

## LightBurn Standard Operating Procedures for Laser Cutting

Author: Dr. Florian

Last Updated: 2/28/2024

**Background:** LightBurn is a graphics cutting software that allows users to import artwork in a variety of common vector graphic formats, including SVG and DXF. It includes features for direct control of the laser cutter, including real-time adjustment of power and speed.

**Users:** Only students who are trained in CHBE 4200 and have completed Vanderbilt's Laser Safety training may operate the laser cutter. To turn on and prepare the laser cutter, please see the [Boss Laser LS1420 operating procedures](#).

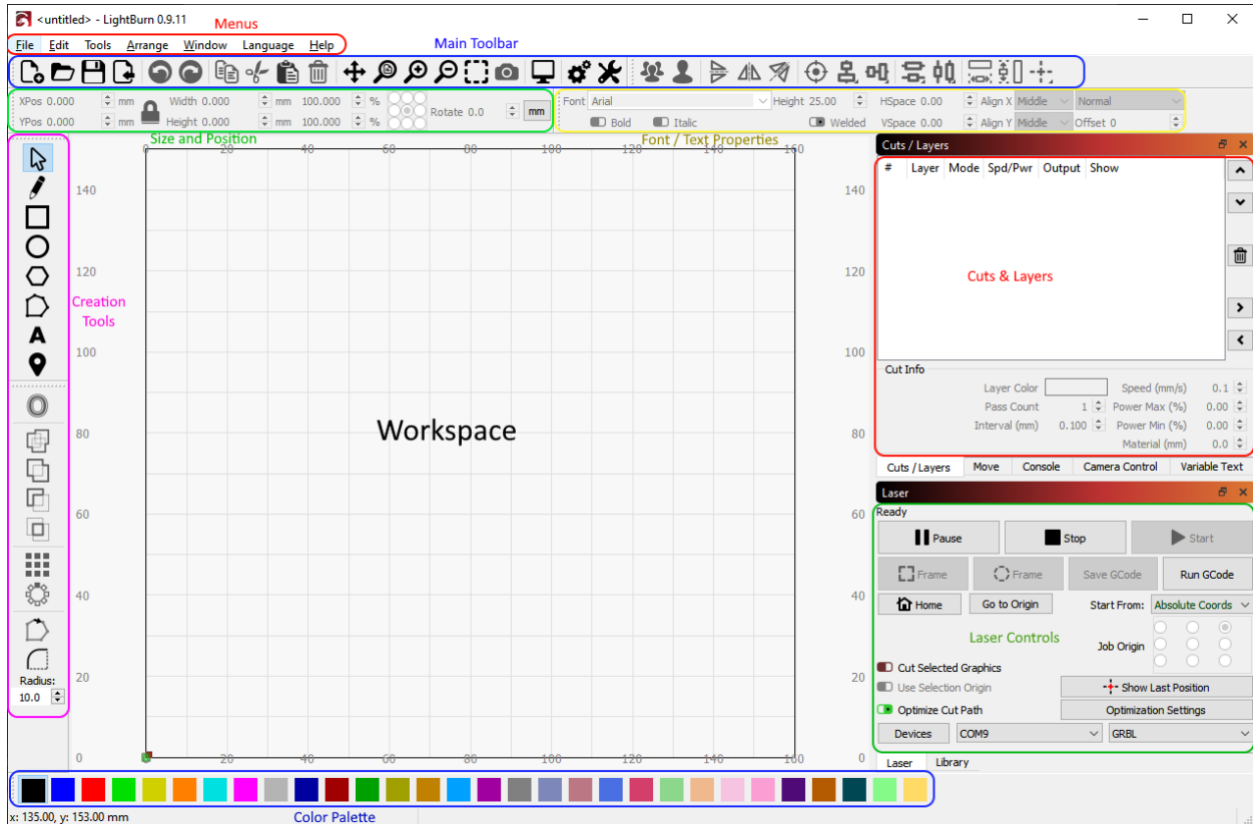
For the most up-to-date information please see [LightBurn documentation](#) and for instructional videos see the [LightBurn YouTube channel](#).

