3D PRINTING IN THE DFAB LAB

If you're relatively new to 3D printing, or if you run into problems with printing, be sure to read the **DESIGN FOR 3D PRINTING** guide, also found in the dFab Lab.

OVERVIEW

The dFab Lab has five **StrataSYS uPrint SE Plus** printers. Each uPrint has a maximum buildable volume of **8"x8"x6"**. The uPrints print with two materials: ABS plastic modeling material, and a support material that dissolves in a chemical bath. The charge for 3D printing is **$5 per cubic inch of material**, which includes modeling material as well as support material, as calculated by GrabCAD. You will need to pay for your print before starting the print.

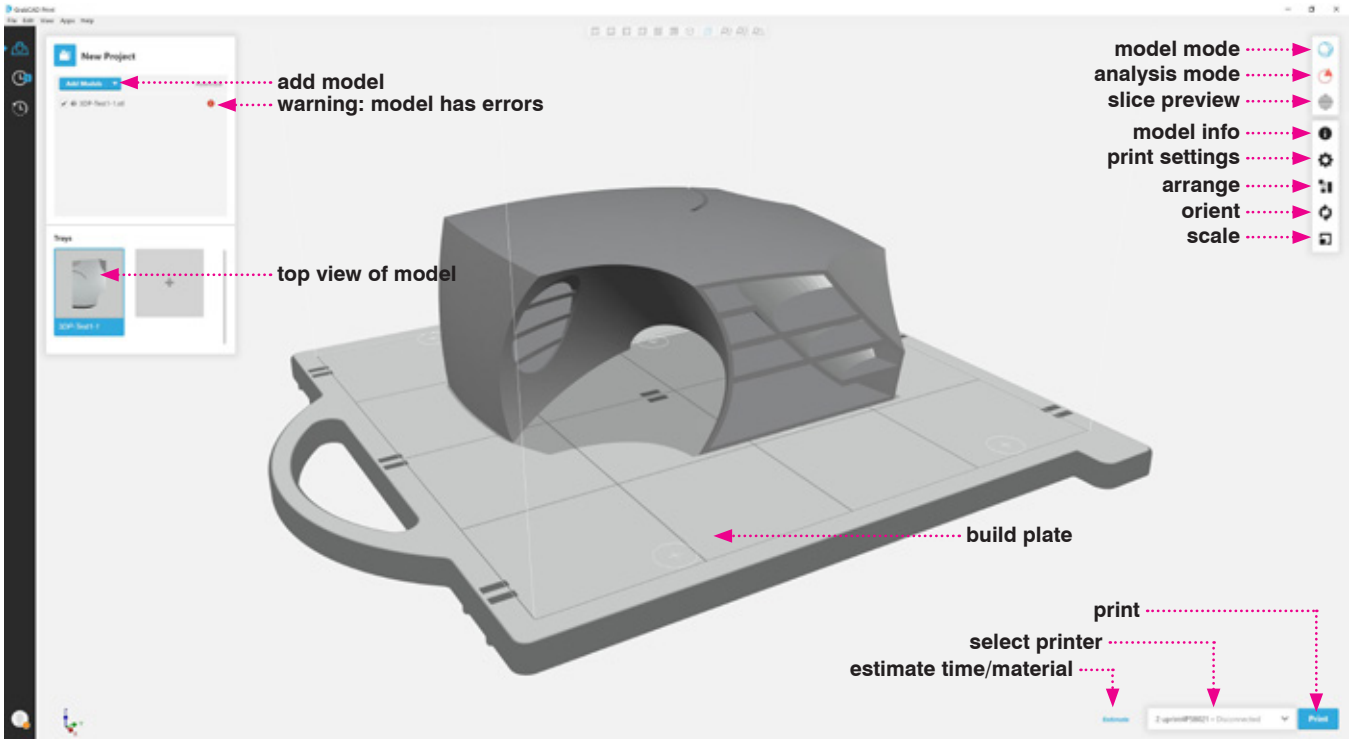
The dFab Lab uses GrabCAD slicing software (installed on all computers next to the printers). This guide will walk you through the steps of importing your model into GrabCAD, slicing it, printing it, and removing the support material in the chemical bath.

IMPORTING YOUR MODEL INTO GRABCAD

1. Open GrabCAD.

2. Click ADD MODELS and select your STL file. Your model should appear on the build plate; visually double-check that it is the right scale, relative to the build plate. You can add multiple models to the build plate and print them simultaneously; when the printers are in high demand, this helps more people to print.

Take a moment to familiarize with the GrabCAD interface, and locate the buttons below.

