Resin Printing

This document is designed to serve as a general introduction to using a resin printer. Other useful guides and videos related to more specific topics can be found at OT3d.org

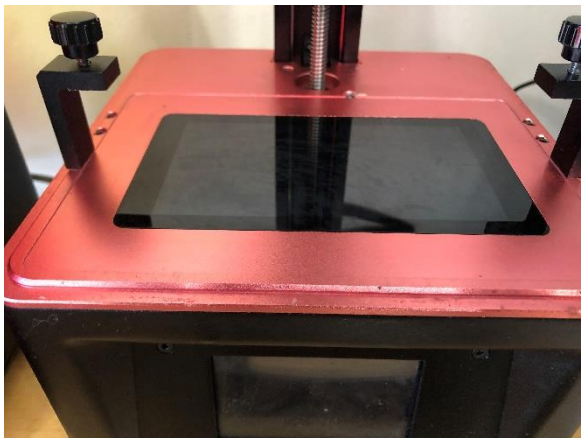
Welcome to the guide to resin printing! Let us start out by learning what a resin printer is and how it works. The specific kind of resin printer that we will be learning about today are called SLA printers. Here is a picture of a resin printer that is available at my local makerspace.

There are 5 main parts on a resin printer that I would like to introduce you to. The first of these is the build plate. Like FDM printers, resin printers need a surface to house the print as it is building. The build plate moves up and down on a rail.

The build plate lowers itself into a pool of resin, which is held below. Different printers are able to hold different amounts of resin, determining how large of a print they can manage. The bottom of the build plate is a sheet of plastic held taught through tension. With time this material may become worn and require replacement.

Resin Printing

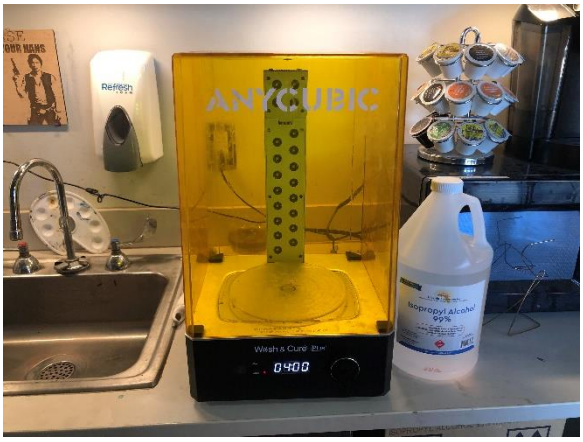
The next part is the lcd screen. You can think of this screen like a screen in a phone or tablet. It emits light in specific patterns on the lowest layer of resin. When the build plate is lowered, the light will turn on and harden or ‘cure’ the resin in between, adhering it to the build plate or previous layer.



The control screen allows you to navigate through options and select which print you would like to make when you have a flash drive or other memory device inserted.

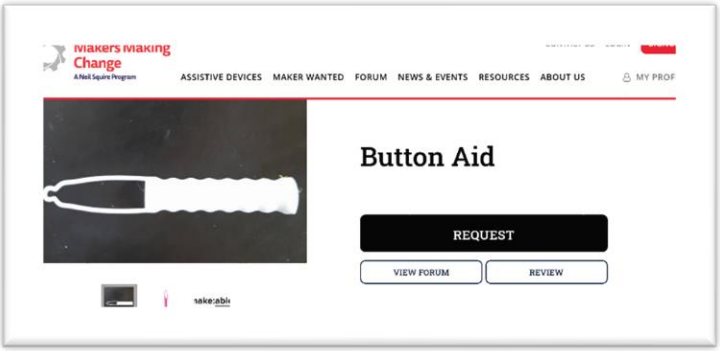
The shield is a plastic shell that limits the amount of light that enters the printing area and keeps the smell and fumes of resin contained to the print area.