### Table of Contents

1. The Main Window	1
2. Connecting to the Laser Cutter	2 – 3
3. Drawing Basic Shapes	3 – 4
4. Cutting/Engraving Paths	4 – 6
5. Importing Graphics	
5.1. SVGs from Online	6 – 8
5.2. DXFs from Fusion 360	8 – 9
6. Kerf Compensation	9

## 1. The Main Window

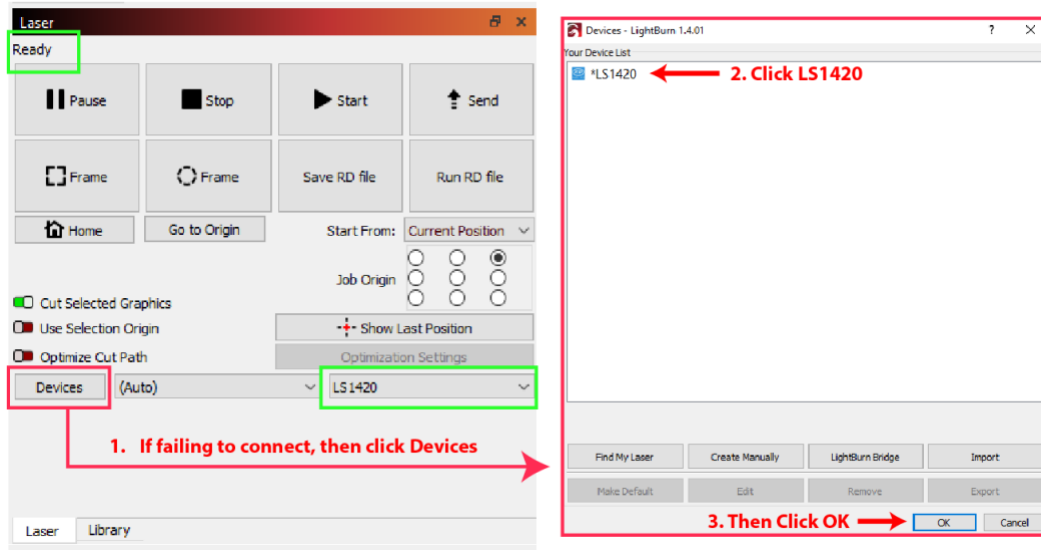
Familiarize yourself with the main window (**Fig. 1**). If panels are missing, navigate to the toolbar, click *window*, and then *Reset to Default Layout*.



**Figure 1.** The main window in LightBurn.

## 2. Connecting to the Laser Cutter

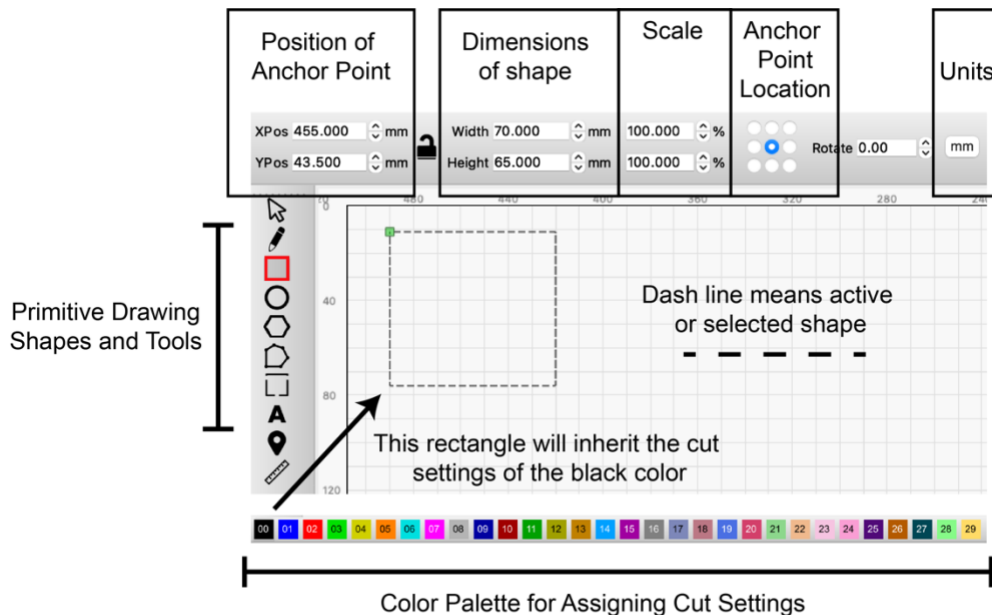
1. In the Laser Controls panel, the model should be *LS1420* and the status should be *ready* (**Fig. 2**). If the status shows disconnected then proceed to step 2.
2. If LightBurn is unable to connect to the laser cutter, then close all other programs open on the computer by using task view (Windows key + tab). Of particular concern are programs that communicate with hardware over USB that may be blocking LightBurn from communicating with the laser (e.g., Bantam Tools and Pronterface).
3. Once other programs are closed click on *Devices* in the Laser Controls panel, then click *LS1420*, and then *OK* (**Fig. 2**). The laser status should now be “ready.” If not, ask for help.



