

Instructions for Resin (SLA) 3D Printing

Last Updated September 2, 2025

- 1. Stereolithography (SLA) 3D Printing**
- 2. Choose Material**
- 3. Design Parts**
- 4. Slice Parts in PreForm**
- 5. Print Parts**
- 6. Remove Parts from Build Plate**
- 7. Clean/Return Build Plate**
- 8. Wash Parts in Form Washes**
- 9. Hand Wash Parts with IPA**
- 10. Dry Parts**
- 11. Post-Cure Parts and Remove Supports**
- 12. Sanding and Polishing**

1. Stereolithography (SLA) 3D Printing and Formlabs Form 4

3D printing, also known as Additive Manufacturing (AM), is an incredible technology to manufacture parts that are difficult to fabricate with other methods. **If your part can be easily manufactured with our other equipment, we will redirect you to use those tools.** Parts that are inappropriate, weapon-related, or used to contact food are also not allowed to be printed at the Invention Studio.

Resin 3D Printing is called **Stereolithography (SLA)**, and it builds a 3D object by selectively curing liquid photopolymer resin with UV light, layer by layer, onto a build platform. The part is printed upside-down as the build plate is lowered into a tank of liquid resin to print each layer. It is able to achieve much higher resolution than FDM 3D printing because the UV curing is much more precise than extruding melted filament.

Because resin is more expensive than FDM filament, **we reserve resin 3D printing for only the parts that need it.** This includes high-detail models with intricate geometries or parts that require the mechanical properties offered by our Engineering Resins.

The Invention Studio has **8 Formlabs Form 4s** to support your resin printing needs. As opposed to some SLA printers that trace layers with a single laser, Form 4s use an LCD screen to print one layer at a time. As a result, they are some of the fastest and most dependable machines on the market.



2. Choose Material

All the resins stocked by the Invention Studio, along with their material properties, are given on the 3D Printing page on the Invention Studio website.

The [Formlabs Material Comparison](#) page can also assist in comparing the mechanical properties of different resins to select the best material for your application.

3. Design Parts

The [Form 4 Design Guide](#) gives specifications for designing parts, such as minimum wall thicknesses, minimum clearances, and minimum hole diameters. Formlabs provides guides for specific materials as well (linked in the spreadsheet above) that may differ from these general guidelines.

Feature Size	Dimensional Tolerance
1 – 30 mm	± 0.15 %
31 – 80 mm	± 0.2 %
81 – 150 mm	± 0.3 %

Desired Fit	Clearance Gap (in)	Clearance Gap (mm)
Press	0	0
Tight	0.005	0.127
Normal	0.010	0.254
Loose	0.020	0.508

