# Inkscape and Laser Cutting

A guide to get you started with Inkscape and laser cutting!

1. Launching Inkscape

Start by opening the program and creating a new document (File > New). First set the dimensions (File > Document Properties). It is essential to make the dimensions match the dimensions of the physical material that you'll be cutting/engraving.

Next set your display measurement units to inches or millimeters. You want your artwork to be sized exactly as you want it on the material.
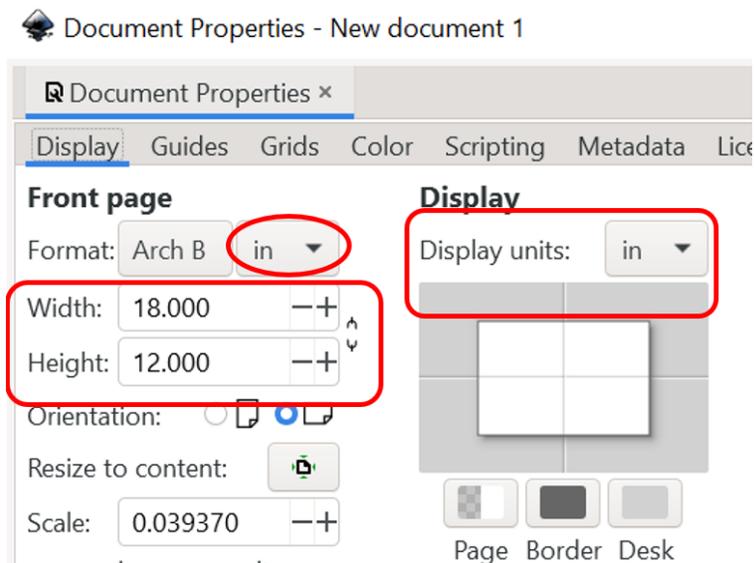

2. Using Inkscape Tools

The most common tools that you'll likely use in Inkscape are the Shape tools, Line tool, and Text tool. These tools can be easily accessed via the sidebar. It's recommended to spend some time with these tools to prepare yourself for future creative projects.
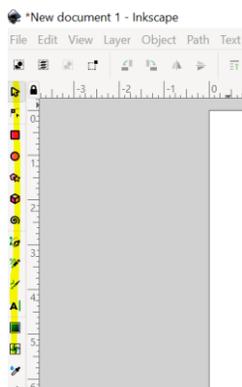
3. Setting correct stroke width, color, and fill

When preparing files for laser cutting, there are some machine-specific and industry standard constraints. For any vector lines or shapes for cutting, it is recommended the stroke width must be set to .01 mm. This will allow the cutter to interpret these lines as paths for the laser to follow. For any raster engravings, the shape must have a fill. This fill must be set to greyscale, with full black being the deepest engraving (full white being no engraving at all). Click the Object menu > Fill and Stroke to adjust these setting in the Stroke Style tab.
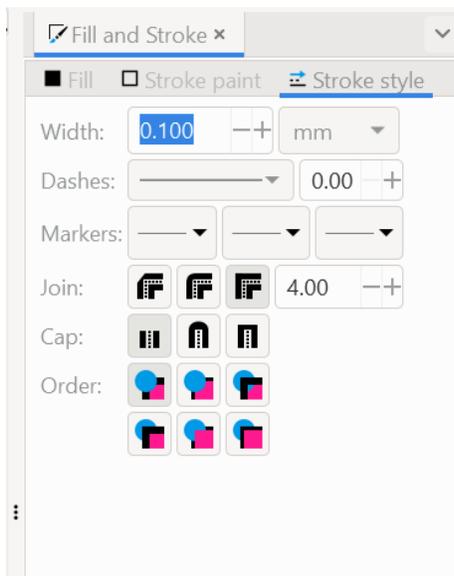

For legibility, it is recommended that you follow industry standard color guidelines for your laser cutting file. The guidelines are as follows:

- Red – Cutting lines
- Blue – Engraving lines
- Black – Rastering shapes/images

4. Rastering Images

If you have any pixel-based images that you'd like to add to your file, just make sure those images are converted to greyscale and have a high contrast! You can adjust contrast in photo editing software such as Gimp or Photoshop.
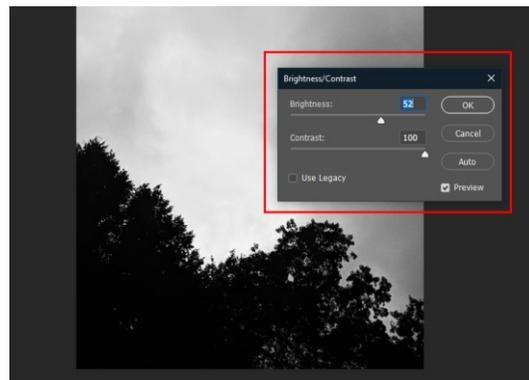
5. Common issues and Solutions

Occasionally you may run into an issue where the laser is going over your cutting/engraving lines multiple times. This can result in a less-than-optimal edge, or even possible melting. This issue can be easily resolved by making sure you do not have lines overlapping each other. Always double check your stroke width to make sure the laser is recognizing the .010 mm stroke.