**Figure 2.** Laser control panel showing a successful connection to the LS1420 laser cutter

### 3. Drawing Basic Shapes

1. The Creation Tools panel on the left allows for several basic shapes to be drawn (oval, rectangle, polygon) (**Fig. 3**). Alternatively, the pencil can be used to draw freeform geometries or the text tool can insert letters. Depending on the tool selected you may have to click and drag or click once to see the vector art populate the workspace.



**Figure 3.** The Creation Tools, Location and Position, and Color Palette panels allows for vector shapes to be drawn and modified.

- Once your design is drawn, the position and dimensions of each closed path can be adjusted with the Size and Position panel. However, the geometry must be selected, which is denoted by moving dotted lines, before altering dimensions.
- Each closed path can be assigned different cutting parameters by choosing a different color from the Color Palette Panel. This allows each features to be cut/engraved to different depths all during the same cutting operation.

#### 4. Cutting/Engraving Paths

- To cut out a square and engrave a circle in the same workspace, each shape will require a different color from the Color Palette panel to assign different cut settings. For this example, the square will be black and the circle blue (Fig. 4).
- For each color, which are referred to as layers in the Cuts and Layers Panel, you must select a cut mode of which there are three:
  - Line** – Laser follows the perimeter of a shape. Line is used to engrave thin features or to cut a 2D shape out of a stock material.
  - Fill** – Rasters (i.e., scans) the laser inside the closed interior. The scan direction is typically left to right and bottom to top.
  - Offset Fill** – Similar to fill but the scan lines follow the perimeter of the shape and spiral inwards. Not commonly used, but you can read more [here](#).
- To cut out the rectangle, a line cut is assigned to the black layer. Never cut a hole/geometry by completing vaporizing the interior area through fill. This is slow and degrades the lifespan of the laser. Fill should only be used for engraving (i.e., not cutting completely through the model). A fill cut is assigned to the blue layer of the circle.