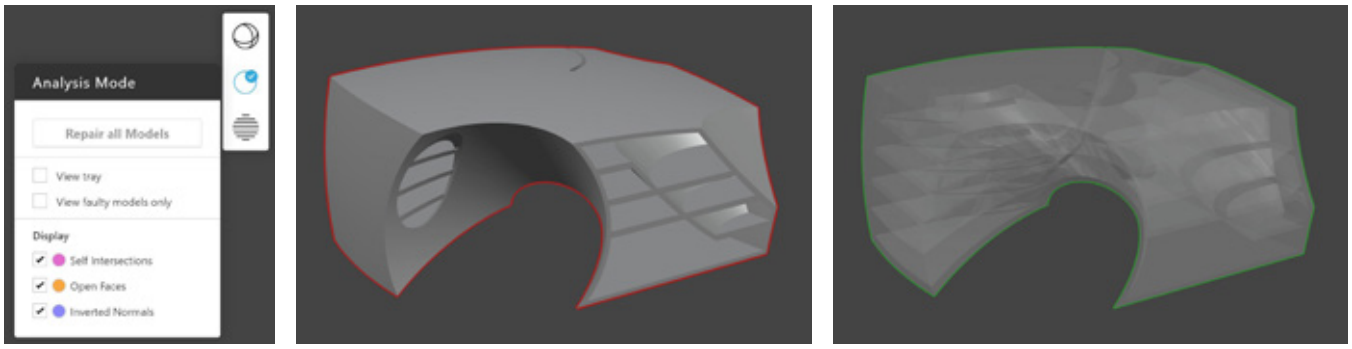


GrabCAD interface, with imported model

ANALYZING AND REPAIRING YOUR MODEL

1. After importing your model, you may see a red warning symbol next to your model's filename. This means there are some errors in your geometry. If these are minor errors, you can easily fix them in GrabCAD. Switch from Model Mode to Analysis Mode; any models with errors will have a red outline. The errors might be visible (refer to the color-coded error types in the window), or they might be too small to see. Click REPAIR ALL MODELS to repair the model.

After the model is repaired, it should have a green outline. Visually inspect the model to make sure the "repairs" didn't inadvertently alter the model.

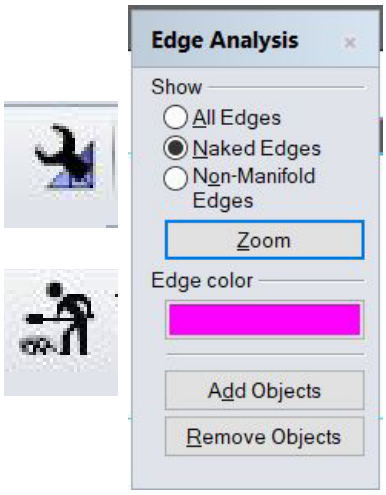


Analysis Mode: before and after

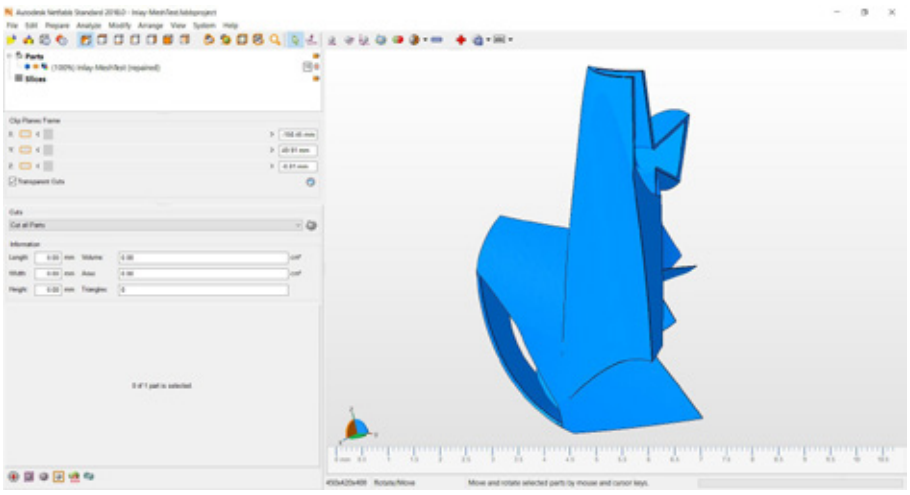
2. If REPAIR ALL MODELS doesn't work, there are a couple options for fixing your geometry:

Netfabb is a mesh repair software produced by Autodesk. It is free to download for students at:
www.autodesk.com/education/free-software/featured#free-software.
If you have trouble installing Netfabb, try using Autodesk's download manager.
Click on AUTOMATIC REPAIR (red cross) > EXTENDED REPAIR.