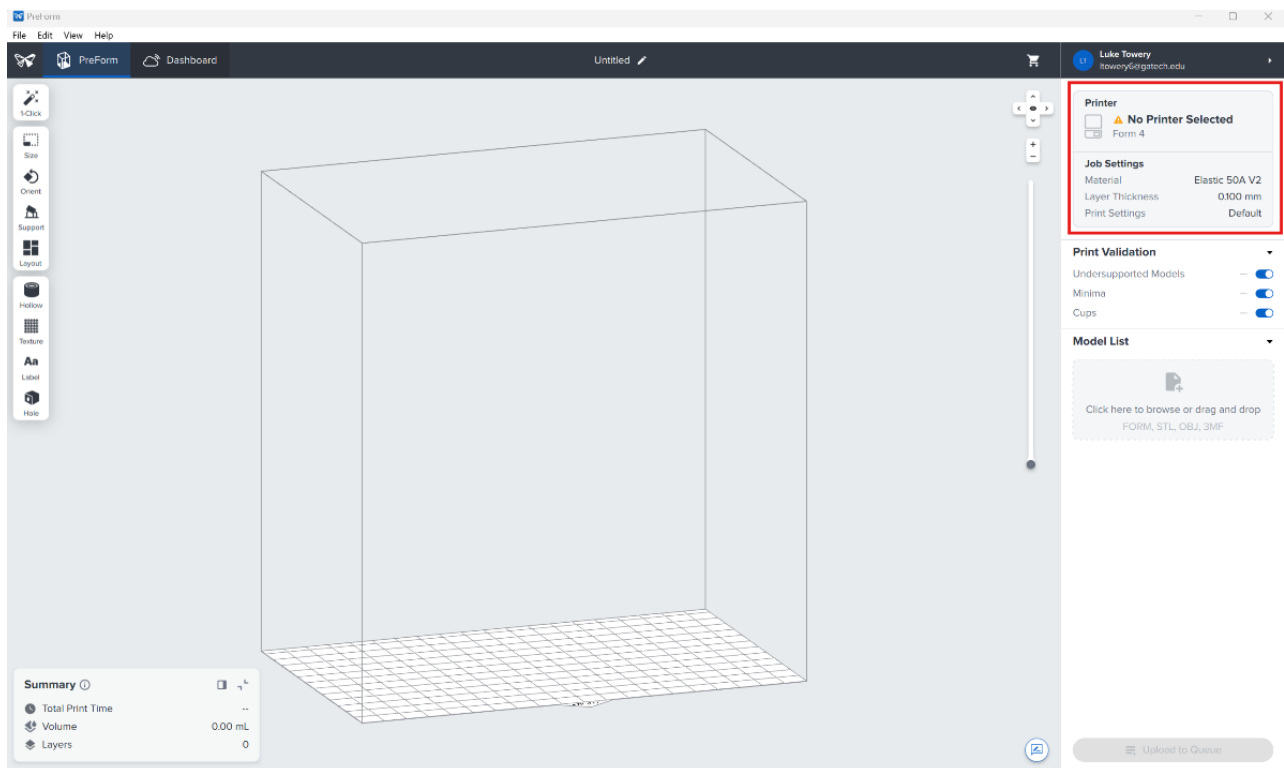
4. Slice Parts in PreForm

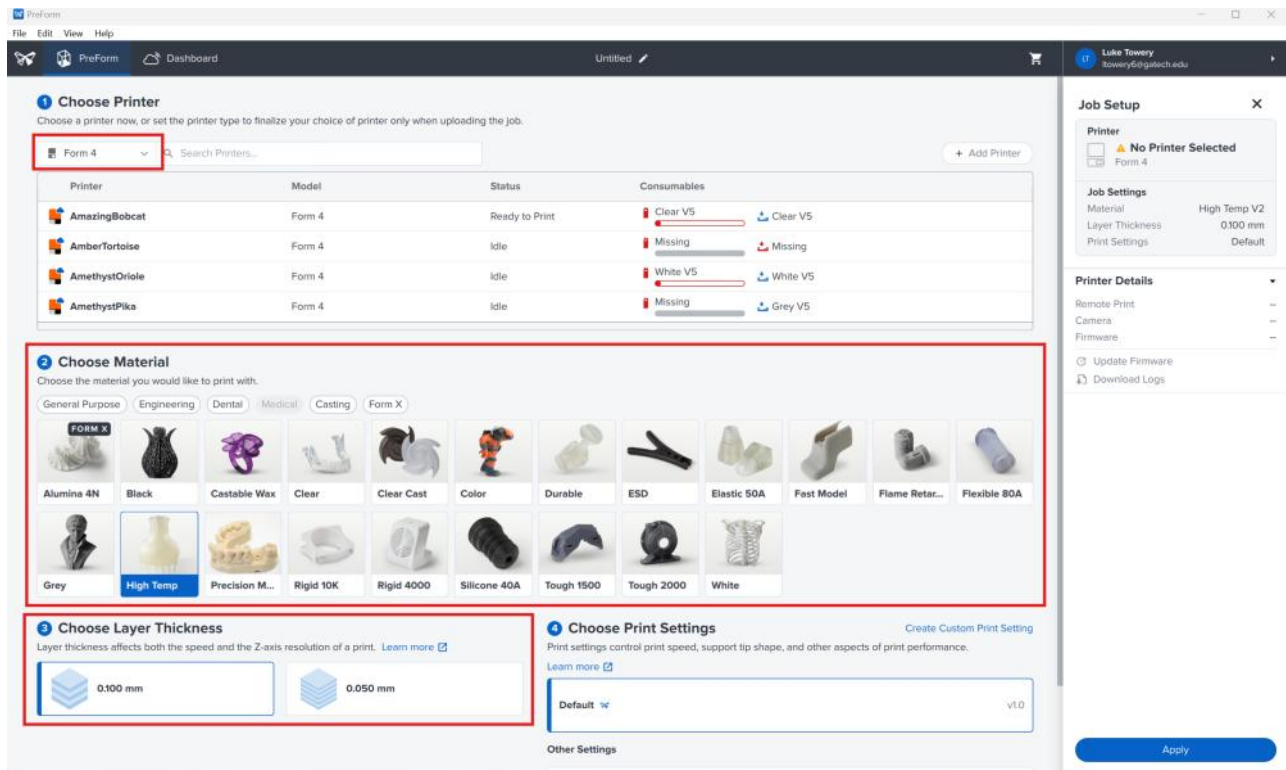
PreForm is the dedicated slicing software for our Form 4 printers, which converts your part into printing instructions for the machines. It is available on the Invention Studio computers and [free to download](#) to personal computers.

