

Work Flow for Embedding a Raster Image into a Vector File before Loading it to the GFUI

As an alternative to adding a raster engraving file onto an object using the Glowforge interface, I have the option of adding the engraving to a separately created vector file, and then loading the vector file to the Glowforge interface. The vector file would then contain both the raster engraving part and the vector cutting part of the whole creation that I want to make.

The advantages to doing this are that I will have more control over the relationship between the engraving and the cutting operations. It's much easier to center a raster design precisely in external design software than it is to use a camera to align it visually. It's also currently easier to rotate files through certain angles, and control the size of an object if precise measurements are required, from within separate design software.

The creation of a vector file has a few more rules that need to be followed, if you are designing for a laser. This is a summary of what I would do or consider when creating one.

*(A good tutorial to start with for definitions is here: [Laser Design Basics](#), and there are other tutorials that explain the steps discussed below listed in the **Table of Contents** for the **Glowforge Tips and Tricks Category** on the Forum.)*

1. Prep the raster file to use for engraving.

(This is usually a Jpeg file, but it can also be a PNG or GIF file.)

- If the file size is large, (over a MB or so), you might want to crop the image and/or save it at a lower resolution. This will help the file to upload faster. If you are planning to do an Auto-trace on the raster though, you might want to keep the resolution a little higher, to get a good trace.
- If you want to feature only part of an image, you might want to clean up and remove any distracting background first. (Adobe Photoshop, GIMP, Corel Paintshop) It reduces file size and makes for a neater trace.
- Save your prepared file as a Jpeg, or PNG file.

2. Prepare your vector file.

(This can be done in any of the 2D Vector Modeling programs like Adobe Illustrator, Inkscape, CorelDRAW or Affinity Designer.)

- There are specific rules that need to be followed when designing vectors for use in the laser that are listed in a bit more depth in the [Laser Design Basics](#) tutorial. But as a quick refresher list I usually:
 1. Make sure that any **text** has been converted into a path. If I want to engrave the text, I leave it as is after conversion. If I want to cut the text, I set the Fill color to none, and give it a Stroke color.
 2. Make sure that any vector shapes that I want to **engrave** have been created using closed vector shapes that consist of a Fill color without a Stroke color associated with it.
 3. Make sure that any vector shapes that I want to **cut** or **score** consist of an Unfilled shape (no Fill color) but that **do** have a color associated with the Stroke.
 4. Make sure that I have used a **different color** for each operation that I want to perform.
(For example: Fill color for engraved parts might be Black, with Cut lines set as Red, and Scores set as Blue.)

5. If I am planning to do both an Engrave, and a Score around it for definition, I will create two copies of the shape, aligned right on top of each other, and use one copy for the Fill (Engrave) and another copy for the Stroke (Score). A single shape that has both a Fill color and a Stroke color associated with it might not be interpreted correctly by the Glowforge interface. Setting it up as two parts ensures that you will get two separate operations created by the Glowforge interface, and you can tell it to engrave one and score the other.
 6. If I have done an Auto-Trace on a raster image, (ex: to create a Cut line around it), I check for hidden paths, duplicate lines, overlapping cuts and follow the various tutorials listed in the **Laser Programs Matrix** and the **2D Vector Programs Matrix**. (Accessible through the **Table of Contents** for the **Glowforge Tips and Tricks Category** on the Forum).
 7. If I have created a Compound Shape, that has open areas enclosed inside of Filled areas, I make sure that the open areas have **null Fill** associated with them. (Instructions are in the tutorials.)
- It might seem like a lot to remember at first, but after a few files, they'll get to be second nature. You can refer to the checklist until you get comfortable with them.

3. From inside your 2D Vector software, import or Place your raster file.

(This can usually be accomplished by dragging the file and dropping it onto the open vector file.)

4. Embed the raster image into the vector file.

(Instructions are in the tutorials for the different programs.)

5. Use the alignment tools in the 2D software to align the raster image with your vector cut lines.

(Instructions are in the tutorials for the different programs.)

6. When the raster image is aligned the way you want, Group the raster image and cut/score lines together.

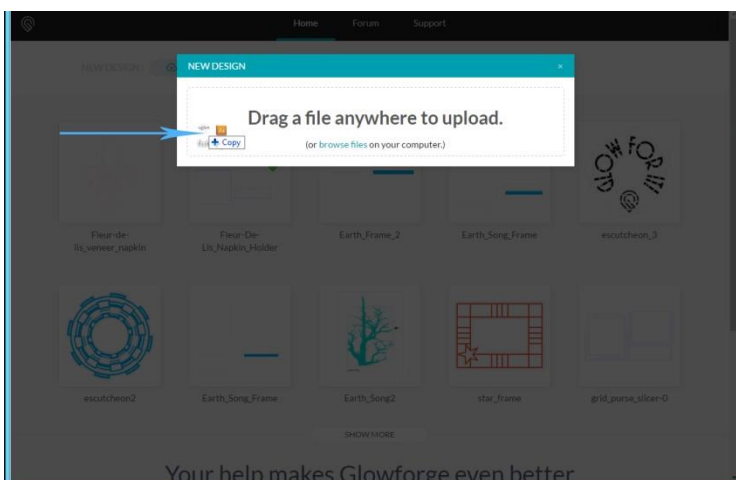
7. Save the file as an SVG format file.