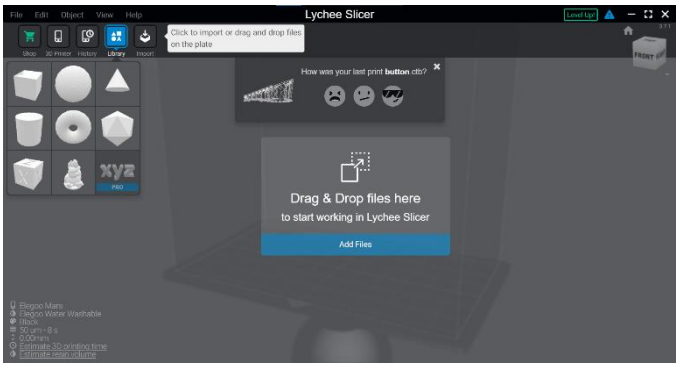
Helpful Hints: Small resin printers like the one pictured in this guide are very affordable and run about the same rate as many entry level FDM printers. Most resins will cost around $40 for a 1 Liter bottle.



In addition to the resin printer, you may have access to a curing machine. This machine offers two purposes, the first being a washing function to remove excess resin and clean it. It also serves as a curing station, exposing the print to additional UV light to its surface.

Now that we know about our equipment, let's dive into the process of slicing a print. To start, we can find an .stl or .obj file just like we would use for an FDM printer. For this demonstration, we will use a button hook from Makers Making Change.

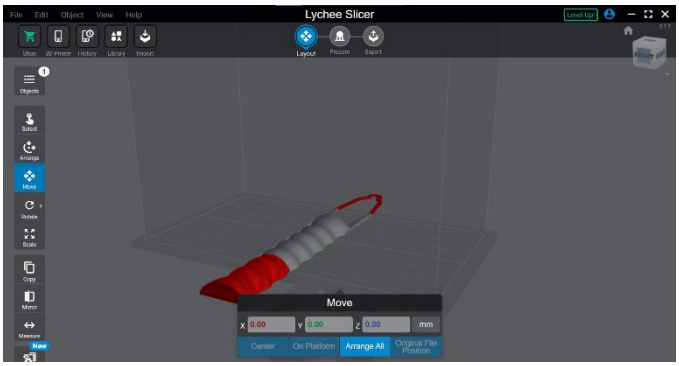


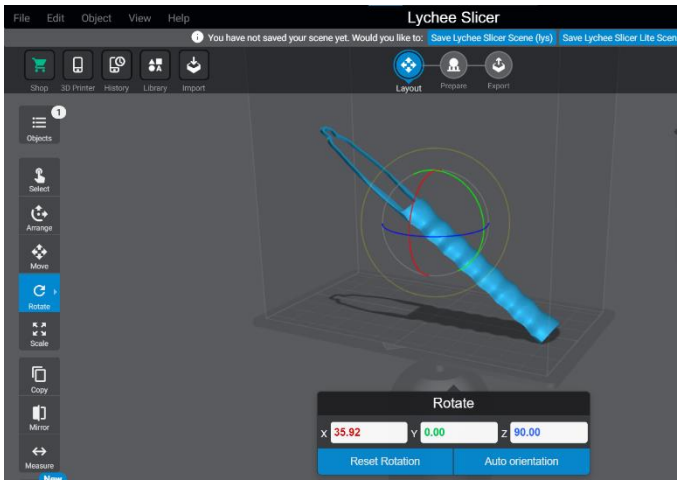


Next, we need to upload the file into a slicer program designed for resin printers. Two good options that you might try including Chutu box and Lychee. For demonstration, we will slice our file in Lychee. We can upload the file by importing the .stl or .obj file that we downloaded.

Helpful Hints: Resin can be harmful if it gets on your skin. Be sure to wear gloves when working with uncured resin.

