layer of your part. Use the Step Up and Step Down buttons to examine each layer of your part. Note that the current Z position is displayed at the bottom left of the screen. You can click on the part and zoom into areas of interest using the mouse wheel. Notice that your part will not be filled in perfectly in all places. Pay especially close attention to areas of your part that you anticipate will experience high stresses. (Note: It is always a good idea to run a quick Finite Element Analysis (FEA) on your parts in SolidWorks or Inventor *before* you print them!) If there are large gaps (i.e. weaknesses) in important places, then click on the Undo STL Orientation button and repeat Step 4 to try to find a more optimal orientation for your part.



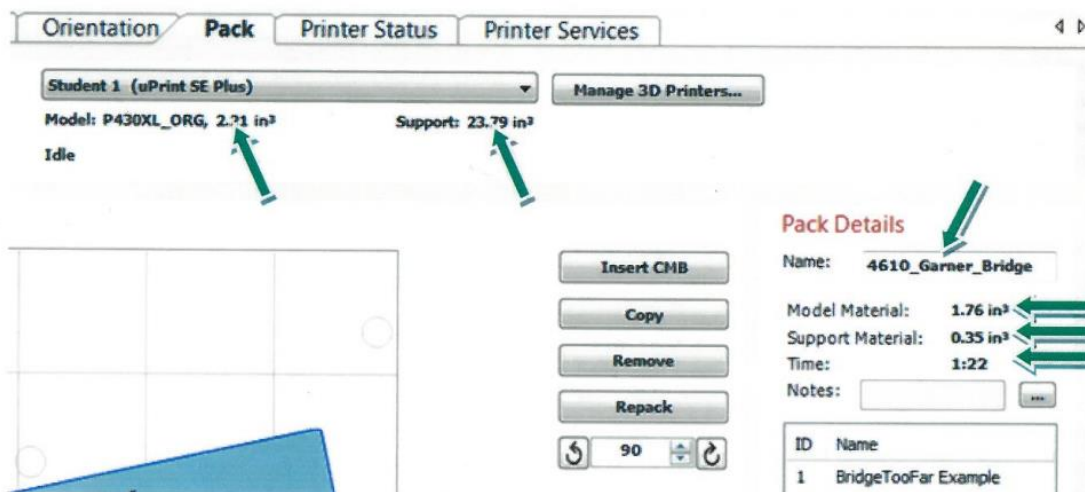
- Once you are happy with your part's orientation and toolpaths, click on the "Add to Pack" button. Then click on the Pack tab at the top of the screen. Select the printer that you wish to use (note the color of the model material ORG=orange and BLU=blue).



You will see the shape of your part laid out on a grid (the same grid that is on the black plastic trays in the uPrint that the parts are printed on). Examine the black tray in the uPrint that you wish to use and make sure that it is inserted properly so that it is lying flat and that the blue tabs are in their full upright and locked position. If the previous print has not yet been removed, the screen will say “Completed [previous pack name] Remove Part and Replace Model Base”. If this is the case, simply follow the instructions in Step 13 to remove the previous part. Note: In general, you should only replace a black tray with a brand new one if the uPrint fails to calibrate during its warm up procedure before it starts printing.

8. At this point, you can repeat Steps 2-7 to add more parts to your pack. You can also make multiple copies of a part in the pack by selecting it and clicking on the “Copy” button. Note: Because of the way the uPrint switches back and forth between its model and support material extruders, you can save around 10-25% percent of total printing time by printing lots of parts at once versus printing them all separately. (Hint: If you’re in a rush, parts placed in the upper left portion of the tray print slightly faster because the print head doesn’t have to travel as far after switching between model and support material extrusion modes.)
9. When you are happy with your pack and are finally ready to print, take a look at the total amount of model and support material that your pack will consume and the total time that it will take to print. The lab charges \$7.50 for each cubic inch of model and/or support material consumed. (Remember: Think twice print once!) When you are printing a part for a course assignment, you need to change the pack’s name to the 4-digit course number (e.g. for MAE 4610, you would enter 4610) then use an underscore “_” and type your last name. Then type another underscore and add a word or two that describes your part(s). Do not type “MAE” at the beginning of your pack name (it is implied since only students in MAE courses may use these printers). The uPrints automatically keep a log of these pack names and this is how the lab keeps track of who is printing what and how much needs to be charged for the materials used. If you are printing a personal part or a part for a research lab (with a PTAO), see additional instructions by the computer.

