xTool M1 Laser Cutter
Usage Guide
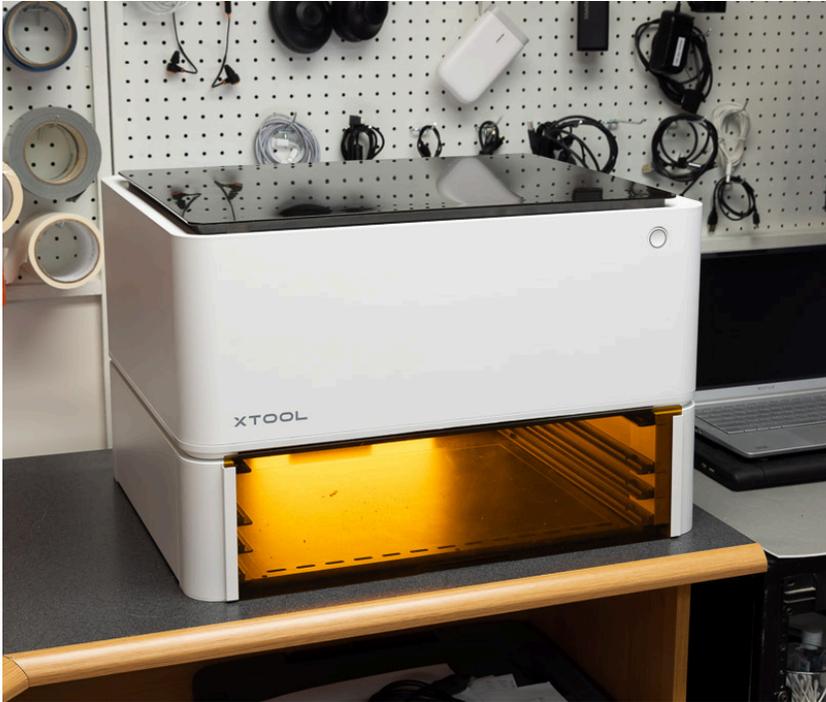
Scan this QR code to access the user guide online

Feb. 26

Shoreline Tool Library
A project of Seattle REconomy

Materials Guidelines

Approved Materials:

For cutting, your material MUST be less than 1/4" thick

- Unfinished, raw wood
- Opaque acrylic
- Fabric
- Paper
- Cardboard
- Coated metal (engraving only)

Restricted Materials:

- Finished/painted wood
- Clear/translucent acrylic
- ALL non-acrylic plastics
- Raw metal
- Thick, synthetic fabric (faux leather, vinyl, ect.)
- Foam
- Anything reflective
- Anything stacked (multiple sheets of paper, ect.)

Safety Information

- You must always stay with the laser while it is running. Do NOT leave it unattended.
- Always use the air filter system.
- Keep all machine doors and panels closed while running
- Never look directly at the laser
- In case of fire, stop job, use fire extinguisher (located in corner near first aid kit)
- Do not touch the laser head or attempt to move it while the machine is on
- Regularly clean debris and dust from the inside of the laser to reduce fire risk

Adding Your Design in the xTool Software

The xTool software is already installed on the computer next to the laser.

You can create your design within this software with the provided tools or you can import an image or vector file.

To import, click the green X logo and go to file -> import.
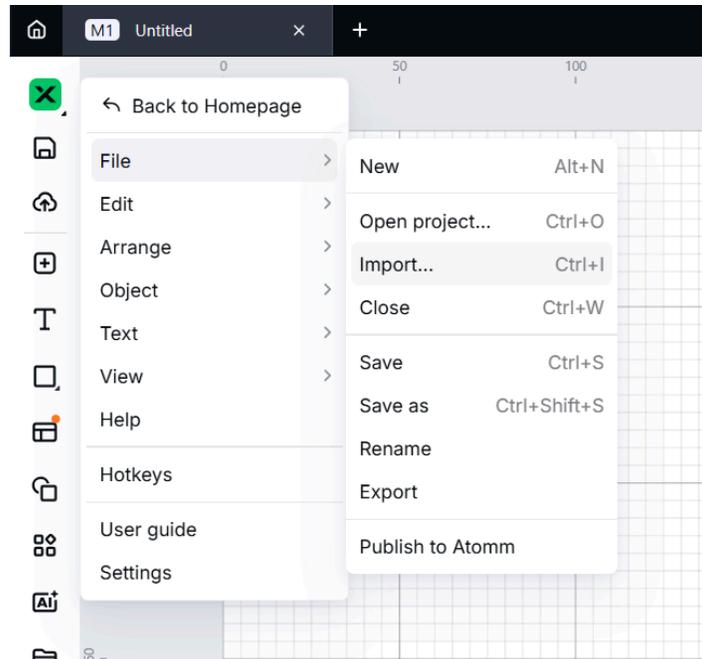
When you upload an image file, it will only be able to engrave. If you want to cut through a material, you must use a vector file. There is a way to convert an image file to an SVG within this software, but it is not extremely accurate. If you want to upload a cutting file, you will need to upload a vector based file like an SVG