Rhino has some methods for analyzing and repairing meshes. You can use the MESHREPAIR wizard. The SHOWEDGES command is also helpful for highlighting naked edges, which can then be repaired using FILLMESHOLE. Fixing meshes in Rhino can be really tedious, and sometimes you will just need to re-model your geometry. It's sad but true that some geometry is just bad beyond repair.



Rhino



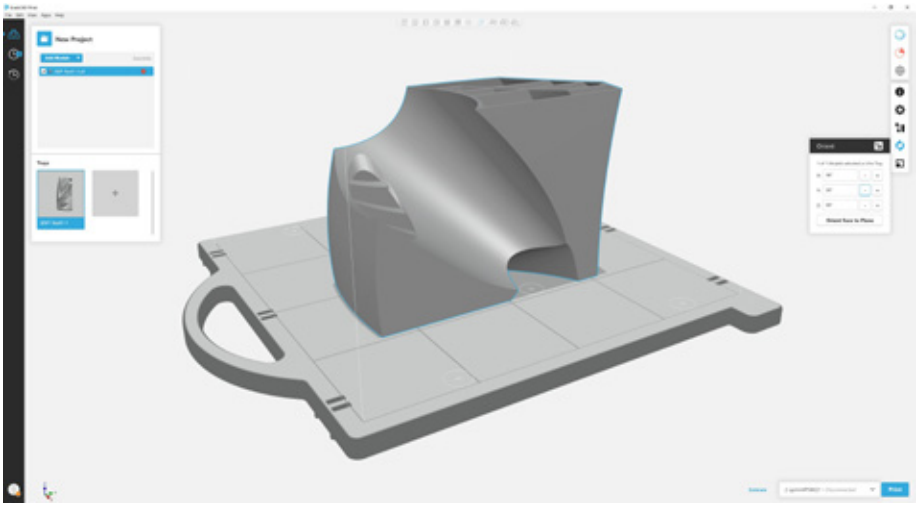
Netfabb

ORIENTING, SCALING, AND POSITIONING YOUR MODEL

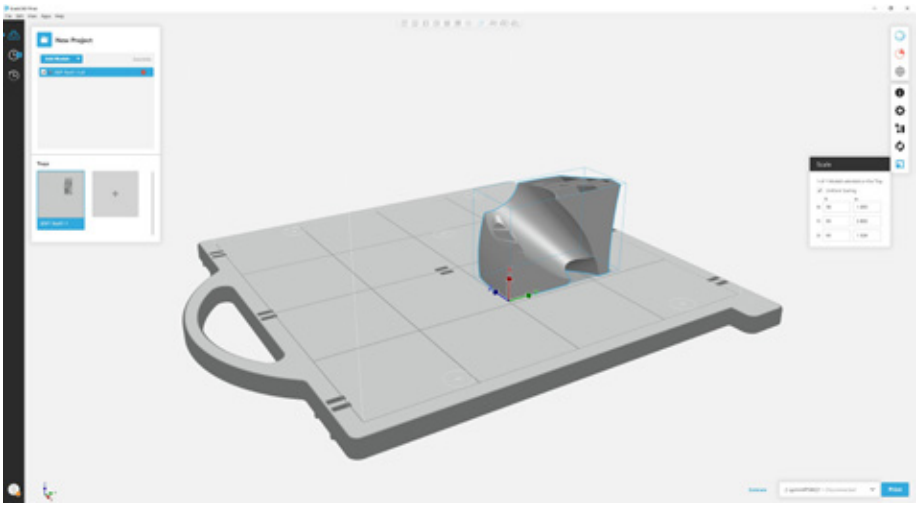
1. After you've imported your model, visually check that it is the right size. By default, GrabCAD imports models scaled to **inches** (an STL file has no inherent units). If your model is huge, it's possible that you saved it in millimeters; if your model is tiny, it's possible that you saved it in feet. You can use the MODEL INFO button to select the desired units for the model, and GrabCAD will scale it accordingly. Note that you must **click on your model to select it** before changing the units or any other options.

2. Make sure your model is in the optimal orientation for printing. Orientation can have a significant effect on print quality, the strength of the printed part, and the amount of support material used (and therefore the time and cost of printing). Using the ORIENT button, you can either rotate your model in 90° increments in any direction, or select a face of the model to orient to the build plate. In the example shown, it made more sense for my floorplates to be oriented vertically, so that I wouldn't need support material between every floor. If you're unsure of the optimal orientation of your model, you can try slicing it in a few different orientations, and compare the material estimates.

