How to Laser

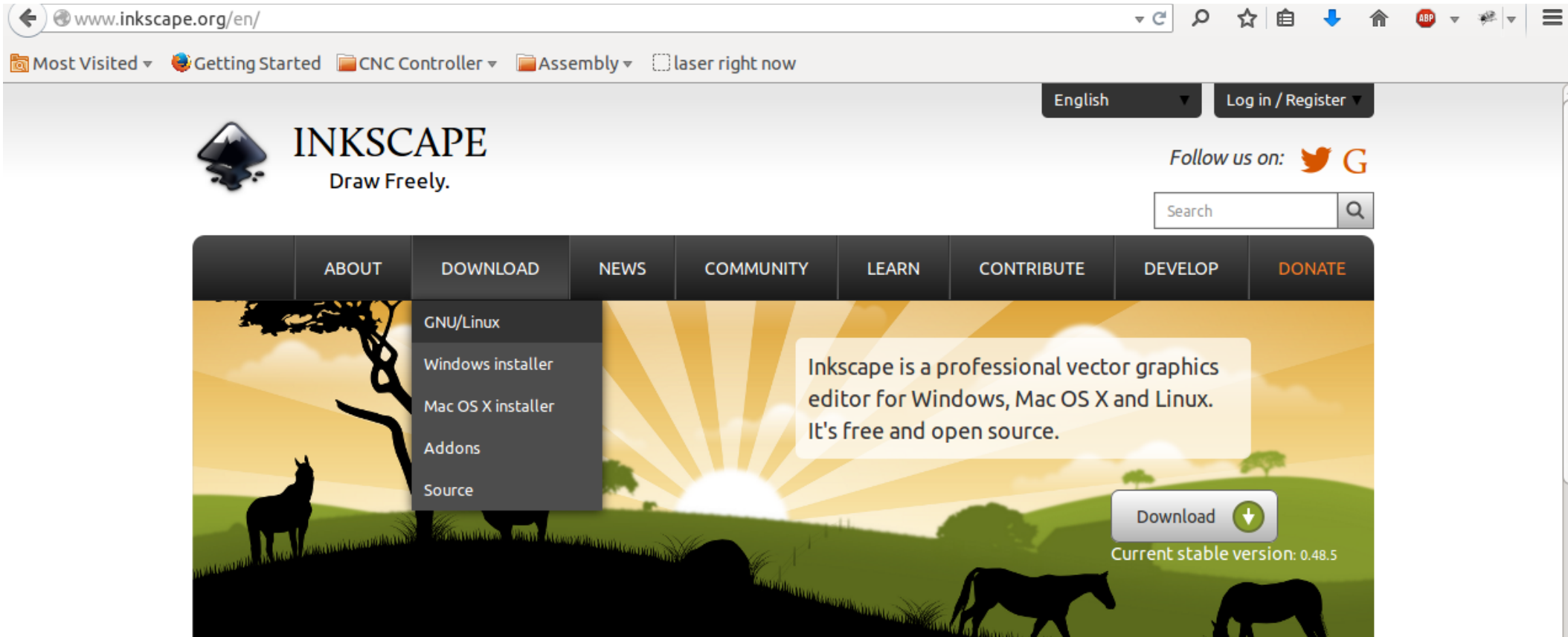
Computer Setup
Material Selection
Running the Laser

Computer Setup

Inkscape Install

Download and install Inkscape

Go to www.inkscape.com and follow the install instructions



Download gcodetools

Gcodetools is an extention for Inkscape.
It converts your vector drawing to a machine readable language called g-code

search for “gcodetools inkscape” and if you got to a russian forum, you're in the right place

or click here for the link to the file

Install gcodetools

Toss the extracted gcodetools files into either:

Program Files\Inkscape\share\extensions\

or

/usr/share/inkscape/extensions/

Depending on what operating system you're on

**Don't put the files in a sub-directory. Just put all of the files directly in 'extentions'

Material Selection

Do's:

Acrylic, Wood, Paper, Polystyrene

Dont's:

PVC(REALLY DON'T), Shiny things, Polycarbonate, ABS, Pizza

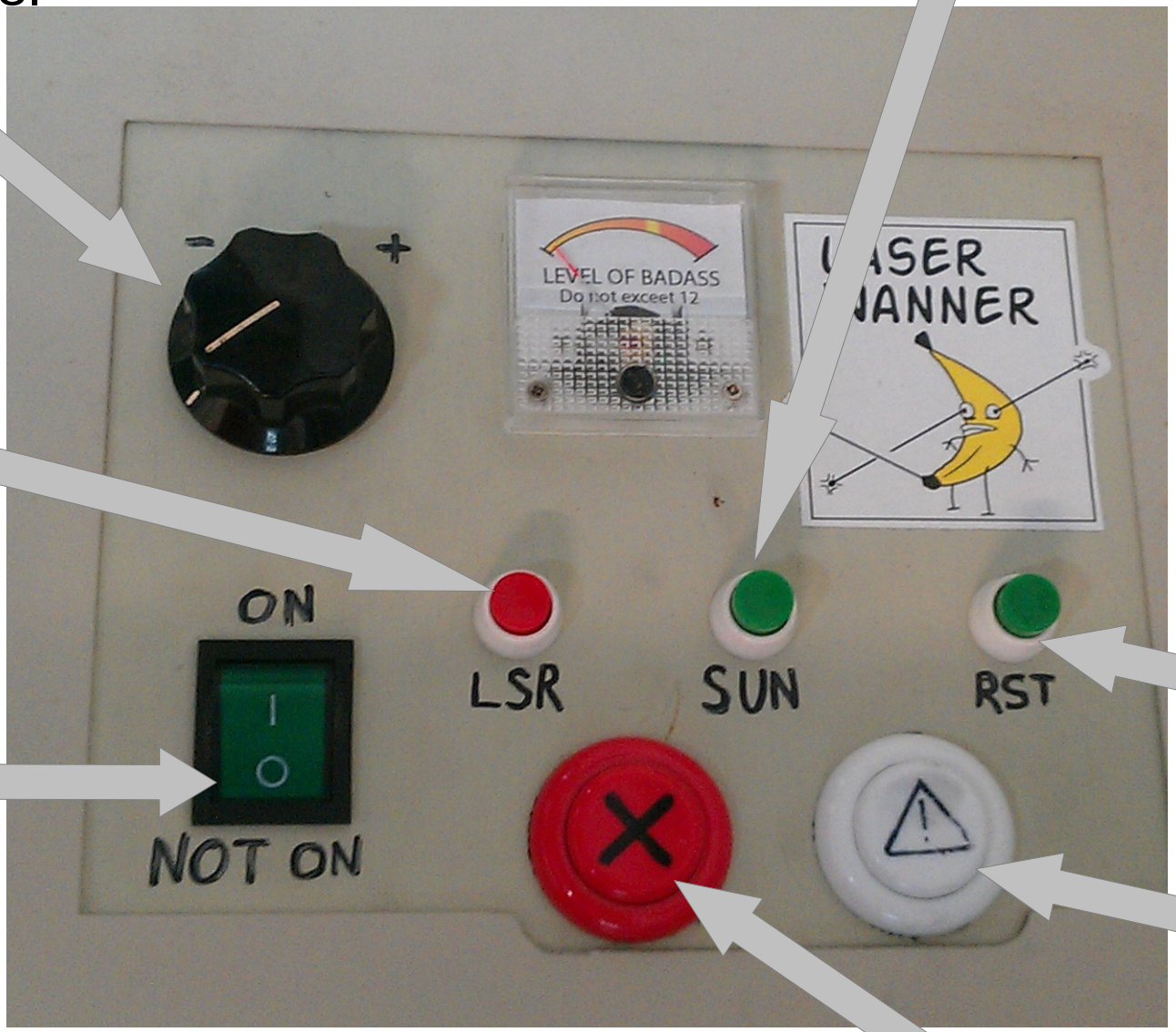
If you aren't sure, just use some of the scraps hanging around the laser.