Image file (jpg or png)   Vector file (svg)
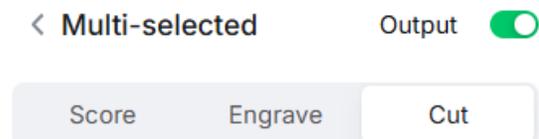
If you add an image file, you can use the 'trace' function to convert the image into an SVG file. This will trace all visible edges of the image, potentially splitting your design into many small parts.

If you only want to cut the entire image out, you can use the 'cutout' feature to remove any backgrounds and then the 'offset' function to add a cut line. (pro tip: if you try to add the offset and it gives you a weird scraggly line, what you can do instead is trace the image, then 'release' the lines and merge them to get a solid shape.)
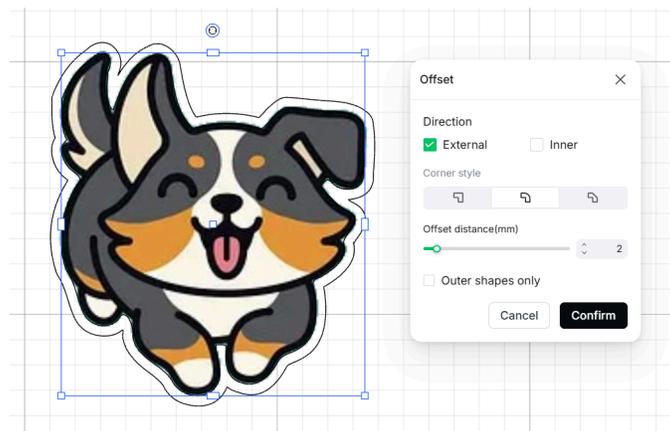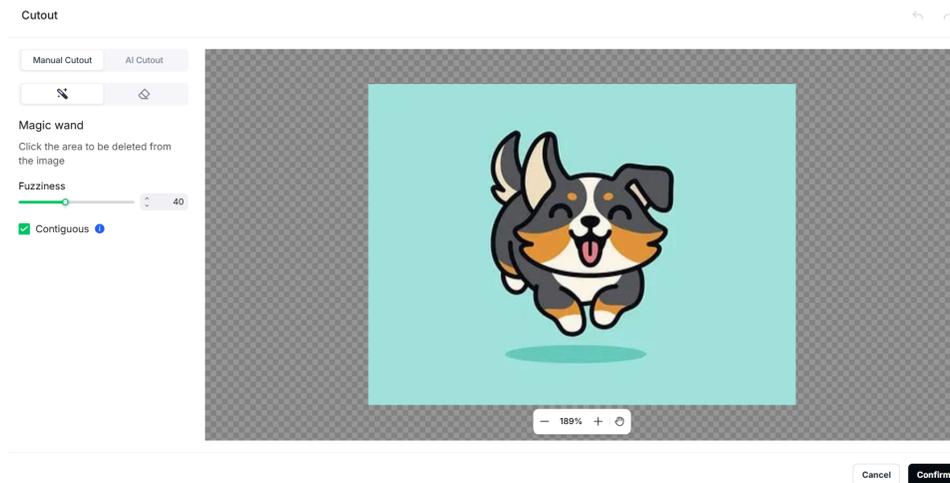
When you import SVG files, the background of the image may appear as purple. If you change the type to 'Cut', it should change to only outlines. Additionally, often this will add a cut line to the border of the image. You can remove this by ungrouping the image and deleting the unwanted cut.
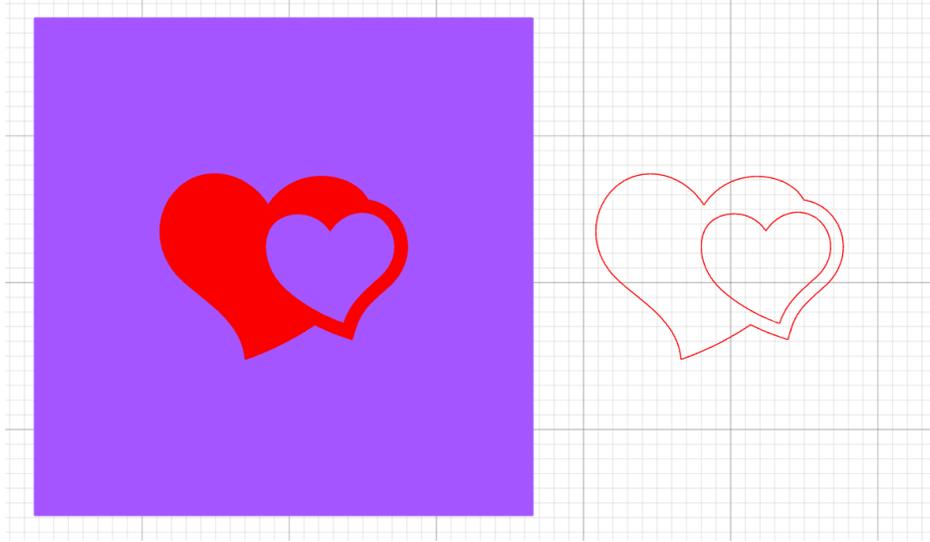
Setting up Software Settings