*(If you are saving it from Illustrator, be sure to use **File > Save As > SVG**, and check the **Embed Image** radial dial in the pop-up dialog. Make sure that **Preserve Illustrator Editing Capabilities** is **NOT** checked.)*

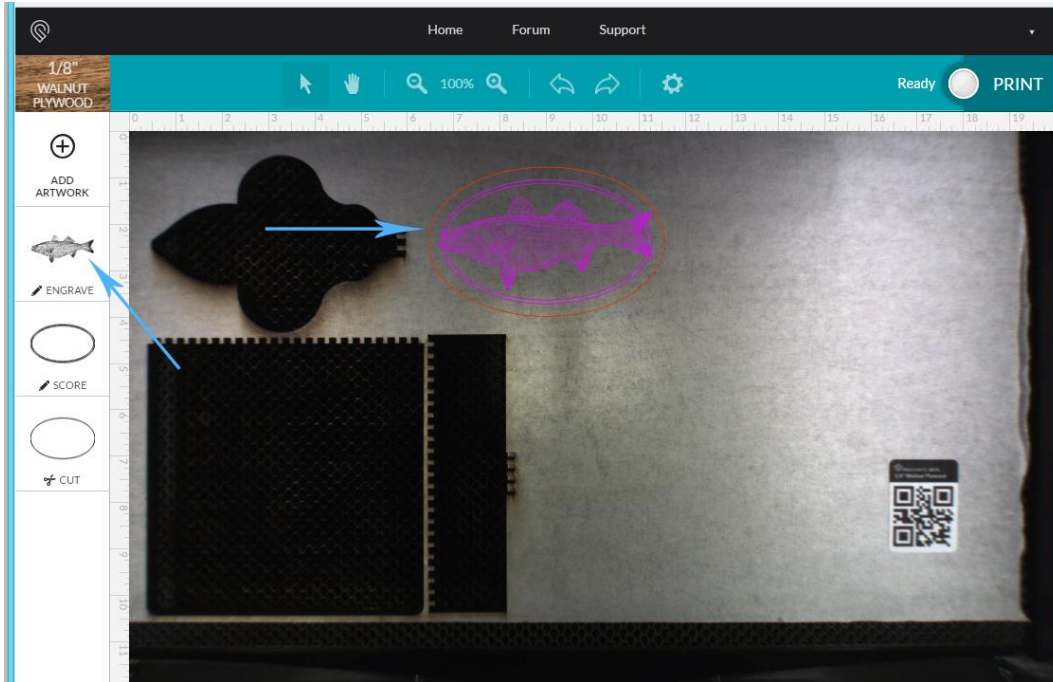
8. Load your material into the Glowforge, and turn it on.

9. Open the Glowforge Interface on your computer.

- Open your SVG file by dragging it over to the Catalog area and dropping it on the bar.

