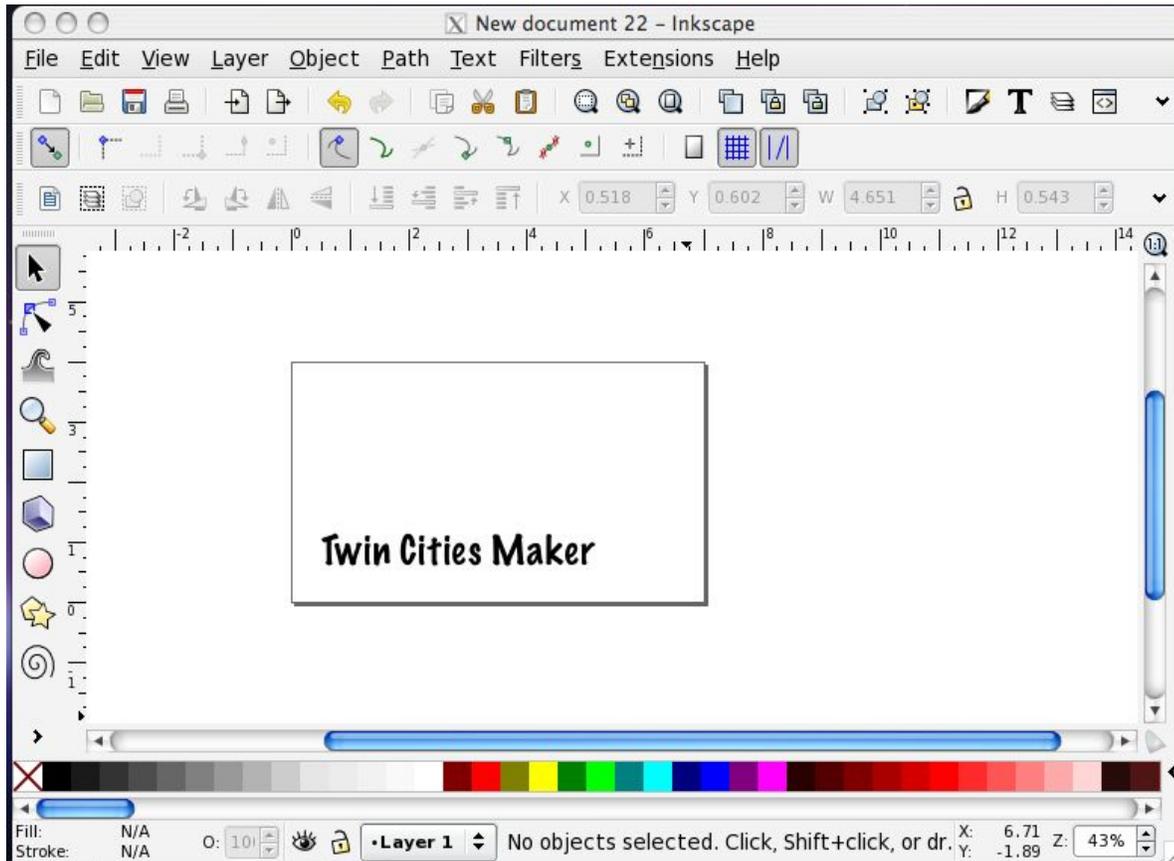


Lets start with a simple task we want to engrave some text into an object.

Start Inkscape, select a document size.

Pick a font and type some text. You should end up with a screen that looks something like this.



You will want to ensure that the text is sufficiently large that you can “outline” the letters with a bit you have available without destroying the neighboring letter, “V” bits or something similarly pointy are best for this, Gcodetools refers to this type of tool as a cone.

Introducing the Gcodetools Extension. At the top of Inkscape you will see a menu for extensions Open the Gcode-tools-dev “All in one” option.