Material Settings

After your design is ready, you must set the software to match the material you are working with. **Please review the accepted materials before using the laser!!!!**

Once you have confirmed the material you are working with is allowed, you can find it in the material list on the xTool Software. Don't worry too much about the thickness of your material here, there is an auto measuring tool we will use later. When you find the correct material, select 'Apply'.

There are materials on this list that are NOT ALLOWED at this establishment. Although the machine can technically process them, they are not allowed in the space due to safety concerns.
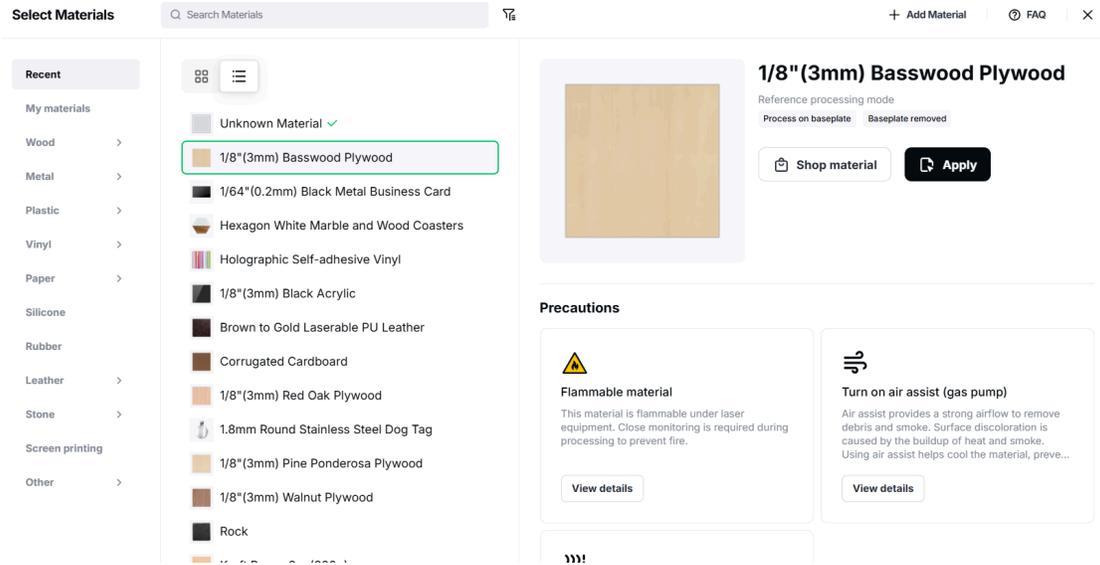
Setting Material Height