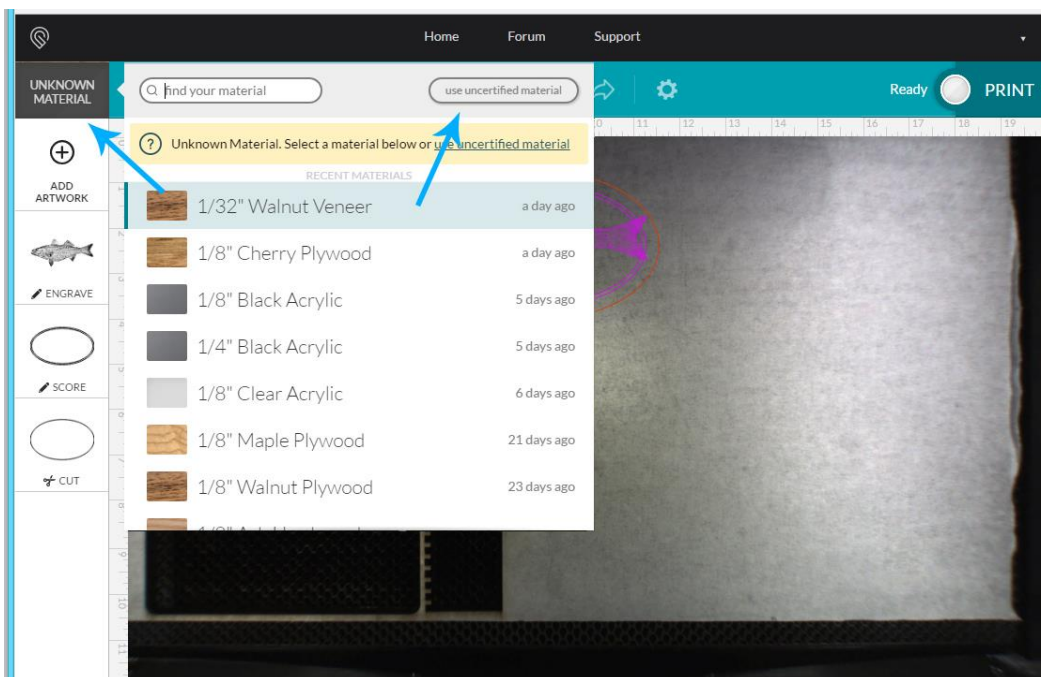


- Select your file and position it on your material.

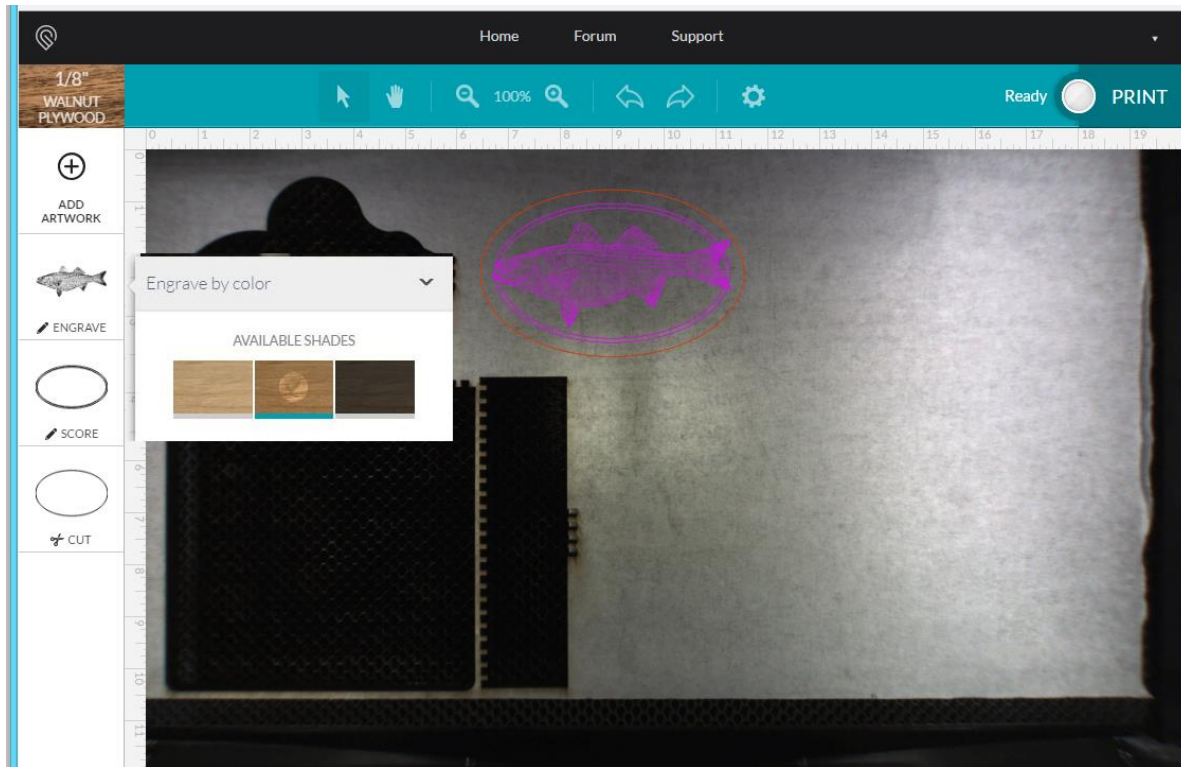


If you want to change the order that the operations are performed, you can click and drag on the thumbnail image in the left column, and release it higher in the stack. The operations are performed from the top of the stack and move down.

- If you are not using Proofgrade material with a QR code, be sure that you have entered the actual measured height of the material, **in inches**, by clicking on the **Unknown Material** button at the top of the left column, and then clicking on **Use Uncertified Material**. (You will need to enter Manual settings for Uncertified Material.)



- Choose your settings for the Engraves, and/or Scores. (Click on the thumbnail in the left column and choose a Light, Medium or Dark Engrave setting for each engraved part. Same for the Scores.)



- Click the **Print** button in the upper right corner. After the file has been uploaded to Glowforge for processing, and the results have downloaded to the machine, the button on the machine will start winking at you.
- Press the big Glowie button on the machine to start the print.
- *(Do not leave the laser unattended while it is in operation.)*
- Wait for the fan to spin down after the print finishes to open the lid. (Cuts down on the smell.)