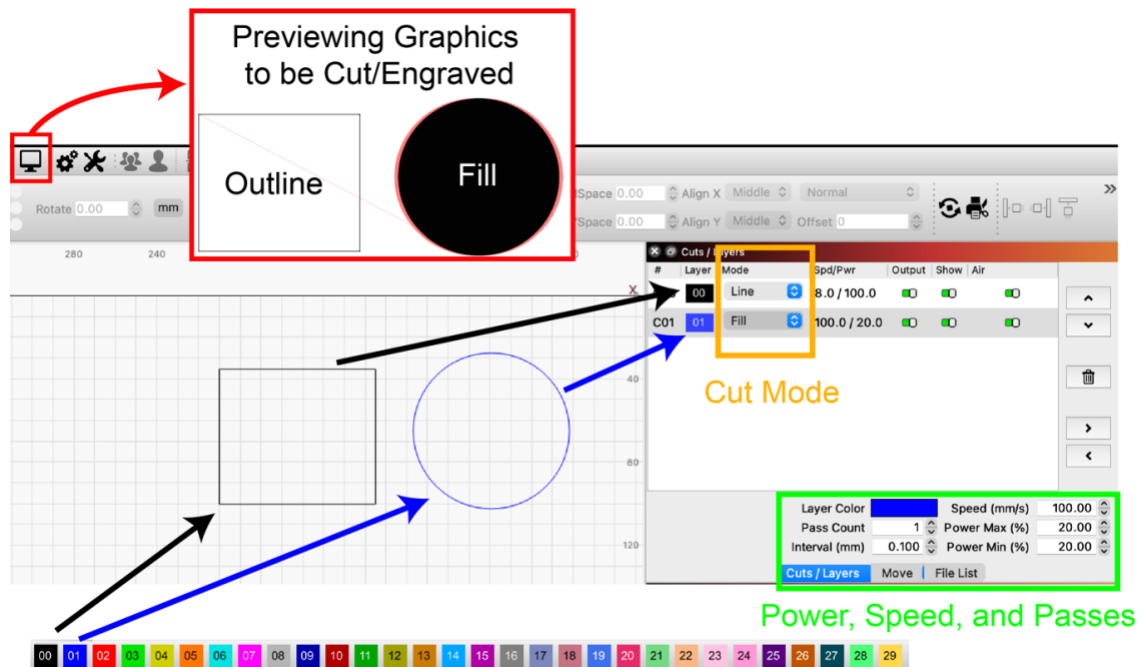


Figure 4. Configuring two separate cutting operations for a square cutout and an engraved circle.

4. For each layer/color, the laser must be configured with the following settings:
  - **Speed** – Speed is how fast the laser is moved. For a constant power level, higher speeds will decrease the depth of cut. Severing a material will require slow speeds while shallow engraving can take advantage of fast movements. The max speed of the LS1420 is 350 mm/s.
  - **Power Max** – This is the maximum percentage of power that the laser will produce during the cut. This is the power level the laser will be for a majority of the cut.
  - **Power Min** – This is the power setting that will be used during a cut when the machine slows down to reverse direction or make a sharp corner. If the min power is too high, then the laser will over burn the corners, but if too low, then the laser will fail to fire. When cutting slower than 50 mm/s, Power Min = Power Max. For high-speed engraving, power min must be calibrated.
  - **Pass Count** - The number of times the laser will trace the same path. Pass count is only increased after the laser fails to cut to the correct depth when the power max is set to 100% and the speed is decreased to a low enough speed that it is causing artifacts.
  - **Interval** - Line interval is the spacing between scan lines calculated during a fill cut. The lower this value, the less space will be left between successive passes. Rarely changed from 0.1 mm.
5. To determine the laser setting for each type of cut, start with a lookup table, such as [this one from Boss Laser](#), and match your material to the settings. The LS1420 is equipped with a 70W laser. As CO2 lasers age their power output drops. Therefore, you must perform a test cut to determine if a higher power and/or slower speed is necessary to reach the required depth of cut. Increase power max before lowering speed.
6. **ALWAYS PREVIEW BEFORE CUTTING.** Click on the monitor icon in the Menu Toolbar (**Fig. 4**). A preview of the operation will appear showing cutting moves (black) and non-cutting travels (red). Carefully inspect that the preview matches your design. Open profiles/paths and details too small to be resolved will not appear as expected. You must fix the design.
7. The coordinate system settings in the Laser Control Panel tells the CNC machine where the square and circle are located (**Fig. 5**). There are three possible coordinate system selected from the *Start From* menu:
  - **Absolute** - The workspace in the main editing window represents the laser cutter's real-life platform. The material-to-be-cut must be placed on the laser cutter's bed at the same location of the shapes.
  - **Current Position** - Your job cuts relative to the current position of the laser head. Current Position and Job Origin must be used concurrently to tell the cutter, which direction the design is located. When operating in Current Position mode, a green square will appear on the design marking one of the 9 possible job origins. In our example, the laser must be positioned, in the top-left of the design. Current Position provides the most flexibility when cutting remnants (i.e., leftovers).

