Mill RIGHT

Basic CAD/CAM Tutorial for Inkscape: Engraving using G code tools extension

Version 1.00

Important safety rules for operating your MillRight CNC M3:

Never place your hands near a spinning end mill or bit.

Unplug the router before changing cutting tools.

Always wear eye and hearing protection while operating your machine.

Always run dust collection or wear a mask while performing a milling operation.

Do not leave the machine unattended while running a milling operation.

Do not operate your machine while under the influence of alcohol or drugs.

Do not place your hands between the bed and frame while the machine is moving.

Visually inspect wires prior to power up to prevent short circuits.

Ensure work pieces are properly secured before running a milling operation.

Inkscape is a graphic design program that allows you to create and edit vector files. Inkscape should not be used like a traditional CAD program where you precisely dimension features and set them relative to one another. It is, however, a great program for making things look cool and manipulating existing vector art that you might find on the internet. A Google search for something you are interested in and "SVG" at the end will likely yield something that you can open with Inkscape then, with some patience, practice, and skill, you can make those files ready for cutting on your machine.

I like to use Inkscape for generating either a SVG or DXF (two kinds of files often acceptable by more advanced CAD programs), or directly generating G code directly for cutting a particular font. There are two types of fonts: Outline fonts and single stroke fonts. Essentially every font that is presently on your computer is an outline font. An outline font is just that, one that has lines along the outline. A single stroke font is made of just one line for each feature of the font. Quite often, an outline font is not suitable for smaller engravings. A single stroke font is preferable. You can find some free single stroke fonts at this website: http://www.mrrace.com/CamBam_Fonts/. Download them, unzip them, and then copy and paste them into your system's Font folder. On a Windows system, you can find this in the control panel or simply search windows for "Font."

Here is a link to the Russian CNC forum where the author of Inkscape's G Code Tools extension has posted the program for free:

http://www.cnc-club.ru/forum/download/file.php?id=151&sid=a4904a0eda6563b48d9cd512dba5e807

Once you download this, you will need to unzip it to:

Program Files\Inkscape\share\extensions

The actual path may be different on your computer, but just make sure you get it in Inkscape's extensions folder.

Now, we are going to do a simple engraving using a single stroke font. I'm going to use a core box bit, but you could use a V bit or even a flat nose end mill if you would like. You can adjust the Z depth accordingly.

Let's get started. Open Inkscape, then go to File/Document Properties. It's not critical to do this, but I like to set my document size to roughly the size of the intended cut. Set the width, height, and units as I have shown below. There isn't a save button here. Once you set it up just close the document properties by clicking the X at the top right.