Laser Operation

Turning on the Laser

1. Turn on the power strip under the table
2. Flip the green switch to the “ON” position
3. Close the door
4. Turn the black nob to 7 o'clock

Controls



Work Light On/Off

Badass Level Adjuster

Laser Enable/Disable

Big Green Power Switch

Doesn't work

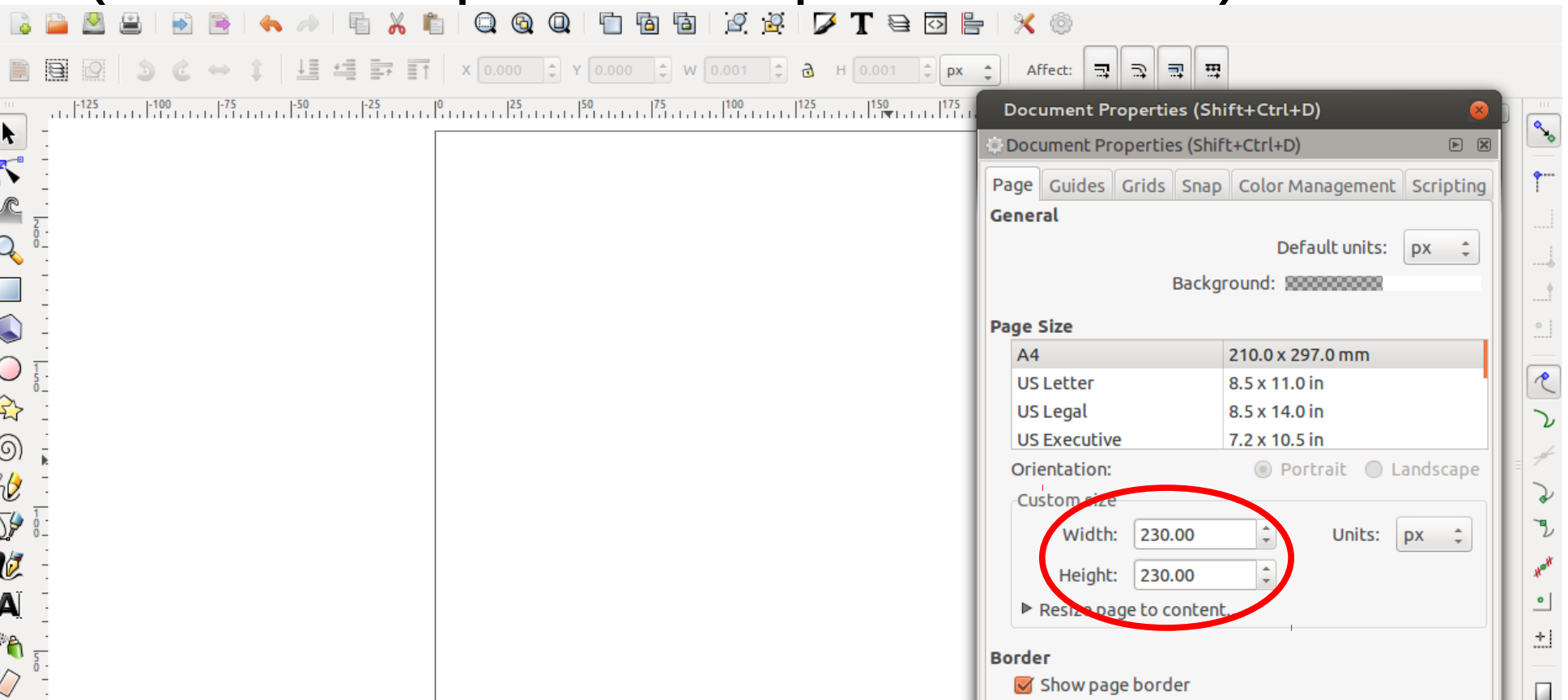
Test Fire

Doesn't work

Lets cut a part: Rectangle

1. Open up Inkscape
2. Set the page size to 230 by 230 pixels

(note that 1 pixel will represent 1 mm)