- **User Origin** – A separate coordinate system set through the laser cutter control panel. This feature is not commonly used.
- LightBurn can be told to cut all features in the workspace or just the selected graphic through the Cut Selected Graphics toggle. It is recommended to leave Cut Selected Graphics ON to prevent the unwanted cutting of geometry that may have been left over from a previous session. If the green origin square is not near your design(s), it is likely there are other paths presents.
  - After the cutting, laser, and coordinate system settings have been configured, you can press Start. The laser cutter will start immediately. Be sure all doors are closed and that the supporting equipment are turned on (compressor, chiller, blower).

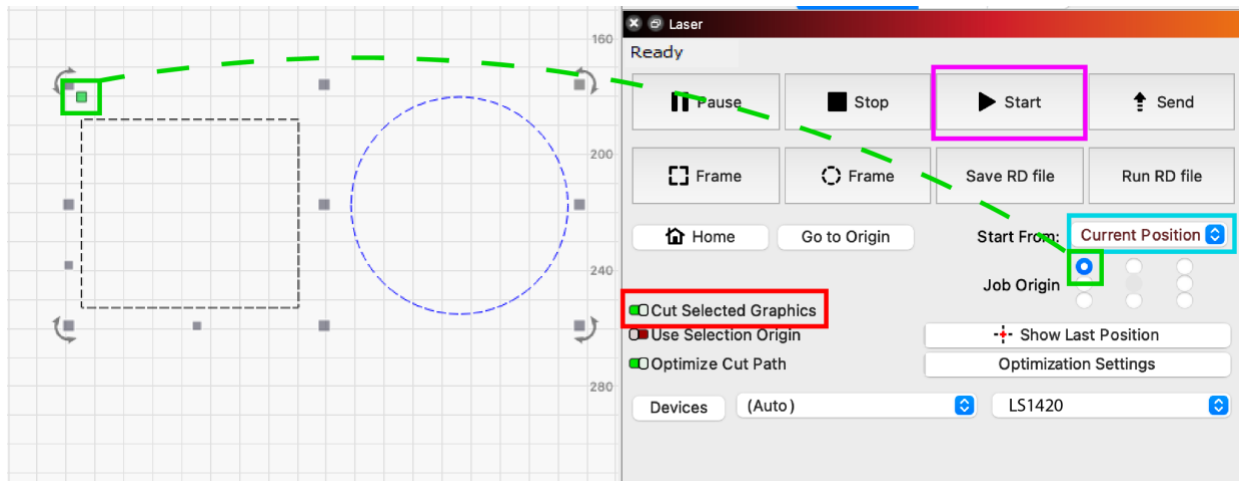


Figure 5. Coordinate system controls for setting the starting position of the laser.

## 5. Importing Graphics for Cutting/Engraving

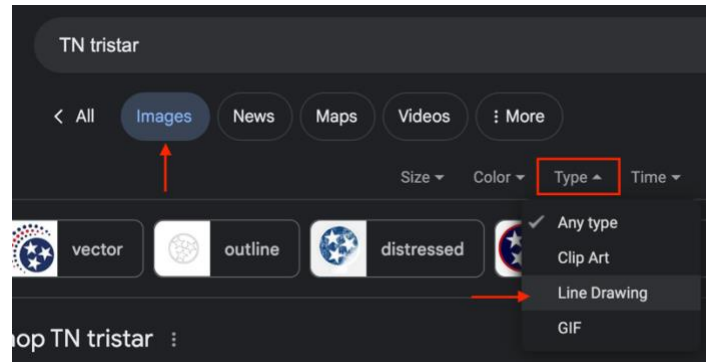
By importing graphics into LightBurn from CAD (Fusion 360, Solidworks, etc.) or vector graphic (Adobe Illustrator, Canva, etc.) software users can cut and engrave more intricate designs.