3. In the example below, I decided that I didn't have the time or the money to do a big print, so I scaled it down 50%. Remember that volume is cubic (*length x width x height*), so doubling the size of a model might make it take 8 times as long to print, and cost 8 times more. This isn't *quite* true when it comes to 3D printing, since "solid" parts are not typically printed with solid material, but it does make a big difference.



model rotated for better printing



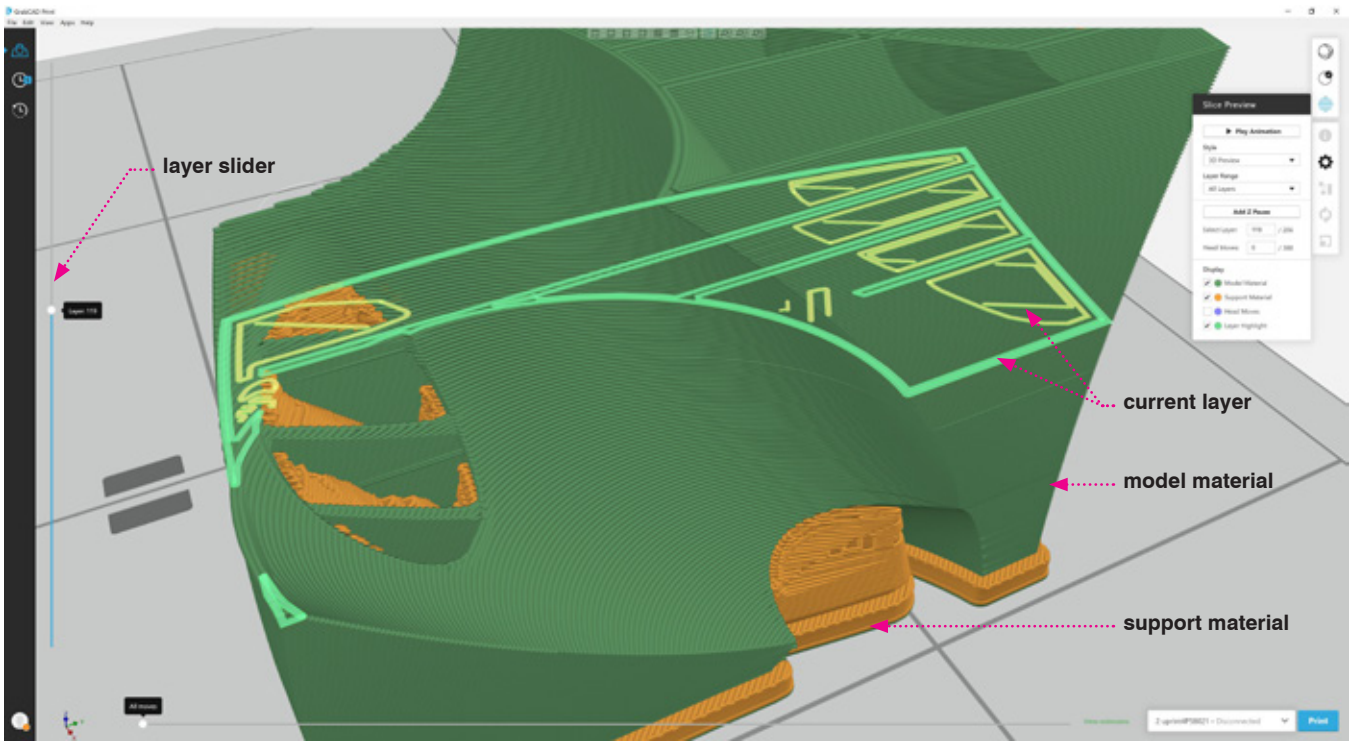
model scaled for time and money

SLICING AND ESTIMATING

1. Click on your model to select it, then click PRINT SETTINGS. There are only a couple settings:
   -Support Style: SMART minimizes the amount of support material and generally works well.
   -Part Fill Style: SPARSE - LOW DENSITY is the cheapest. SOLID is the strongest.
   -Slice Height: 0.01" gives better detail and print quality, but will take a little longer.

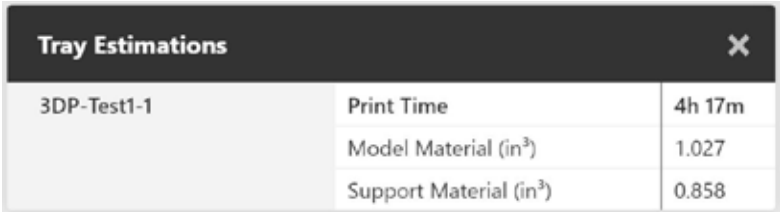
2. Click on the SHOW SLICE PREVIEW button to slice your part. This may take a few minutes. Your part should change colors when complete.