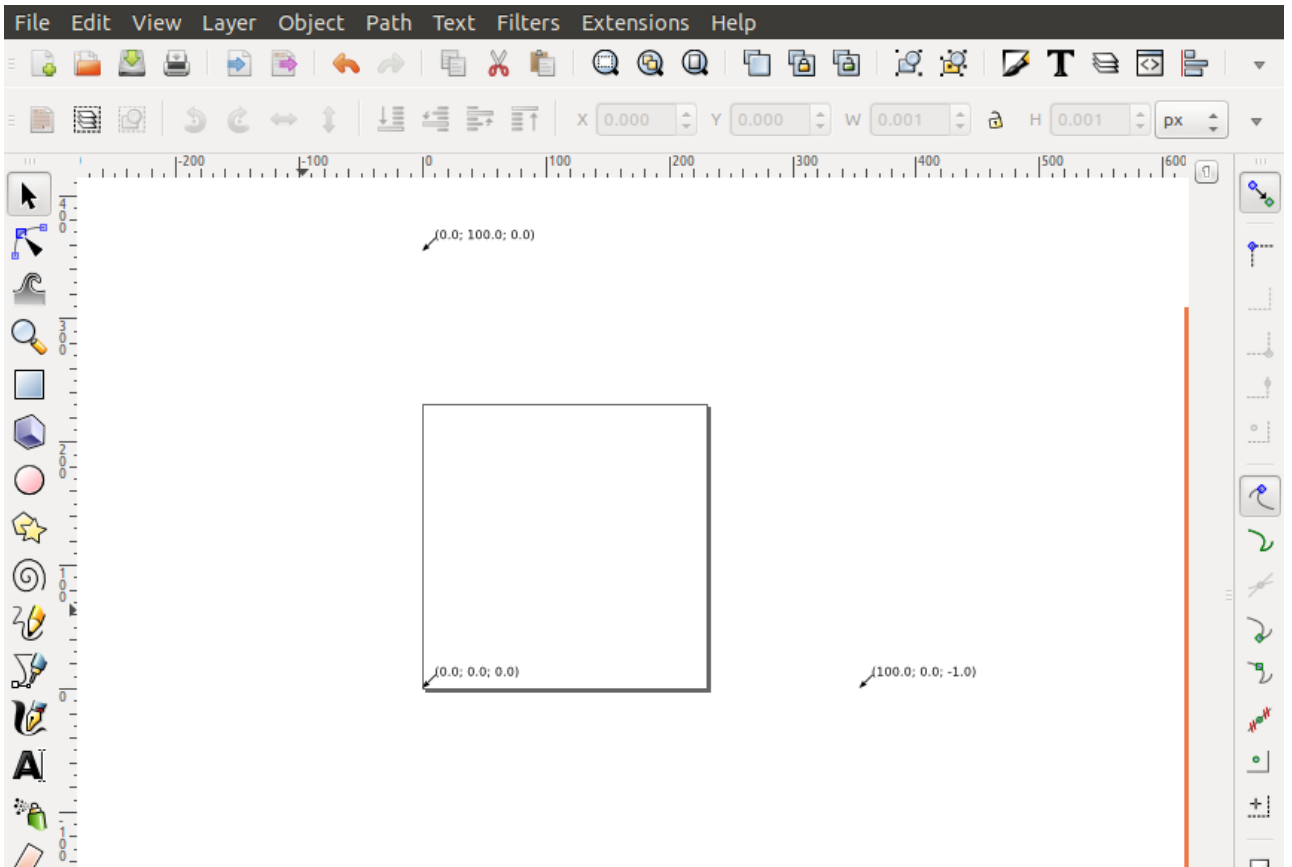


Lets cut a part: Rectangle

3. Go Extentions>Gcodetools>Orientation Points...

4. Select “3-Point Mode” and press apply

Your canvas should now look like this:

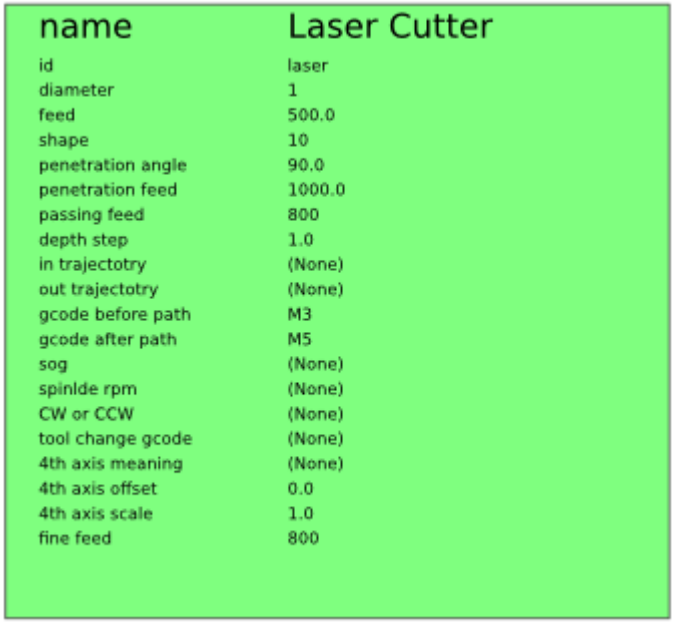


Lets cut a part: Rectangle

5. Go Extentions>Gcodetools>Tools Library
6. Select “Default Tool” and press “apply”

It should appear somewhere above the canvas

7. Fill it out like this:

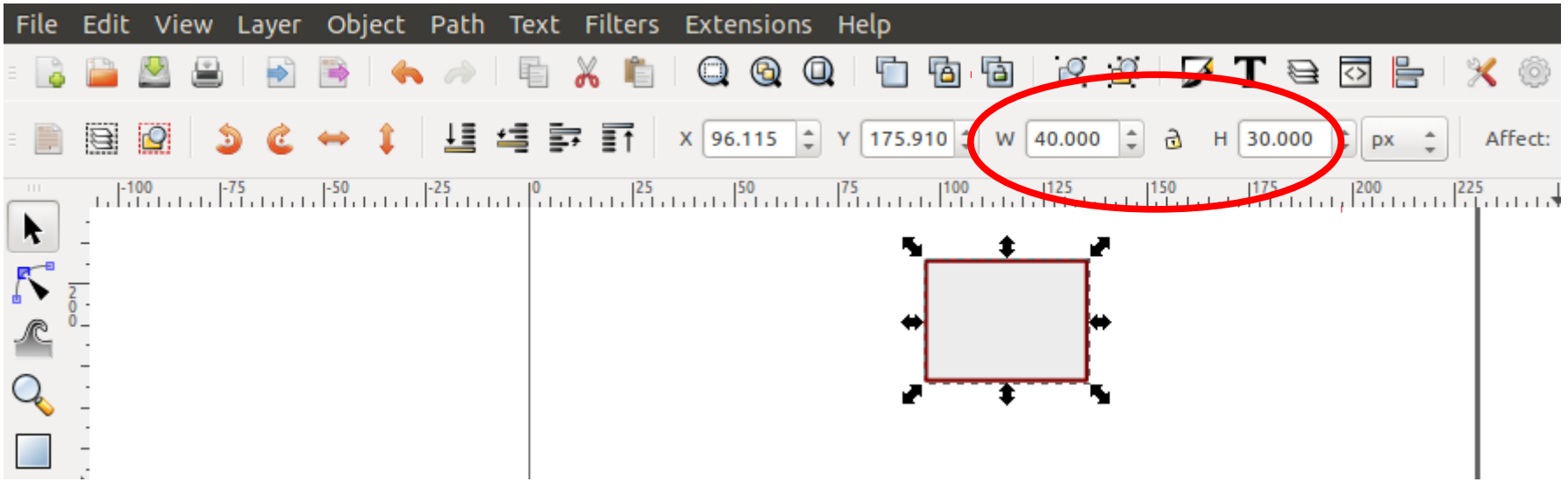


Lets cut a part: Rectangle

8. Select the rectangle tool and draw a rectangle

9. Change the dimensions of the rectangle to 30mm to 40mm

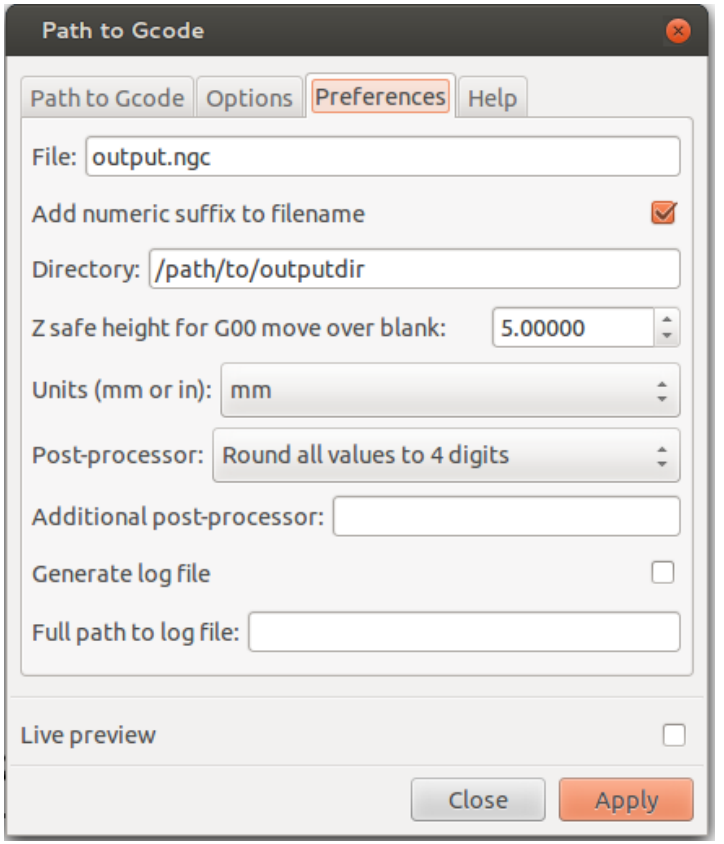
10. You need to turn things that arent “paths” into “paths” by going Path>Object to Path



Lets cut a part: Rectangle

11. Put the rectangle in the bottom corner where the orientation arrow is.
12. Go Extentions>Gcodetools>Path to Gcode
13. Set the Preferences:
14. Click on Path to Gcode

tab and click “apply”



Lets cut a part: Rectangle

15. You just created a text file called output-something.ngc. Get rid of the first M3.

output_0002.ngc

```
%
(Header)
(Generated by gcodetools from Inkscape.)
(Using default header. To add your own header create file "header" in the output dir.)
M3
(Header end.)
G21 (All units in mm)

(Start cutting path id: rect4541)
(Change tool to Laser Cutter)

G00 Z 0.0000
G00 X 11.9218 Y 36.1061
M3
G01 Z 0.0000 F 1000.0000(Penetrate)
G01 X 31.6780 Y 36.1061 Z 0.0000 F 500.0000
G01 X 31.6780 Y 23.5031 Z 0.0000
```

Lets cut a part: Rectangle

16. Turn laser on by turning on the power strip followed by the green laser power switch.

17. Nav to laser.interlockroc.net in your web browser

18. Copy and paste your G-Code into the text box.

19. Select “mm” for the units

Lets cut a part: Rectangle

20. Adjust the laser head to the bottom left corner of your material.

21. Close Lid

22. Adjust power knob to 12 o'clock position

(or wherever you want)

23. Press “Send G-Code”

Hey, be safe. You could shoot your eye out with this thing.