*Please note:* It is possible to engrave photos (i.e., raster image formats, such as PNGs and JPEGs), but these file types will not be covered in this SOP. Please see [this YouTube video](#) and review the [Image Mode settings](#) on LightBurn's documentation for laser engraving these types of images.

### 5.1 Importing .SVGs from Online

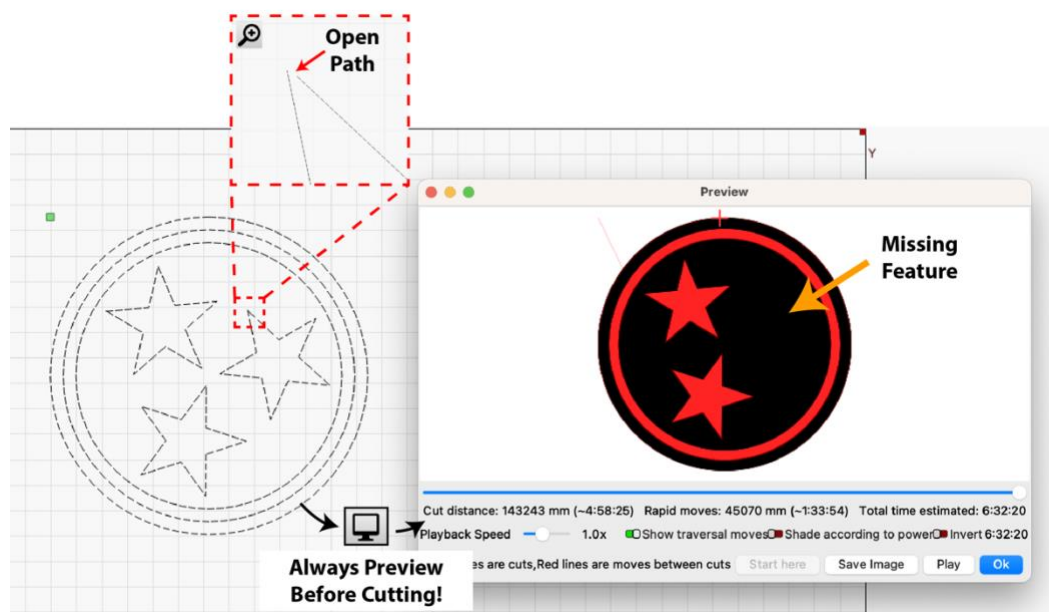
Scalable Vector Graphics (SVG) files are a common format for logos and similar illustrations due to their vector-based nature, which allows for infinite scaling without any loss of quality. Unlike raster images, which can become pixelated when enlarged, SVGs maintain crisp lines and clear details at any scale. Further, the vectors or paths that make up the graphic can be cut/engraved with the same procedure and tools as described in section 4.

1. Find a .SVG file online or create one with your preferred software package. You can search for exclusively “Line Drawings” on Google, which filters for SVGs (**Fig. 6**). Please exercise caution when searching for logos to ensure you do not inadvertently infringe upon a company's copyright protections. **Do not download any file types besides .SVG on the laser cutter's computer as these could be malicious.**