3. Take a look at your sliced part. It is color-coded to help you differentiate model, support, infill, and the toolpath for the current layer. Click PLAY ANIMATION to see a layer-by-layer animation of the printing process, which will help you check for problems. You can also drag the LAYER slider on the lefthand side of the screen to move through layers manually, and check any tricky areas. In this example, I'm noticing that the walls and floors in my model are only two filaments-thick—that is because I scaled my initial model down 50%. Those walls may be weak or of poorer print quality than if they were just a bit thicker.



4. If you don't need to re-orient the model and the toolpaths look good, click VIEW ESTIMATES near the lower-right corner of the screen. In this example, the print will take 4 hours and 17 minutes, and use a total of 1.885 cubic inches of material. At $5 per cubic inch, this is a $9.43 print. As a comparison, before this part was scaled down 50%, it was estimated to take 18 hours, and cost $50.

5. This is when you **pay for your print**. The dFab Lab will not send your job without payment.



PRINTING

1. Right next to the VIEW ESTIMATES button, you'll see the name of the printer. The dFab Lab printers are appropriately named uprint 1, uprint 2, uprint 3, etc.—notice the number on the printer's front panel. Click on the printer name, then check that you are connected to that printer (green check mark). Click on the printer again to get information on it:

2. Click on the queue and see if any jobs have already been scheduled for this printer, and others.

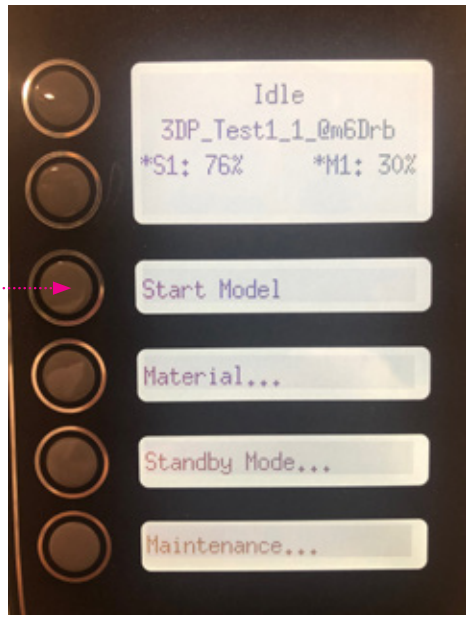
3. Check to see how much model material and support material are left (this should also show up on the printer itself), and make sure there is enough for your job.

4. Empty the blue filament waste bin located at the back corner of the printer.

5. Click the PRINT button at the lower-right corner. A green banner with "Print job queued successfully" should appear. Walk over to the printer. The screen should be flashing START MODEL; push the button. The printer will calibrate for a few minutes, then begin the job. If you look at your job in the GrabCAD queue, you'll see an estimated time of completion.

6. When your print is finished, remove the build plate from the printer by rotating the blue latches, and sliding out the plate. Remove the printed part from the build plate; you may need to use a scraper tool

filament waste bin



start model

REMOVING SUPPORT MATERIAL

1. Remove as much support material as possible using **needle-nose pliers**. Wear **safety glasses** while doing this—you don't want a flying shard of plastic in your eye. For simpler prints, you might be able to remove all the support this way.

manual support removal

protective equipment for chemical support removal

2. For more intricate parts, you will need to use a **chemical bath** to dissolve support material that is located in hard to reach areas. Even if this is the case you should **still use pliers** to remove as much support as you can—it will drastically reduce the time required for the chemical bath to dissolve the remaing support.

3. If your print is small or delicate, put it in a metal cage. This will protect it from breakage, and make it easier to find in the bath.

4. The chemicals used in the bath are dangerous, and will burn skin and cause blindness. **Put on an eye mask and rubber gloves**. Open the two covers of the bath, and gently place your print inside—do not throw it or drop it in. Close the covers.

5. Thoroughly wash the rubber gloves until the chemical solution is fully removed. Then remove your eye mask, and wash it as well.

6. Make sure the bath's power supply is turned on (right side on the bottom), then hit the power button on the main console. Set the temperature for 70° C; a timer should appear, and the bath will slowly heat up. It may take up to 24 hours for the support to fully dissolve, but often 12 hours is adequate.

7. When retrieving your print from the bath, use an eye maks and rubber gloves. Thorougly wash your model, in addition to the protective equipment. Congratulations! You now have a little piece of plastic.