1. **Export your part(s) as an STL file** from your CAD software. Make sure that your STL is exported with appropriately high resolution, a prerequisite for taking advantage of the high resolution of resin 3D printing. For step-by-step instructions for SolidWorks, see the *Exporting STLs for 3D Printing from SolidWorks* document on the 3D Printing page of the Invention Studio website.

2. **Open PreForm and define print settings.** Select the *Data Card* and choose a *Form 4* printer, resin type, and layer thickness*. Select *Apply* after making these changes.

*Click [here](#) to learn about when to use different layer thickness and understand the different between XY Resolution and Z Resolution in 3D printing.





3. **Import your model(s)** with *File > Open*. Scale your part to the correct size if necessary with the *Size* tool.

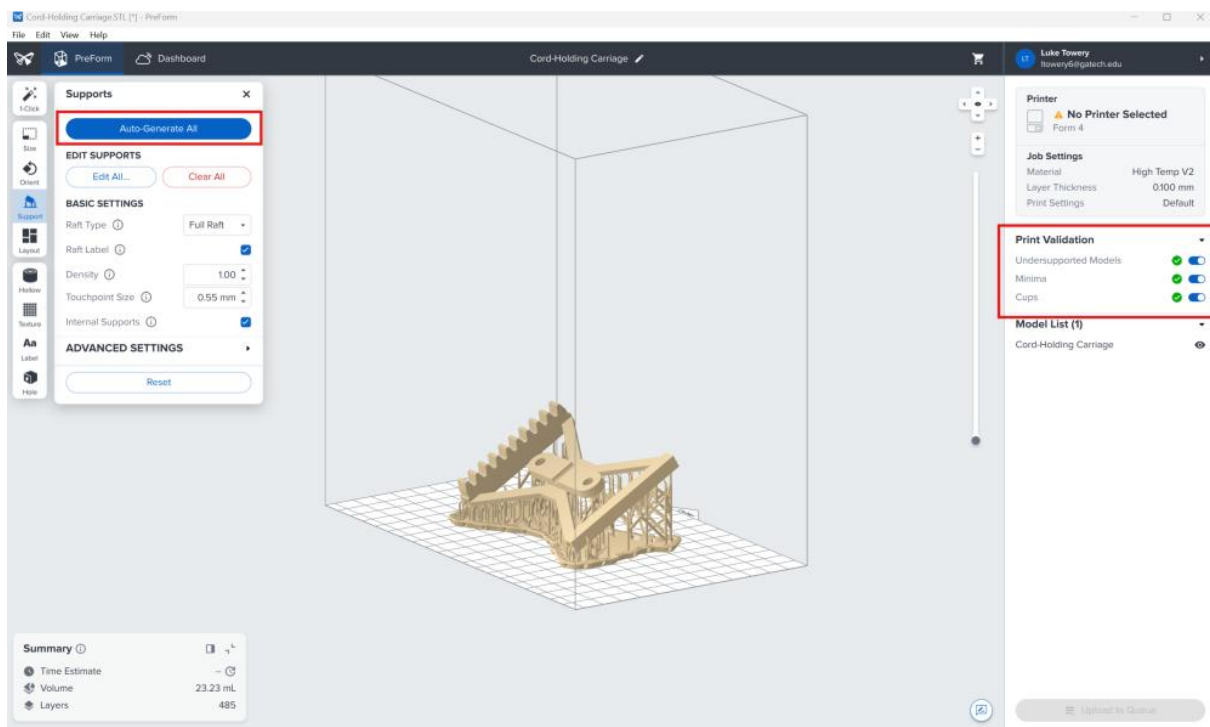
4. Unlike FDM 3D printing, resin prints are completely solid and do not have an infill structure. If you have a larger part that does not need to be completely solid, use the *Hollow* tool to reduce resin use and print time.