Default units: mm ∨

Portrait   ○ Landscape

Width: 180.00000   Units: mm ∨
Height: 180.00000
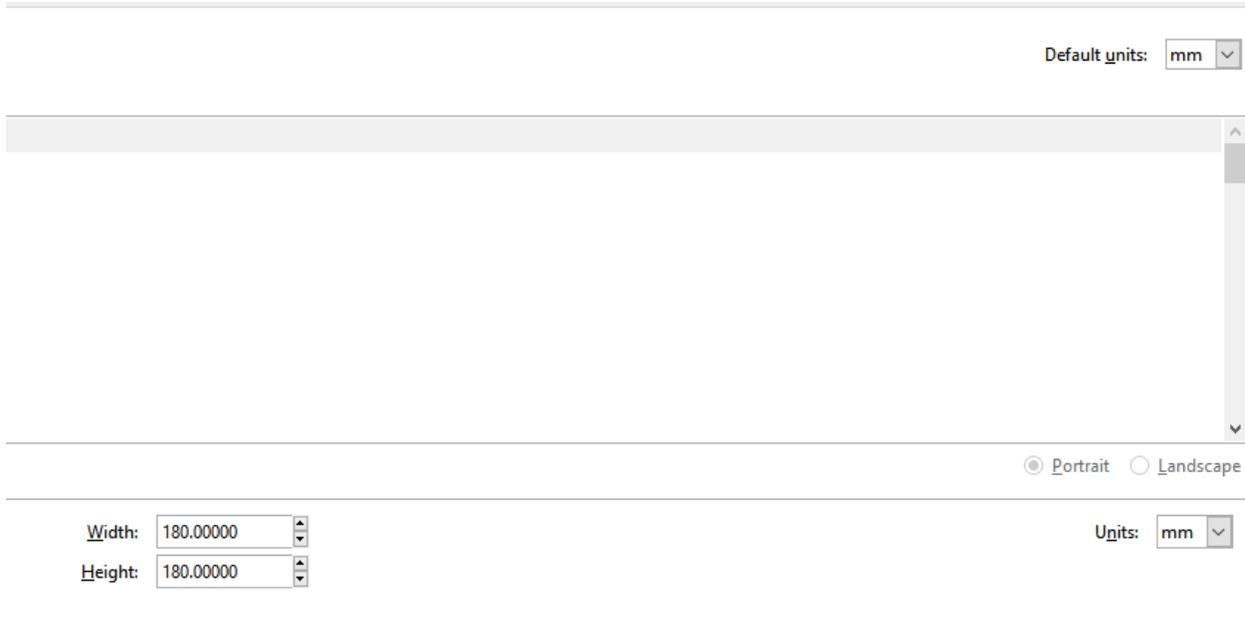
Now let's go ahead and get the "Fill and Stroke" menu up by hitting Ctrl+Shft+F (control and shift and F at the same time). This will allow us to change how the font we are about to create will look.
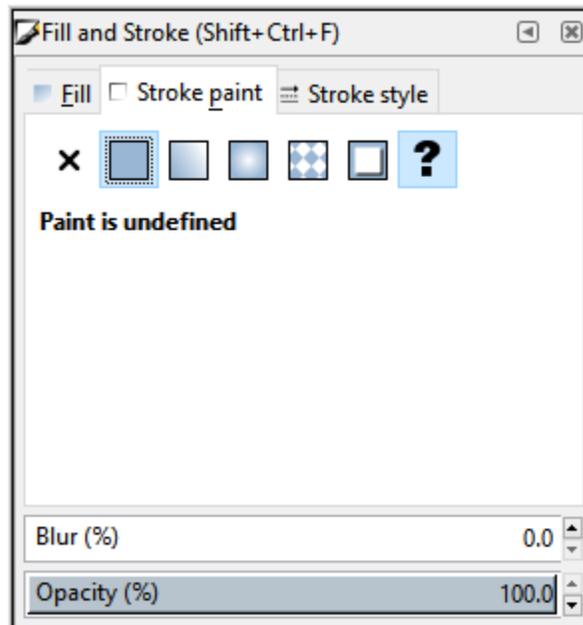
Many of Inkscape's most frequently used tools are on the left bar. Click the "A" icon. The default font is Sans Serif. Let's go with that for the moment. Click in the Workspace and type "Mill Right CNC".

You will notice at first all of the text is overlapping.

This is fixed by changing the "space between lines" to 1.15. This is at the top of Inkscape (see the two stacked "A" letters in the below picture) as long as you have the text tool selected. You might need to highlight all the text for the change to be accepted. After that, click the "Align center" button. That option is just below and to the right of the Help drop down.
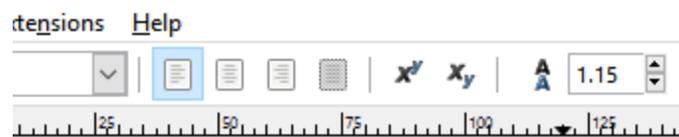

It should not look like this.