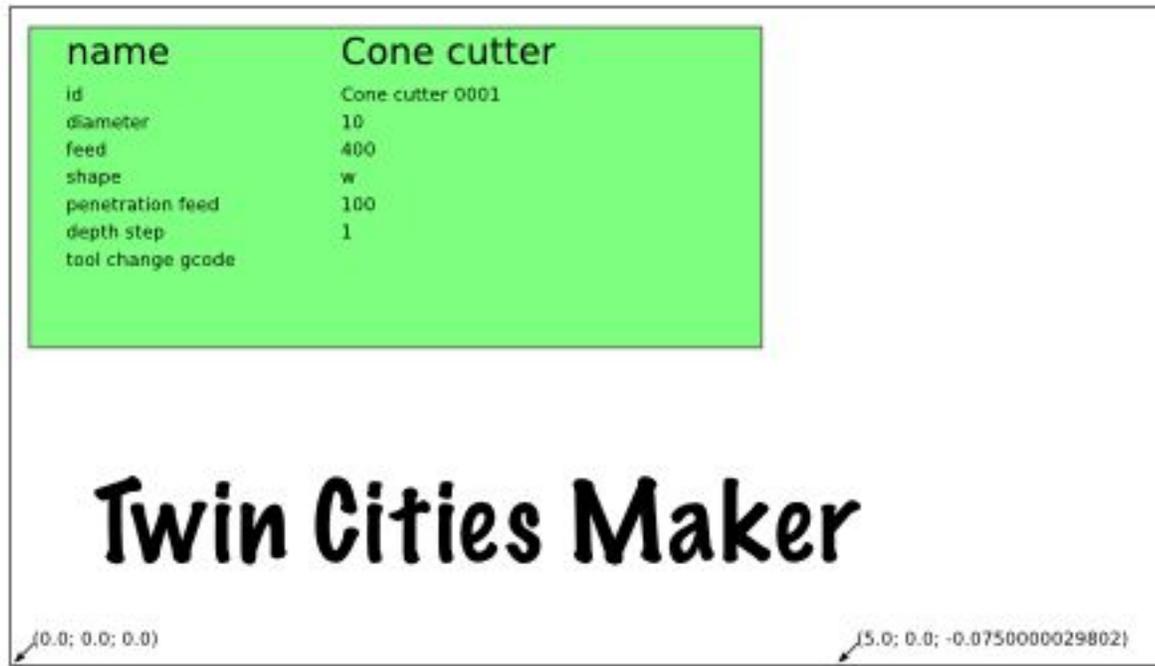
Start at the Right and set your. (See screen shot appendix at the end for example settings)

- Preferences
- Options
- Orientation
- Tool

Hit Apply for each tab before moving on to the next.

Close the Gcodetools Extension

Your screen should now look something like this. You can drag the green tool box to any convenient location, the orientation indicators can also be moved, but we will leave them at the bottom of the page and touching the bottom left corner for now.



The added tool and orientation objects have text elements that can / must be adjusted to correspond to the tool you will be using and the location referenced (optional) Edit the tool diameter to represent the bit you will be using, in the case of a cone the diameter is the shaft and not the point size (at least I think it is) An 1/8 inch .125 or 1/4 inch .25 inch shaft size would be common.

Feed speeds depend upon the tool, the Gcodetools defaults assume metric systems and likely need to be adjusted, I.e 400 IPM is extremely fast for a small hobby machine. For the Hack Factory CNC Router a feed of 80 with a penetration or plunge feed of 30 is reasonable.

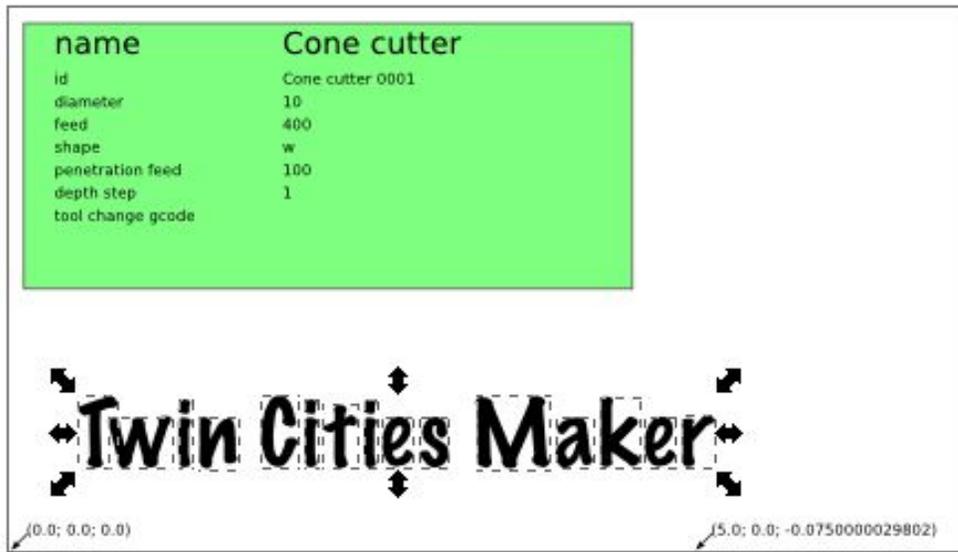
Depth step determines how many passes will be made for each path being cut, leave this at 1 for now.