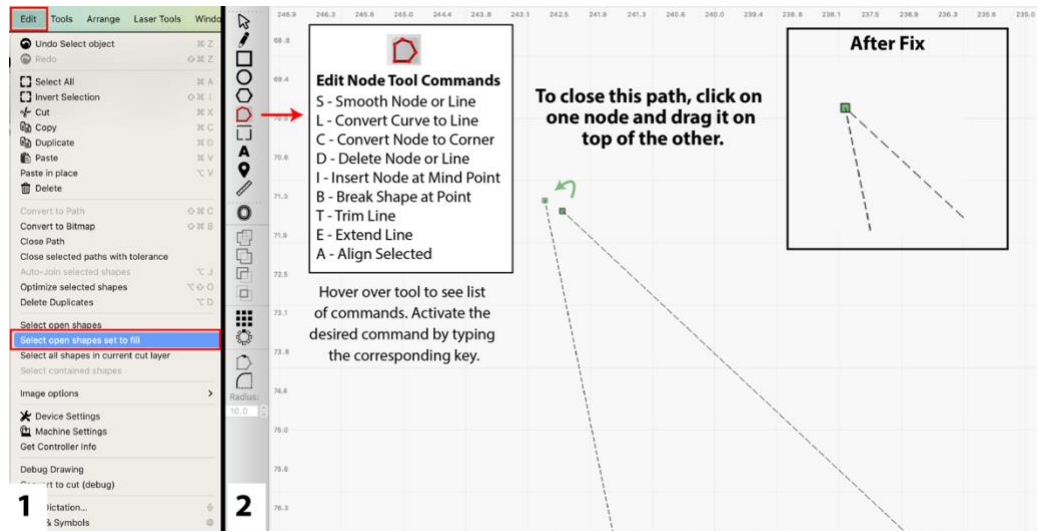
**Figure 6.** How to search exclusively for SVGs with Google Images.

2. Import your graphic into LightBurn by dragging and dropping the file into the workspace or through “file” then “import”. Resize the graphic to the desired size through the main toolbar or by clicking and dragging on one of the corners.
3. Select either the “line” or “fill” cut mode for each feature of the design. Please note: **fill only works on closed paths** (i.e., the start of the path is connected to the end of the path). Designs downloaded from the internet, especially complex ones, may have paths that appear to be closed but in reality, are not. Always preview designs before cutting/engraving (**Fig. 7**).



**Figure 7.** Importing a SVG file and previewing a fill engraving operation. A feature is missing from the preview due to an open path.

- If you receive the warning “shape(s) were set to fill but weren’t closed” or if the preview is different from the design on the workspace, then it is likely that there is at least one open path present. These paths can be identified by navigating to “edit” then “select open shapes set to fill.” The Edit Node tool can be used to close any open paths (**Fig. 8**). For designs with many open paths, it may be best to find a new design online then to repair the broken one as this can be tedious.



**Figure 8.** Identifying and closing open paths with the Edit Node tool.

- After the preview matches the design, set the appropriate coordinate system then start cutting.

## 5.2 Importing DXF Files from Fusion 360

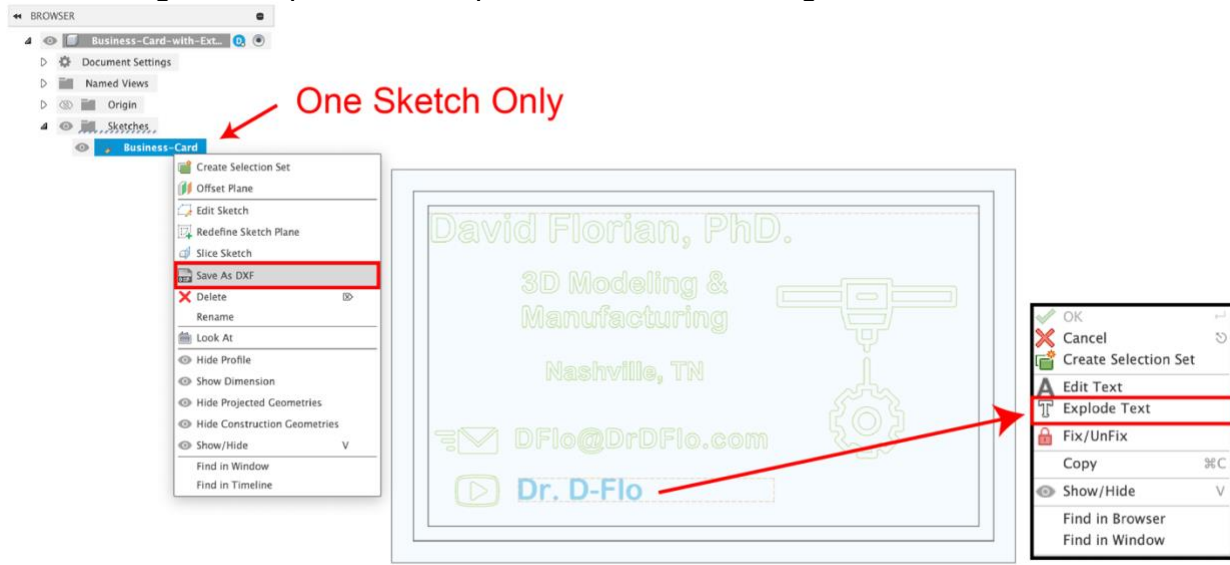
While SVG files are designed for flexible scaling without losing quality, DXF (Drawing Exchange Format) files retain their original dimensions upon import into LightBurn. DXF files are especially useful for translating designs from CAD software packages like Fusion 360 into laser cutting projects, ensuring laser cut parts fit together with each other and additional components of an assembly.