Now highlight the text and, from the font drop down towards the top left, select "CamBam 1". You should have installed CamBam single stroke fonts to your Fonts folder in a previous step. Notice how when you do this it doesn't look like there's any text. That's because a single stroke font can't have a fill. From the Fill and Stroke menu, select Stroke Paint and then click the solid box. Now click the Stroke Style tab from the Fill and Stroke menu and set the units to "mm". Set this value to the engraving width that you are going to want. **TAKE NOTE THAT THIS VALUE DOES NOT ACTUALLY AFFECT HOW WIDE THE ENGRAVING WILL END UP.** This setting simply helps you visualize your spacing when the work actually gets done. Inkscape in general will simply follow a path (same as a trace operation in Autodesk Fusion 360). As such, the engraving width is a function of the type of tool you are using (V bit vs ball nose vs flat end mill) and, in the case of rounded nose bits and v bits, the depth as well.
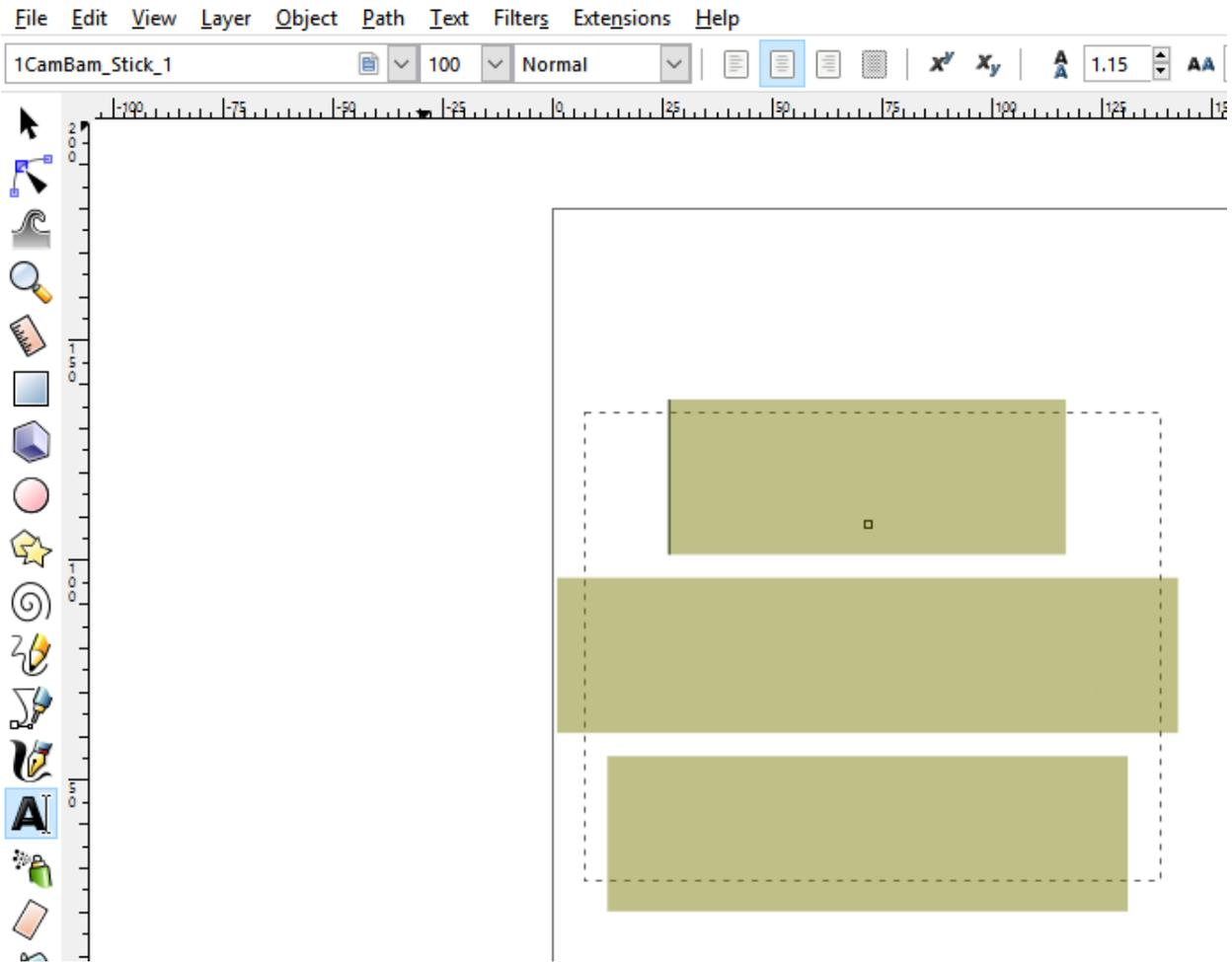
I set mine to 6 mm. I'm using a quarter inch core box bit, so that's about right. Now I'll scale it and set the bottom left of the selection box to the bottom left of the workspace. Click the cursor icon on that left toolbar and make sure the text you just typed in is selected. Make sure the padlock icon is selected, then type in "180" next to "W". Ensure that the units are in mm. The height will adjust proportionately. Now set X and Y both equal to 0. Note that this is setting the center of the lines on the bottom to the 0 point, so if you have a 6mm stroke width then you will have 3mm of overhand below and to the left of the origin. See the below picture.