Note that the amounts of model and support material left on the uPrint's spools are displayed under the uPrint’s name on in the upper left portion of the screen. If your print will be consuming more than these, then it is likely that your part will automatically pause mid-print and wait for the spools to be changed. Once the machine has automatically paused and is asking for more model or support material to be loaded, please see Dwight or Prof. Garner for assistance. Once new material has been added, your part will continue printing right where it left off as if nothing had happened.



10. Make sure that the printer that you wish to use has finished printing its previous part. Please do not hit print to add a part to a busy printer’s queue and then leave the lab hoping that someone else will start your print for you. Once you are confident that you have followed all of the previous steps exactly (*especially the pack naming protocol*), click on the “Print” button to send your pack to the uPrint that you have selected. Now close the CatalystEX program. (Note: It is very important that you remember to close the CatalystEX

program since only one user can have it open at a time. Thus, if you forget to do this, the next person to log in will not be able to open CatalystEX and the entire operating system will need to be rebooted.)

11. Finally, go to the uPrint to which you sent your print job. Make sure that *your* pack name appears on the screen. Make sure that there is an empty black plastic tray in the uPrint and that the blue tabs are in the upright and locked position, then press the “Start Print” button. Note: Occasionally these uPrints may encounter a support material jam while printing the first few layers of a part (i.e. the white base/stage on top of which the ABS part is eventually printed). If this happens, ignore it and simply press the continue button on the screen to resume the print. It is a good idea to watch or check in on your print during the first 5-15 minutes of printing (or more specifically, until the first bits of blue or orange ABS material start to be laid down on top of the support material base – after which point, support material jams seem to be much less of an issue). Note that after the calibration (homing the stepper motors) and warm up procedure are completed, you can toggle between the “Show Time” and “Show Material” options on the uPrint. For some reason, this countdown timer is not 100% accurate and parts often finish a 5-10 minutes ahead of schedule. Make a note of when your part will be completed and plan to return to the lab at that time to promptly remove your finished part.

Removing a part after 3D printing is completed:

12. When your print is finished the screen will say “Completed [your pack name] Remove Part and Replace Model Base”. Open the uPrint’s door and turn both of the blue tabs 90°. Now remove the black tray by sliding it out toward you. *Make sure that you close the uPrint’s door at this point.* (Whenever the uPrint’s door is open, not only does it let heat escape, but it also lets in moisture that can lead to more frequent support material jams.)
13. Now go back to the sink area of the lab and make sure that you put on safety goggles before crossing the yellow and black lines on the floor. Place the black tray up against the wall on the countertop and use a putty knife and mallet to hammer your part off of the tray. Please be careful not to gouge into the tray while you do this. Once the part is off of the tray, you can either try to peel the support material off manually (recommended for simple, mostly-flat parts since it’s quicker and less messy) or have the support material dissolve away automatically in a heated Sodium Hydroxide (aka lye) solution bath. If you decide to use the bath, put gloves on and then carefully open the lid to the student-use Cleanstation bath. By tilting the lid forward about 120°, you can get it to rest in the built-in plastic groove on the top while leaning it up against the wall. Now use the steel handles to lift the inner section of the Cleanstation up out of the lye solution. When it is at the top, it can be locked by pulling it slightly toward you. Place your part in one of the metal mesh boxes and close the lid to make your part easier to retrieve later (sometimes small parts can get sucked into the machine’s pump and damaged or lost). Note: You can leave your part in the bath indefinitely without harming the ABS plastic. However, please try to be considerate of others and remove your part as soon as possible after all of the support material has been removed. Use the metal handles to slowly lower the inner portion of the Cleanstation back down into the lye solution. Replace the lid and then press the “On/Off” button until 4:00 (4 hours) is displayed and the recirculation pump turns on.

BE SURE TO CLEAN UP ANY MESSSES YOU MAY HAVE MADE!!!

14. Now place the empty black tray back into the uPrint. The screen should say: “Completed [your pack name] Part Removed?” At this point, you should select “Yes”. Congratulations, you have successfully 3D printed something!

Additional things to note:

- These student uPrints should be operated on a first come, first served basis. If both printers are busy printing when you arrive, do not send another print job to them (i.e. do not add a new print to their built-in print queue) and then leave the lab. While these printers can keep track of multiple print jobs in their queue, it is not fair to make other users have to print, wait for, and then remove your parts for you.
- If you see someone not following any of these steps/rules, making a mess and not cleaning up after themselves, not wearing gloves or goggles in the sink area, and/or doing anything that seems unsafe or bad for the equipment, please alert Dwight and/or Prof. Garner immediately.