Now that our material is set, we must measure it. If you are working with a material less than 1 inch thick, you can set it directly on the honeycomb grid. **Make sure that the red laser is visible on your material.** Then, you can press the ruler icon to auto-measure your material.
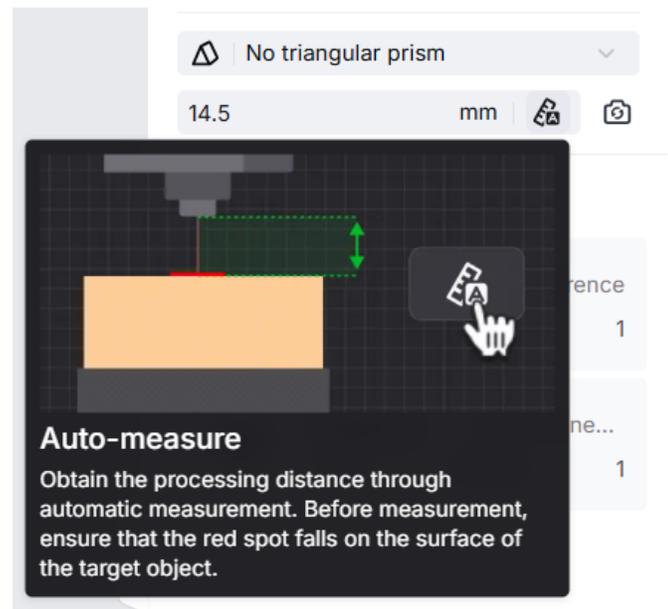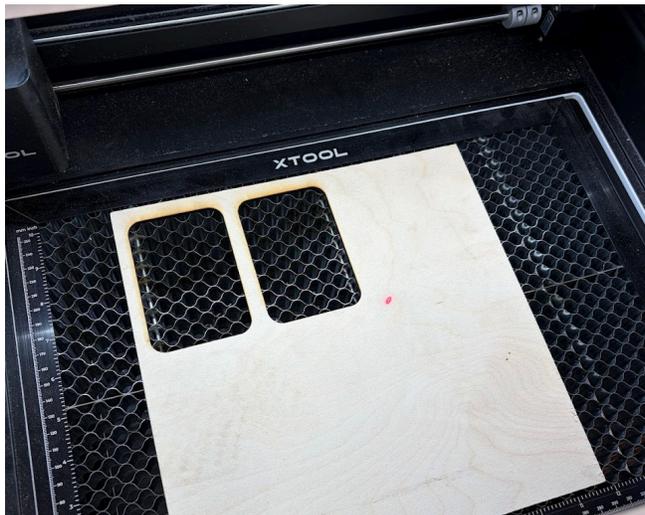
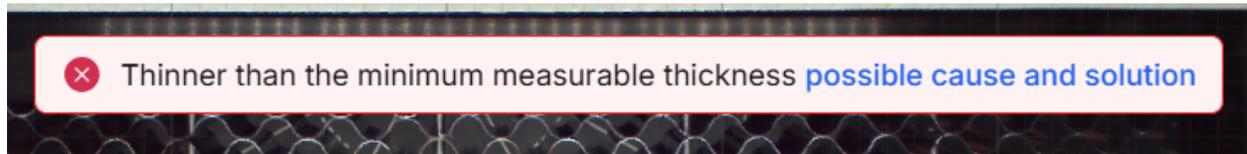If your piece is too thick, not under the red laser, or the bed is placed in the wrong position, you will get this error. Make sure that the cutting bed is in the top-most position (you have to place it in from the top of the machine) and adjust the material so that the red laser is hitting a different spot.