Laser cutting is fundamentally a 2.5D process, capable of cutting materials to varying depths but unable to fabricate fully 3D objects. Therefore, in Fusion 360, it's the 2D sketches that are critical for laser cutting, as these can be exported as DXF files, which accurately represent the design's dimensions and can be cut, unlike 3D models or bodies.

- Draw the design you wish to laser cut or project all features into **1 sketch**. Only a single sketch can be exported as a DXF file at a time. If text is present, right click on it and select “explode text.” This process converts the text to paths, but it is not possible to edit the text after this. See **Figure 9**.
- Exit the sketch. In the browser menu under sketches, right click on the sketch you want to laser cut, and select “Export as DXF.”



3. Drag and drop or file -> import the DXF file into LightBurn.

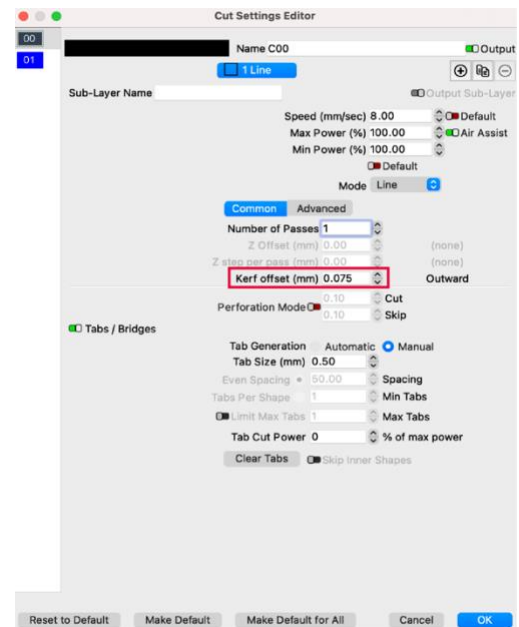


**Figure 9.** Exporting a single sketch as a DXF file in Fusion 360. All text must be “exploded” before export.

## 6. Kerf Compensation

The purpose of the kerf offset is to account for the width of the cut resulting from the laser’s operation. Without kerf compensation LightBurn will fire the laser directly on the path in a given design, leaving undersized exterior features and oversized holes and cutouts. The actual size of the kerf (i.e., vaporized gap of material) depends on several factors: laser power and speed, focal distance of lens, and material type and thickness. Consequently, the kerf must be measured for each material if accurate dimensions are required. To compensate for the kerf, follow the below steps:

1. Zero the current kerf compensation by double-clicking on the layer/color in Cuts and Layers panel. This will open the Cut Settings Editor. Set Kerf offset to 0.
2. Draw and cut out a square of predefined width and height (say 25 mm x 25 mm).
3. Measure the square with a pair of calipers. It should be undersized. The kerf offset compensation =  $(\text{theoretical} - \text{actual})/2$
4. Re-cut and check.



**Figure 10.** Cut Settings Editor dialogue.