Resizing artwork at the laser cutter is clunky and inaccurate. You will want to make sure your artwork is sized in inches so that is the size you want cut/engraved. Double check the size of your artwork before heading to the laser cutter.

6. Outline View

We also suggest viewing your design in "outline view" mode multiple times during the process of designing your parts for laser-cutting (View menu > Display Mode > Outline). This will make it easier to catch empty objects or stray points that could potentially create problems down the road.
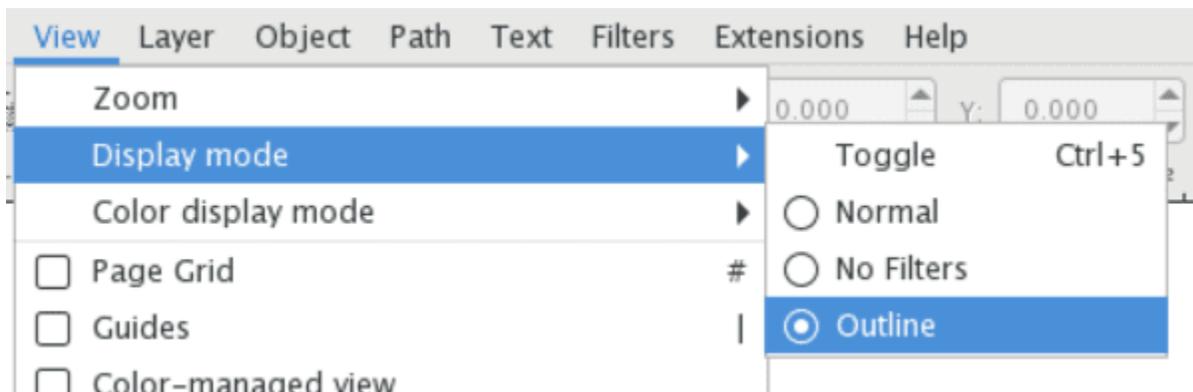

Watch for open contours in the outline mode, especially if the line is intended to be cut. An easy way to find these openings is to view your design in Outline display mode.
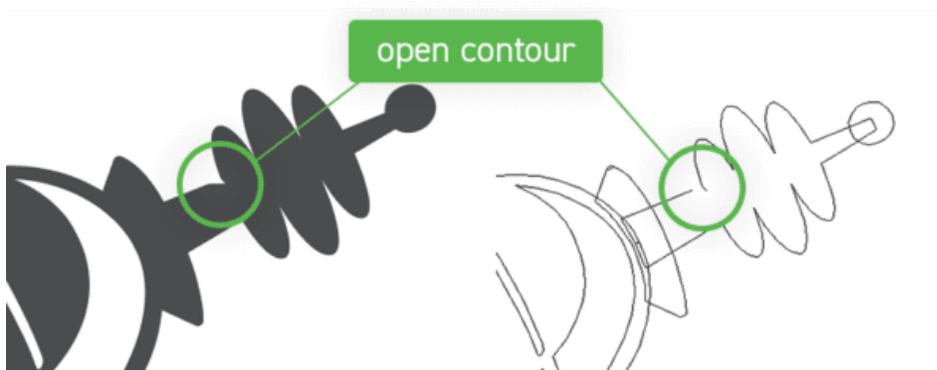
If you do find open contours, you should join or close the nodes. To do this, use the "edit paths by nodes" tool (F2 or N.) The nodes are the gray boxes you see around the outline of your object. Select the two nodes you want to join. They will turn blue when they are selected.
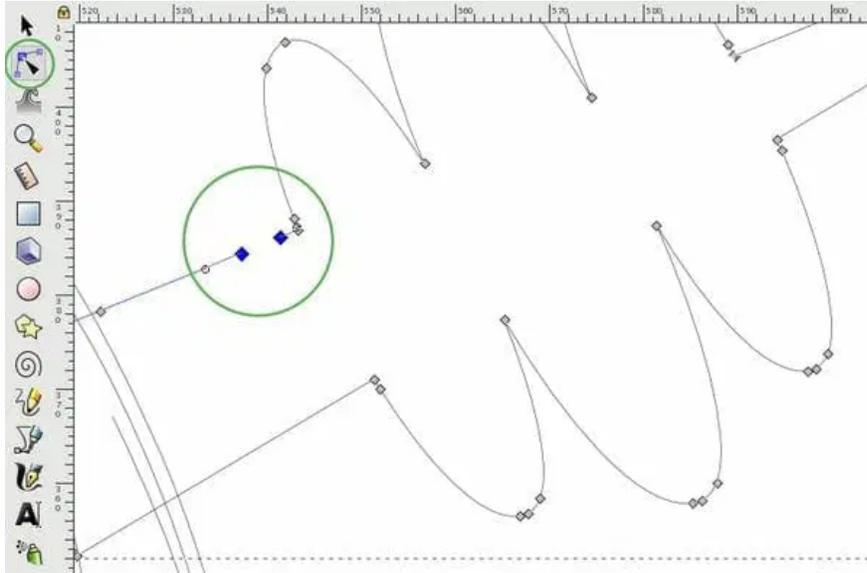
Click the "join selected endpoints with a new segment" button that appears in the top left corner when you have two endpoints selected. This will connect your nodes and close your contour.
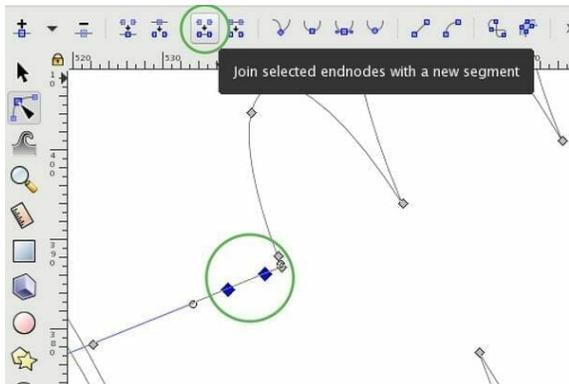
In addition to disconnected lines, you need to make sure your design does not have any intersecting lines. Intersecting lines or lines that cross each will yield unwanted results in the laser cutter, creating more cuts in the material than you want. You can find these once again by viewing your design in Outline display mode, and you can remove them with the "break path at selected nodes" button.