If you are engraving on a material thicker than 1 inch, you will need to move the bed lower. You can take the bed out of the top position by reaching inside the machine and removing it from the top. Then, slide the orange cover up on the front of the machine and slide the bed into one of the 3 slots depending on the height of your object.
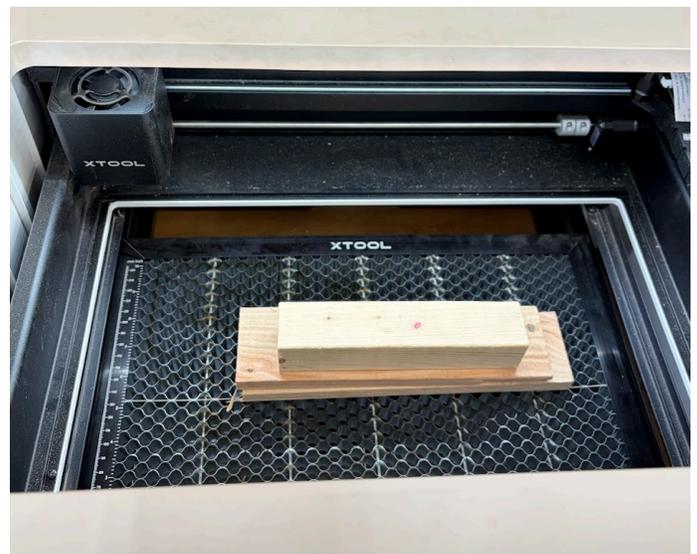
The top of your surface still has to be within that 1 inch range, so you may need to prop it up on some scrap wood. Just make sure the red laser is falling on the surface you want engraved and that it is level.

Placing Design

Once you have the height set, you can use the built in camera to place your design exactly where you want it on the material. You can do this by clicking the camera refresh icon. **Everytime you move the material, you will need to click this refresh button!!!**
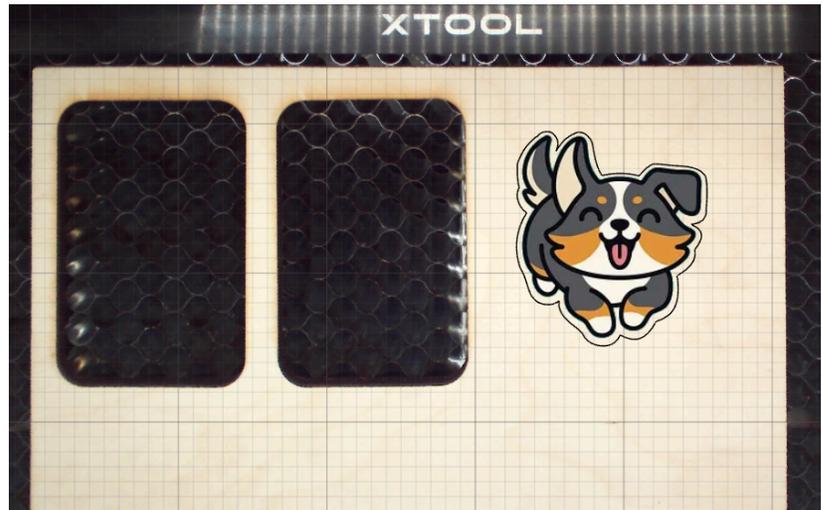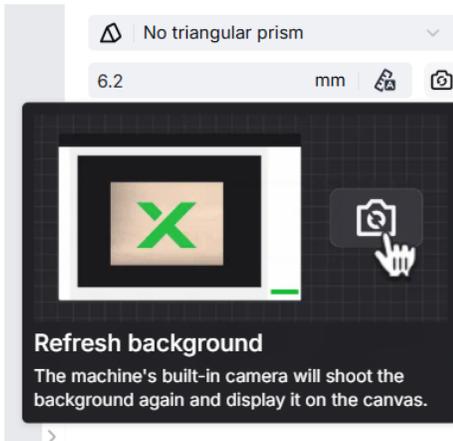
Determining Power and Speed Settings

Both power and speed determine how intense the laser will be. Higher power is more intensity, but so is slower speed. The system automatically sets these values based on the material you selected, but it does not account for the thickness of material you measured.

For many materials, you will need to do multiple tests to ensure you are getting the results you want. Different materials behave very differently, and you may need to mess with these settings to get a clean cut or a sufficiently dark engraving.

The most important vector (cutting) settings are power, speed, and passes. Because these are set according to the thickness of the material selected and not the actual measured thickness, you will likely have to adjust them. For this piece that was about 6mm thick, I turned the speed down to 4 and increased the passes to 2.

In engraving, you have a few more settings to play with. You still have power and speed but you now also have bitmap mode and lines per cm. The bitmap

mode is how the software processes the image into a grayscale array. It does not greatly affect the quality of the engraving unless the image is very detailed or has high lines per cm. For darker engraving, you will need higher power and lower speed. Because all types of materials behave differently, always do a test engraving to check if the output is what you want. (pro tip, I tend to bump up the speed and power settings to make the process faster. 200 speed and 90 power will have a similar result as 100 speed and 45 power, it is just faster.)