Now's a good time to save what you have, then go to File/Save As and created another version. Name it "MillRight-2" perhaps.
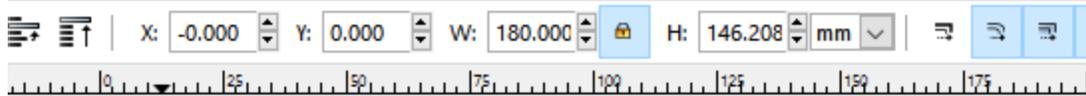
We just saved it as a new document because we are about to take this text object and change it to a vector path. With the text selected, click on Path/Object to Path.
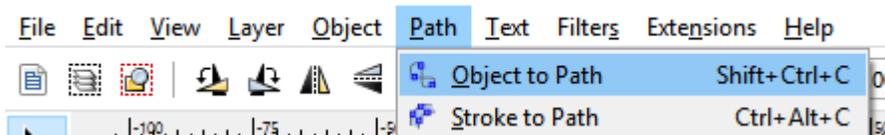

It won't seem like anything has happened, but Inkscape has now created a vector path that we can generate code from. The text is no longer editable with the text tool.

We'll move into the CAM (g code generation) portion of this now. If you've installed the Gcodetools extension as described in a previous step, then you'll find it under the Extension menu at the top. The G code tools in Inkscape are extremely basic and some of them flatly do not work (at least not for me.)

There are no advanced features such as ramping, helical bores, lead ins, tabs, etc. It allows you to create pockets and follow vector lines. We are going to follow vector lines using the "Path to Gcode" feature.

First, set orientation points by going to Extensions/Gcodetools/Orientation Points.
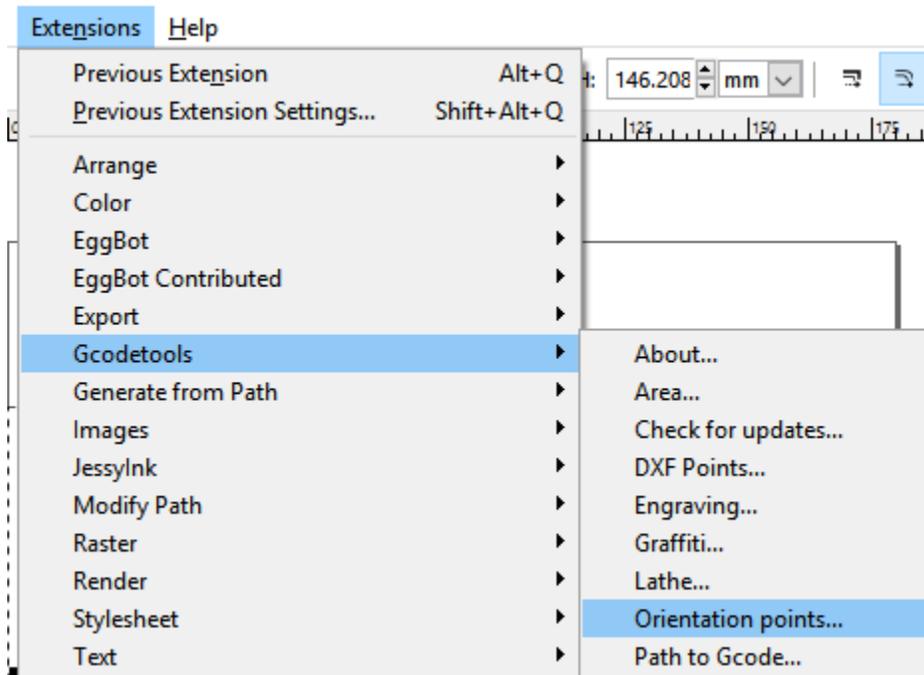