Select the text, and convert it to a path, doing so make take a combination of the following commands

- Shift+Ctrl+G – to ungroup objects
- Shift+Ctrl+C – to turn everything selected into a path.

Hint: If you watch the bottom status window in Inkscape, below the color selection bar you can see what type of objects you currently have selected, this status will change as you execute the above commands.

Once you have successfully converted the text into paths, you should have something like this in Inkscape, not the status bar indicating “15 objects of type path”



We are now ready to generate the gcode to engrave the text.

Under Extensions Open the Gcode-tools-dev “All in one” option, and go to the “Path to Gcode” tab.

I'm using the following settings here:

- Biarc interpolation tolerance = 1
- Maximum splitting depth = 2
- Cutting Order = layer by layer

Change your settings if required and remember to hit “Apply” Close Gcode-tools.

Your gcode file will be in the directory you set under preferences.

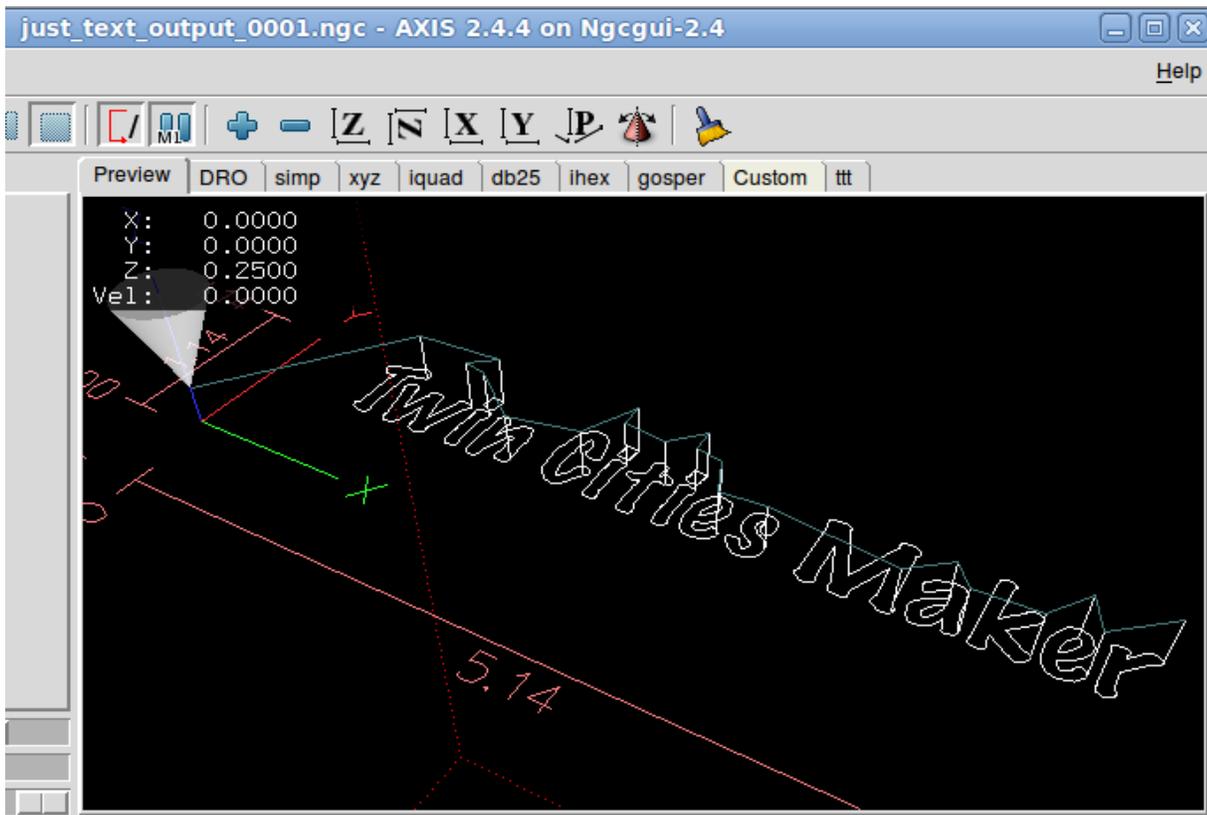
Open it in a gcode viewer, free version of these exist for most operating systems

Pleasant3D for the Mac (requires 10.6)

CNCViewer for Windows

EMC for Linux.