5. **Orient your part** appropriately to define where supports will attach and to minimize the chance of a print failure. Click on the part and use the colored arrows to rotate. See best practices [here](#).

6. During the process of orientation, you may see areas highlighted in yellow. These are **cups** where air will be trapped during printing, and the resulting suction can lead to print defects and/or part blowout. The following strategies can help you eliminate these areas:

- Reorient your model so that cavity openings are facing downwards towards the build plate grid. Click [here](#) for more information or reorienting to eliminate cups.
- If you have a hollow part or cannot reorient a cavity, use the *Hole* tool to add a hole (at least 0.75mm in diameter) in the direction of the build plate grid (or parallel to the build plate) at the lowest point of the cavity for air and uncured resin to escape. Click [here](#) for more information on how to use the *Hole* tool.

7. **Add supports** using *Support > Auto-generate All*. Make sure that your final material and layer thickness is selected before generating supports, as they both determine the algorithm used. **Confirm that only green checkmarks are shown under print validation** for *Undersupported Models*, *Minima* and *Cups*. See more on support settings and best practices [here](#).



8. **Save your sliced file** using *File > Save As* and ensure it is being saved with a *.form* extension.

If you need any assistance in the slicing process, our Full PIs are training and happy to assist you.

5. Print Parts

If you are printing with a **Standard Resin** (Fast Model, Black, White, Clear), bring your STL or .form file into the Invention Studio and find on on-duty Full PI with a yellow armband to assist you in approving and starting your print.

If you need to print with an **Engineering Resin** (all others), fill out the [Engineering Resin Request Form](#) with details about your print and your sliced .form file. Pls can assist you in selecting an appropriate material and slicing your file. Within 48 hours, a 3D Printing Master will approve or deny your print, giving feedback if necessary. Once you receive an approval email, come into the Invention Studio during Open Hours and a Full PI with a yellow armband will assist you in starting the print you submitted.

Safety Warning: Uncured resin is a skin irritant, so disposable gloves should be worn when removing parts from build plates or cleaning uncured resin. If your skin is exposed, promptly wash the affected area with soap and water. Resin spills should be cleaned with IPA and shop towels.

Steps 6-10 may be carried out by a trained PI if the printer is needed for another print. If the print is still in the printer when you arrive, a PI can assist with the following washing process:

6. Remove Parts from Build Plate

Wearing gloves, **remove the build plate from the printer** and rotate it handle-down so that liquid resin does not drop onto you or other surfaces. **Remove the parts from the build plate** at the post-processing station, using a clean scraper if necessary.

7. Clean/Return Build Plate

For *Standard Resins*, **return the build plate** to its assigned printer without cleaning it. If any debris or other resins get onto the printing surfaces of the build plate, it will need to be washed completely using IPA and Kimtech wipes.

For *Engineering Resins*, completely **clean all surfaces of the build plate** with IPA. **Return the build plate** to an Engineering Resin printer or the rack.

8. Wash Parts in Form Washes

Using the Washing Chart, **wash your resin in the corresponding Form Wash(es)** for the listed time(s). These IPA washes dissolve most of the uncured liquid resin from your print. Use gloves when handling parts.