Lines per cm is essentially the detail of the engraving. More lines per cm means the image will have clearer, smaller details but it will also take longer.

< Vector	Output	⬤
Score	Engrave	**Cut**

User-defined ∨   ⎘

🔲 One-click set	>

Power (%)   100

Speed (mm/s)   4

Pass   2

Overcut (mm)   ◯

Kerf offset (mm)   ◯

Focus descent   ◯

Tab generation   ◯

< Image	Output ⬤

Engrave

User-defined ∨   ⎘

🔲 Processing Result Image	>

Power (%)   90

Speed (mm/s)   200

Pass   1

Bitmap mode   Jarvis ∨

Lines per cm   100 ∨

Engraving mode   Bi-directional ∨
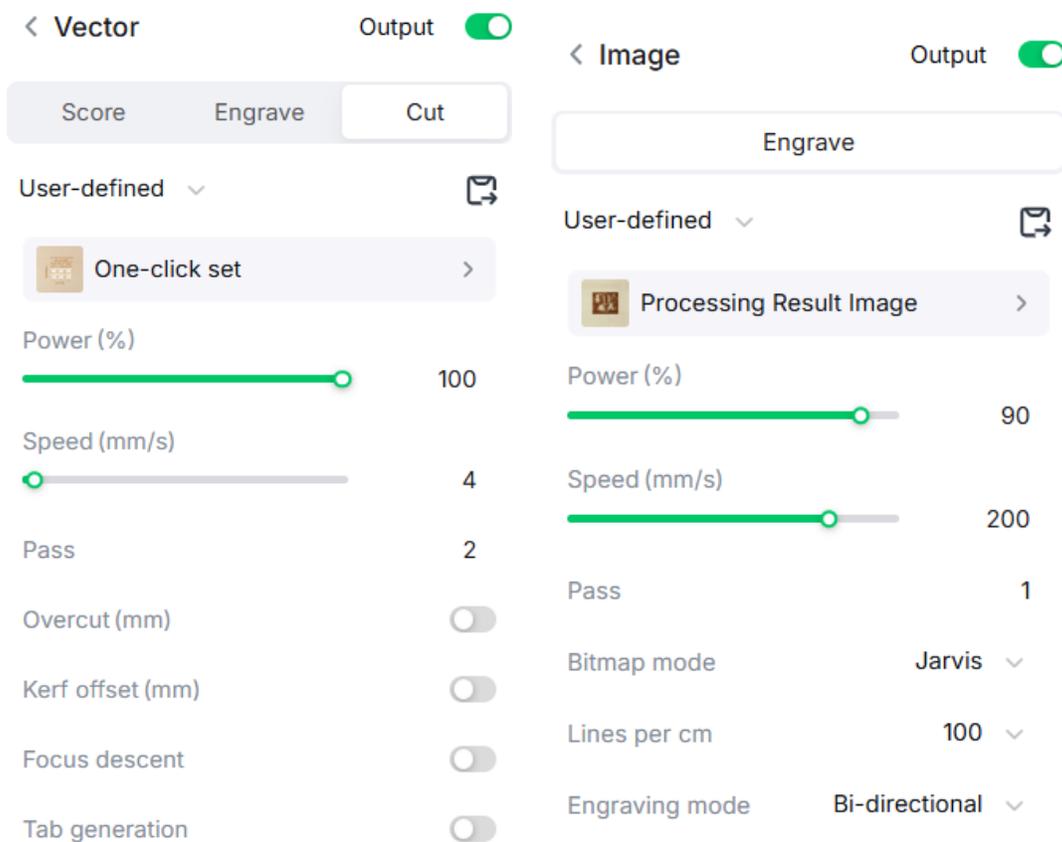
Now you can hit 'Process' and when you confirm everything looks correct, you can hit 'start'. On the processing screen, you can also check the estimated run time. After hitting 'start', you must confirm on the machine by pressing the glowing button on the front. **Double check that the air flow is on before hitting start!!!!**

Now you can sit back and wait for your design to finish! If your design has cutting and engraving, it will engrave first and cut after. Keep a close eye on the job for excess smoke or fire. If there is excess smoking, stop the job and keep the lid closed to clear smoke. If there is fire, stop the job, open the lid, and use the fire extinguisher. Never leave the laser running unattended.

Congratulations on your laser cut!