The gcode should look similar to this example, as seen in EMC.



The vertical lines indicate the plunge paths the tool will take, the blue green lines are the movement between cutting paths at the safe height, i.e. when the tool will be lifted clear of the workpiece to move to the next area to be cut.

So that's a quick overview on Inkscape and the gcodetools extension.

Next lets define a few terms so we have a vocabulary for the more complex operations to follow.

Engraving: Cutting the outline of a path like we demonstrated above

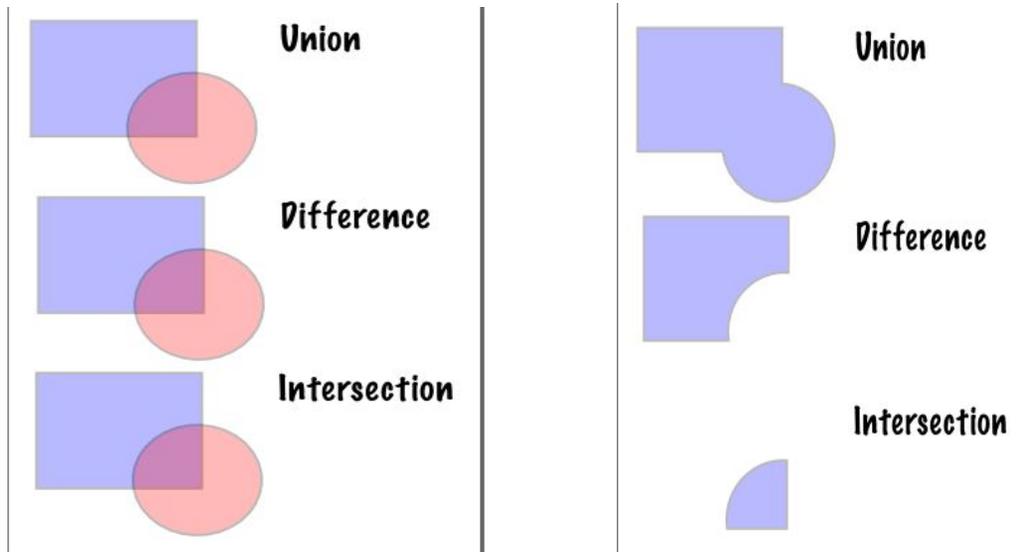
Pocketing: Removing material from a specific area of a work piece to a certain depth

Drilling: Drilling a hole through the material.

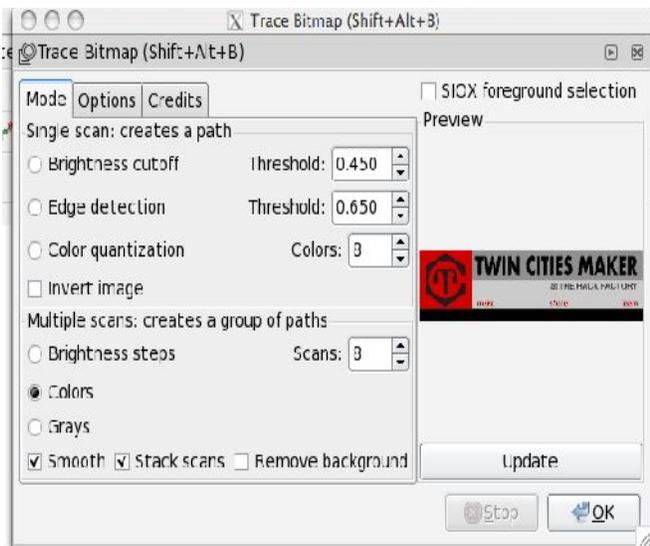
Cutting a profile: cutting along a path through the material.

Height map or 3D relief carving: This will be covered outside of Inkscape, and demonstrated using EMC, though other tools can be used to generate such code.