You'll set the final depth of your engraving here. The stock top (Z surface) should be 0 and the Z depth should be negative. I went with -2.5 because I'm using a core box bit that is ¼" (6.35mm) wide. Plunging 2.5mm is going to give me almost a full tool width engraving stroke. You'll need to adjust this based on your tool. For instance, a 90 degree V bit's engraved line is twice as wide as the depth of pass: 1 mm deep gives a 2mm wide line. With a 45 degree V bit, the width of the engraved line is about 83% of the depth of cut: A 3mm deep pass gives a 2.49mm wide line.
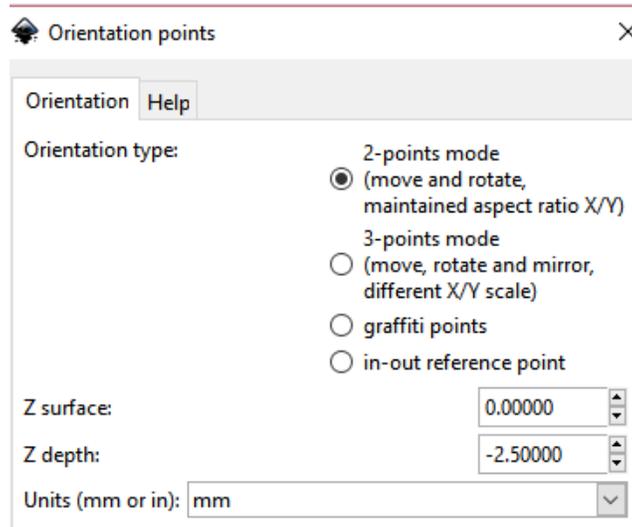
Inkscape will place some text in your work space that describes the origin and another point 100mm from to original. Do NOT mess with what it puts there or try to move or modify it in any way.

Now you need to add a tool. Select "Tools Library" from the same menu. Select Cylinder. Cone might be useful for V bits, but the documentation for the use of this is quite sparse and I've not tested it. Just select Cylinder and click apply.
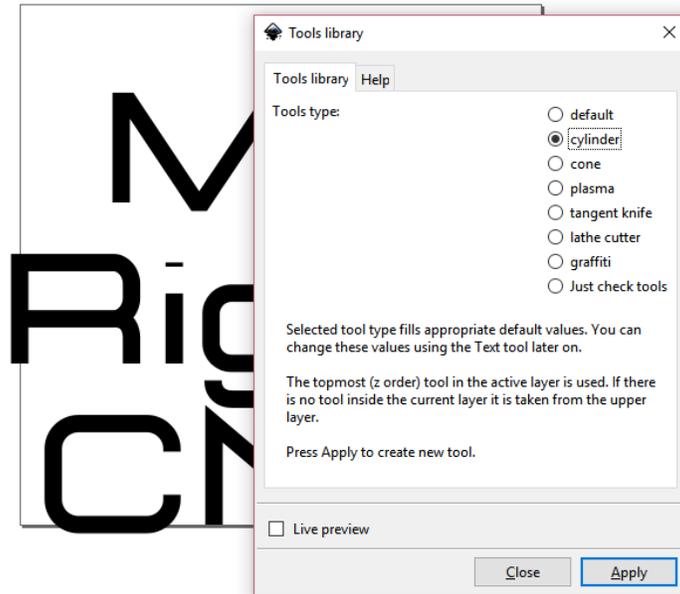
Now it will have added a green box somewhere in the workspace. Sometimes Inkscape adds it in an odd place so you may have to zoom out to find it. Zoom in on this green box. You edit the milling settings by using the text tool (the A icon on the left toolbar). Some of these settings just do not seem to matter. The only things that are going to matter are feed (this is your horizontal feed rate), penetration feed (this is your plunge feed rate), and depth step (how far it goes down in Z on the way to the bottom depth declared in your orientation points). Penetration angle should be a ramp angle, but it doesn't seem to work. Inkscape will only plunge to enter the stock, so be aware and adjust your plunge rate accordingly. Notice that I set a final depth of -2.5 but a positive depth step. It's going to take one pass 1.5mm deep then a final pass at 2.5mm deep.