Resin Type	First Wash	Second Wash
<i>Fast Model</i>	Dark Primary Wash 5 Minutes	---
<i>White V5</i>	Light/Clear Primary Wash 10 Minutes	---
<i>Black V5</i>	Dark Primary Wash 10 Minutes	---
<i>Clear V5</i>	Light/Clear Primary Wash 10 Minutes	---
<i>Elastic 50A V2</i>	Light/Clear Primary Wash 20 Minutes	Let Air Dry for 10 Minutes and remove supports Secondary Wash 10 Minutes
<i>Flexible 80A</i>	Light/Clear Primary Wash 10 Minutes	Secondary Wash 10 Minutes
<i>Durable</i>	Light/Clear Primary Wash 20 Minutes	---
<i>Tough 1500 V2</i>	Dark Primary Wash 10 Minutes	---
<i>Rigid 4000</i>	Rigid Primary Wash 15 Minutes	---
<i>Rigid 10K</i>	Rigid Primary Wash 10 Minutes	Rigid Secondary Wash 10 Minutes
<i>High Temp</i>	Light/Clear Primary Wash 5 Minutes	---
<i>Flame Retardant</i>	Dark Primary Wash 10 Minutes	---
<i>ESD</i>	Manual Wash Bucket 20 Minutes	---
<i>Castable Wax</i>	Dark Primary Wash 5 Minutes	Manual Wash Bucket 5 Minutes
<i>Silicone 40A</i>	1. Place parts in the Silicone 40A Wash Container with enough fresh IPA to submerge them completely 2. Place the closed container in the sonification bath filled with water 3. Sonicate for 3 minutes 4. Repeat 1-3 with fresh IPA	

9. Hand Wash Parts with IPA

Wearing gloves, remove your parts from the Form Wash and inspect them for any areas that feel sticky. Use an IPA bottle and a rubbing motion to **hand wash** those areas over the sink. Flush any internal channels or features with IPA as well.

10. Dry Parts

After hand washing (if needed), **set parts on the drying rack for 30-60 minutes** to allow the IPA to evaporate completely. If your part is not dry after 60 minutes, it likely still has uncured resin on the surface and needs additional washing. Drying can be accelerated with compressed air.

11. Post-Cure Parts and Remove Supports

The last step of post-processing your part is the **Post-Cure process** that **sets the material properties** of the resin used.

Before curing, decide whether you will **remove supports before or after curing** based on the table below. When removing supports using your hands, flush cutters, tweezers, and/or an Exacto knife, wear eye protection.

Remove Supports Before Curing	Remove Supports After Curing
- Easier to remove - Recommended for aesthetic models - Suggested for supports in areas that are difficult to reach - Recommended for mating features	- More difficult to remove - Recommended by default to prevent warping during curing - Especially important for thin-walled features

Open the Form Cure, place your part(s) on the turntable, and close the door. Select the material of the part(s) and press *Start*. The table below shows curing times and temperatures for different materials.

Resin Type	Setup	Cure Time	Cure Temperature
<i>Fast Model</i>		1 Minute	Room Temperature
<i>White V5</i>		1 Minute	Room Temperature
<i>Black V5</i>		1 Minute	Room Temperature
<i>Clear V5</i>		1 Minute	Room Temperature
<i>Elastic 50A V2</i>	Place dry parts in glass beaker filled with room temperature water	7 Minutes	70°C
<i>Flexible 80A</i>		3 Minutes	60°C
<i>Durable</i>		12 Minutes	60°C
<i>Tough 1500 V2</i>		12 Minutes	70°C
<i>Rigid 4000</i>		6 Minutes	80°C
<i>Rigid 10K</i>		3 Minutes with lights off 7 Minutes with lights on	80°C
<i>High Temp</i>		15 Minutes	80°C
<i>Flame Retardant</i>		5 Minutes with lights off 10 Minutes with lights on	100°C
<i>ESD</i>		7 Minutes	70°C
<i>Castable Wax</i>	Do Not Cure		
<i>Silicone 40A</i>	Place dry parts in glass beaker filled with room temperature water	10 Minutes	60°C

12. Sanding and Polishing

To remove support marks, prep for painting, obtain a smoother finish, and/or improve aesthetics, **resin-printed parts can be sanded and polished**. See guidance from Formlabs [here](#). Specific information for finishing clear parts to be transparent can be found [here](#).

If you have any questions, ask a Full PI or email the 3D Printing Masters at 3dprinting@inventionstudio.gatech.edu.