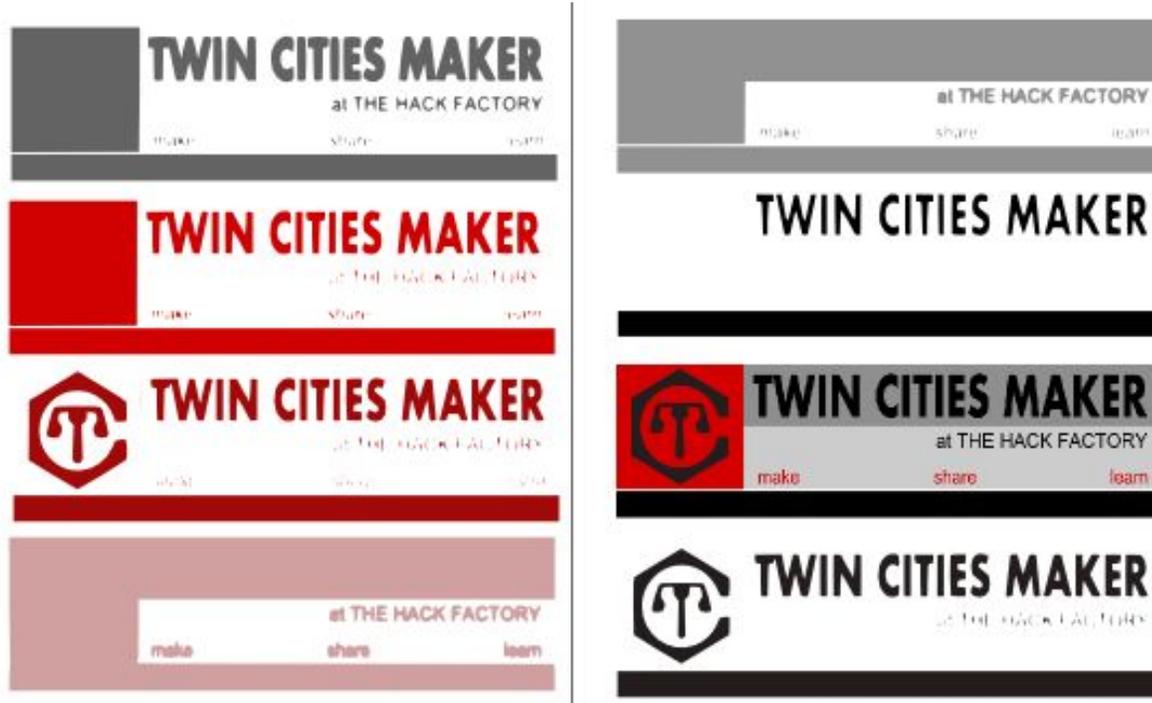
Before we getting into Pocketing and more complex operations it is helpful to be aware of the tools within Inkscape that can be used to manipulate paths. Inscape can modify paths that overlap to create the desired shapes, and setup areas to be machined such as to create an area to pocket, in many ways, three of the most common are demonstrated below.



Another handy feature of Inkscape is it's ability to create paths from raster based files with distinct areas. (so what does that mean) well typically not a photograph of messy organic stuff, like people, and nature, though it may work for that. What I'm referring to is raster files or pictures of objects with distinct areas, Here's a quick example of the Path → Trace Bitmap tool in Inkscape



These are the results of creating 8 paths from the bitmap / raster image, as you can see it autogenerates some paths that can be very handy for making tool paths without having to trace or redraw things yourself.



OK lets use what we have learned and create some more complex tool paths to make what might be a typical sign making process.

The images below are example settings and may need to be adjusted for your needs and tool.

Path to Gcode Area Area artefacts Engraving DXF points Tools library Orientation Options Preferences Help

File:

Add numeric suffix to filename

Directory:

Z safe height for G00 move over blank:

Units (mm or in):

Post-processor:

Path to Gcode Area Area artefacts Engraving DXF points Tools library Orientation Options Preferences Help

Scale along Z axis:

Offset along Z axis:

Select all paths if nothing is selected

Minimum arc radius:

Path to Gcode Area Area artefacts Engraving DXF points **Tools library** Orientation Options Preferences Help

Tools type:

- default
- cylinder
- cone
- plasma
- tangent knife
- lathe cutter
- graffiti
- just check tools

Selected tool type fills appropriate default values. You can change these values using the Text tool later on.

The topmost (z order) tool in the active layer is used. If there is no tool inside the current layer it is taken from the upper layer.

Press Apply to create new tool.

Orientation type:

- 2-points mode
 - (move and rotate, maintained aspect ratio X/Y)
- 3-points mode
 - (move, rotate and mirror, different X/Y scale)
 - Add graffiti points

Z surface:

Z depth:

Units (mm or in):

Orientation points are used to calculate transformation (offset, scale, mirror, rotation in XY plane) of the path.
3-points mode only: do not put all three into one line (use 2-points mode instead).

You can modify Z surface, Z depth values later using text tool (3rd coordinates).

If there are no orientation points inside current layer they are taken from the upper layer.

Do not ungroup orientation points! You can select them using double click to enter the group or by Ctrl+Click.

Now press apply to create control points (independent set for each layer).