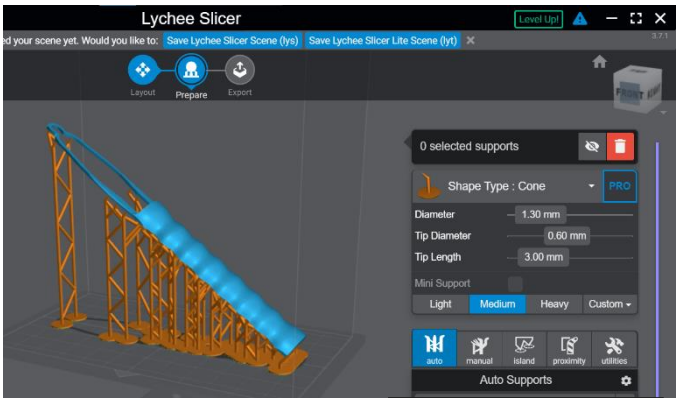
After importing the file, you may need to adjust it so that it fits on the build plate. For this file, I need to rotate it about 90 degrees to remove the red areas that show an error.



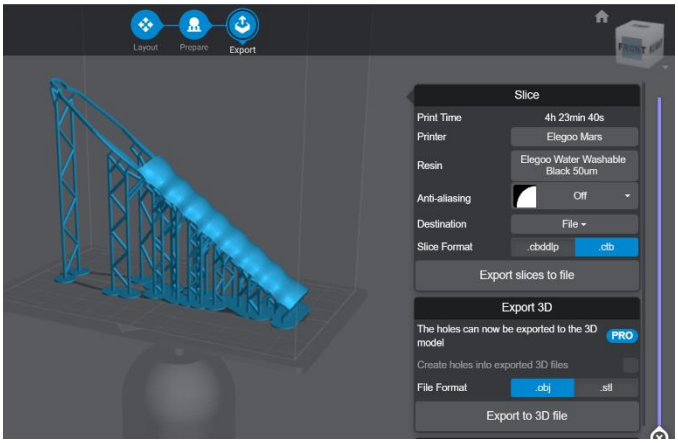


Once the object is on the build plate, you can then position it vertically. I have found the best results when the object is tilted between 30 and 45 degrees vertically. The more the object is tilted, the longer it will take to print. This is because the object's height on the build plate will also increase, requiring more cycles from the printer.

Up next, we can start to add supports to our print. Because we are printing at an angle and off of the build plate, we will need supports for our print. Fortunately, most slicers will have an option to automatically generate supports. Go ahead and select the option to do so.

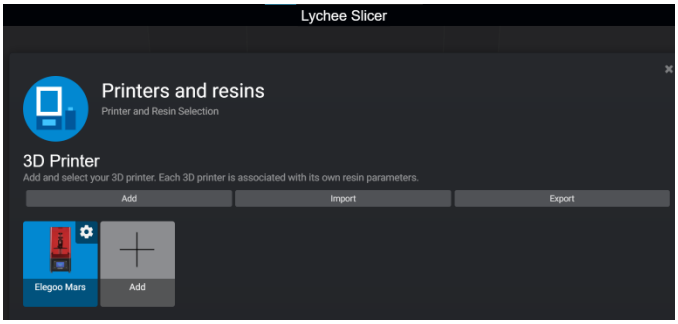


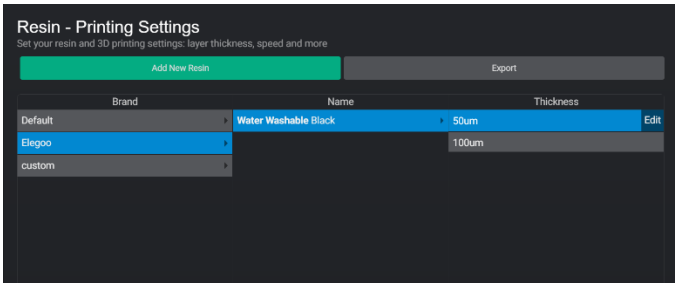
Helpful Hints: Most programs will also allow you to add supports manually, which can be useful if you feel that the automatic supports are not covering enough area or will be in places too difficult to remove.



Now that our supports are complete, we can start the process of exporting our file. After selecting the export file at the top, we can use the menu on the right side of the screen to make our final adjustments.

We need to double check two things at this stage. The first is that we have the right printer selected. Slicers will typically have a list of common printers that you can select from, which makes sure that the build plate size and available height are correct. You can do this by clicking on the box next to 'printer.'