Now, select the text that you have created. Go to Extensions/Gcodetools/Path to Gcode. Make yours look like mine. Note that there isn't a search window for the directory selection. You will have to copy and paste in the directory that you want the output file to go. The post processor of "Round all values to 4 digits" is very important. Your control software will choke on most anything with more than 4 decimal points. The Z safe is how high your tool will lift before rapids, so get it up at least 5mm. Don't click on Live Preview as it is prone to crash Inkscape.

Path to Gcode ✕

Path to Gcode | **Options** | Preferences | Help

Scale along Z axis: `1.00000`

Offset along Z axis: `0.00000`

☐ Select all paths if nothing is selected

Minimum arc radius: `0.10000`

Comment Gcode: [   ]

☐ Get additional comments from object's properties

☐ Live preview

[Close] [**Apply**]

Now the Path to G Code tab is very important. The depth function must have the letter d in it. The cutting order should be subpath by subpath. You must by on this tab when you click apply.
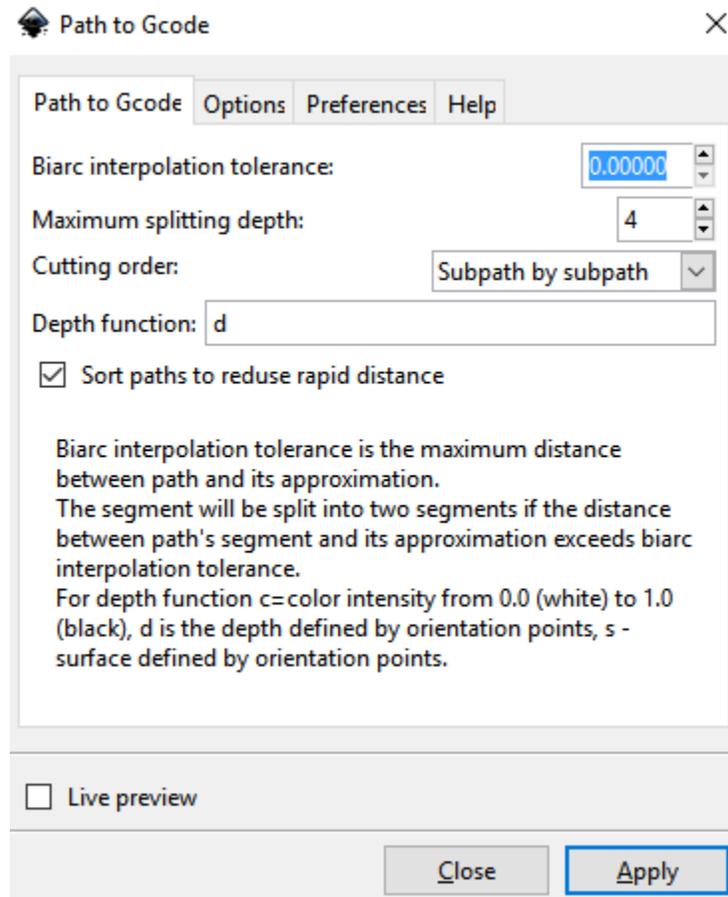
It won't give you a confirmation. It will just put the file (with no file extension) in the directory you specified. Go to that directory, and right click that file and click rename. Add .txt onto the end of that file name. You can now open it up with a text editor if you so desire, but don't change anything. This is the G code file that you will send to your control board using such software as Grbl Controller or Universal Gcode Sender.