



# Glowforge

## -Step by Step Guide-

### Step 1: Power On

Power on the machine. Power button is on the back right corner. Light switch on the wall next to our emergency exit should also be on.

### Step 2: Place Material in the Glowforge

Place the material you wish to laser cut/engrave in the machine.

If it's  $\frac{1}{2}$ " or thinner place it on the black mesh support.

If it's thicker than  $\frac{1}{2}$ " take out the black mesh support.

The front of the machine folds down so you can easily remove it.

If the piece is around  $1\frac{3}{4}$ " then there needs to be a small support holding it closer to the laser.

### Step 3: Connect Computer to Glowforge

Go to any computer that is connected to the internet, laptops work too. Ask a staff person to connect you to the wi-fi.

### Step 4: File Information

In order to cut the material, the material has to be  $\frac{1}{2}$  or thinner.

You will need a file format of .svg in order for the glowforge to cut.

---If it's not an svg file it won't cut it will only engrave.  
There are sites online that will convert jpg to svg if needed.

### **Step 5: Upload File**

Click upload and find either the jpg, png or svg file that you want to engrave/cut.  
The glowforge shows what is in the machine so you can rearrange the image so that it fits on the material you are using.

### **Step 6: Choosing Materials**

If you are using a material type that Glowforge sells you can select it from the material list in the top left corner, if not click the "Use Uncertified Material" button and measure from the bottom of the Glowforge to the top of your material and enter the height. The machine is automatically set up to engrave so if you want it to cut click on the picture in the left column and change it to "convert to cut".

### **Step 7: Speed Settings**

Preferred Speed: Sometimes you can use the preferred speed and power that's listed and it cuts/engraves just fine.  
Sometimes though it won't cut all the way through the material.

### **Step 8: Start Cutting/Engraving**

When everything looks good to you. Click on the white button in the top right corner that says **Ready**. It will scan the space and prepare the design.

#### ***Cutting/Engraving Tips:***

*If it's not going deep enough, you'll want to change the settings on the Glowforge. Click on the image on the left column and then you can change the settings. You'll want to slow down the speed, that allows it to burn more. If that doesn't fix it you may need to change the power to full power.*

*If it's not cutting all the way through, don't take the material out of the Glowforge because you'll never be able to line it up properly again, try to slow it down or change the power to full power. If it still looks like it didn't cut all the way through you may have to change it so that it does multiple passes so that it cuts all the way through.*

*If it's engraving too deep, speed up the laser. Lower the power, and that should help the problem.*

**Step 9: Start Glowforge**

Press the glowing white button on the Glowforge.

As it begins to cut, if you notice it's either going too deep or it's not going deep enough you can lift the lid of the Glowforge and it will instantly stop.

Once it's finished give it a minute to cool down and then you can take your material out.