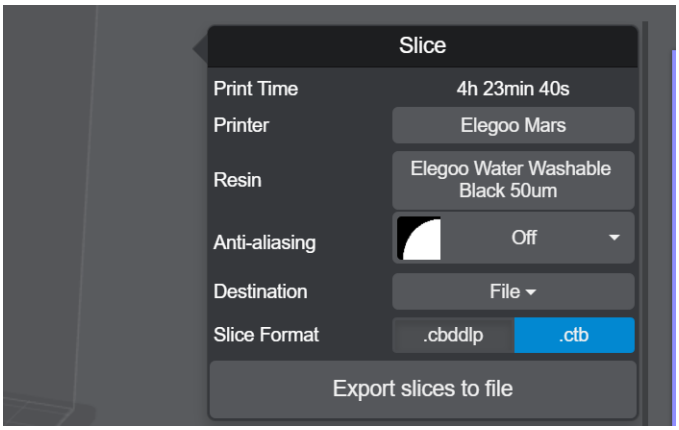




The second factor that we are going to check is our resin settings. Lychee allows you to search through known resins to select your resin's profile, or you can manually type in a resin profile by filling in the required fields from information on your resin bottle or from the manufacturer website.

Helpful Hints: If you start having poor quality prints, first thoroughly clean your printer. If issues persist, check if your resin has gone bad, check the bottom of the resin pool for imperfections, and look at the lcd screen through your phone's camera. Each of these items is susceptible to wear over time.

Now that our settings are adjusted, we can export our file as a .ctb file and clicking the button that says, ‘Export Slices to File’. We can then save that to a flash drive or other memory device used by your printer.



Now it is time to prepare the printer. Be sure to clean off the screen, build plate, and bottom of the resin pool with an alcohol wipe to remove any old resin or debris. If there is still resin in the pool from a previous use, use a gloved finger to stir the resin and feel for any lumps. Wipe dry with a clean microfiber cloth if needed.

Thoroughly shake your resin before pouring up to the fill line of the resin pool. For larger prints, keep an eye on the amount of resin in the pool as you may need to pour more in. Leftover resin can be reused by pouring it back into the bottle.

Helpful Hints: You will have the best results if you start from a fresh pool of resin each time. If you still have resin from a previous print, you can use a funnel to pour it back into your bottle to shake and reuse. Follow standard cleaning procedure before printing.

Resin Printing

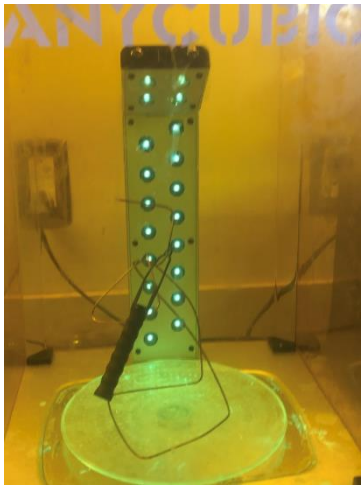
Now that the printer is prepared, we can use the menu screen to select the file that we want to print. Many printers will show a small preview image of the print that you are selecting. The printer will then run until the print is complete.

Now that the print is complete, we can start the finishing process. The first step is to wash the print off to remove any remaining uncured resin. We can do this using our curing machine by placing the part in an alcohol bath and running it for around 4 minutes.

Once the print is done washing, you can take some time to remove the supports from the print. This can be done after curing if desired, but it is a little bit easier at this stage because the supports have not yet fully cured.

Helpful Hints: If you are using water washable resin, you can perform the same steps using water in your curing station. Just be sure that the print is fully dry before curing it, as water will not evaporate as quickly as alcohol.

Now that the print is free of supports and washed up, we can cure it. Swap the wash station out for the rotating curing platform and place the print so that it has its surfaces exposed to the UV lights. Run the curing station for 3-4 minutes or follow your resin manufacturers specifications.



Once the print is done curing, you can finish it like you would an FDM print. You can use sandpaper to smooth any indentations from supports and paint it as desired.

If you have any additional questions or would like to learn more about 3d printing for occupational therapy educators, students, and practitioners, please check out OT3d.org. You will find helpful resources, visual guides, and videos. Happy Printing!