Finally, you should be on the lookout for multiple shapes overlapping each other. When you find these, use the Boolean Operations in the Path menu to combine or exclude your objects. You will use the Union, Division, and Exclusion tools often while designing for laser cutting in Inkscape.

7. Clean Up

Before you export your file as a PDF or SVG file format, be sure to remove everything that is not going to be laser-cut. This includes empty shapes, boxes, stray points, and text areas

Make sure all your objects are on the same layer. Additionally, you should not leave any dimensions or design notes in the file. What you upload to us from Inkscape should be a clean .SVG or .PDF with only your laser cut design in it, with no markings or unnecessary non-design additions.

Finally, make sure all your artwork fits within the artboard. While the artboard doesn't act as a border or a cut line on your final design, the artboard border will cut off anything left outside of it during exporting. To do this, go to the Document Properties under File (or you can use the hotkeys Shift+CTRL+D). Under Custom Size, select the dropdown next to Resize Page to Content and click Resize Page to Drawing or Selection. This will automatically resize the artboard to include all your artwork with room to spare.
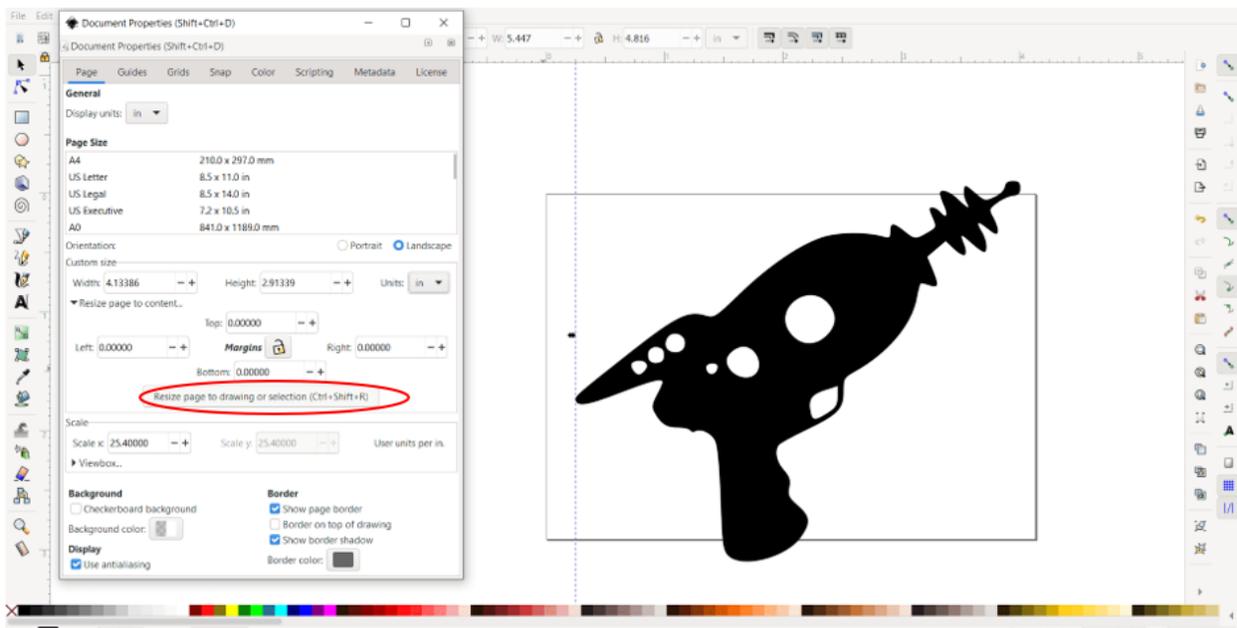
9. Save as .PDF or .SVG

Finally, you should save your file as a .PDF or a plain .SVG file.

Use the **File** menu and select **Save As** to save as either a PDF or SVG file.