

# How to Laser

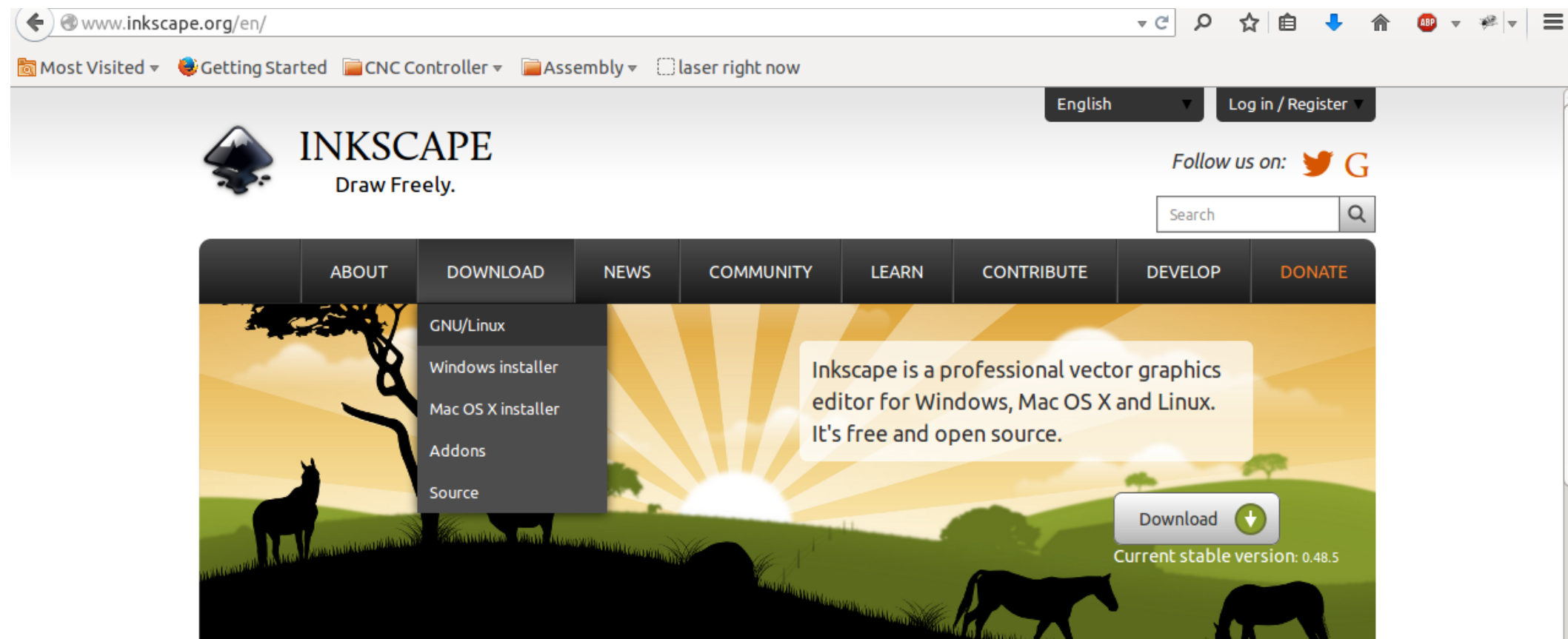
Computer Setup  
Material Selection  
Running the Laser

# Computer Setup

# Inkscape Install

Download and install Inkscape

Go to [www.inkscape.com](http://www.inkscape.org) and follow the install instructions



The screenshot shows the Inkscape website homepage. The browser address bar displays [www.inkscape.org/en/](http://www.inkscape.org/en/). The page features the Inkscape logo and the tagline "Draw Freely." in the top left. On the right, there are links for "English" and "Log in / Register", along with social media icons for Twitter and Google+. A search bar is also present. A navigation menu is visible, with the "DOWNLOAD" tab selected, revealing a dropdown menu with the following options: GNU/Linux, Windows installer, Mac OS X installer, Addons, and Source. The main content area has a background image of a sunset over a field with silhouettes of horses. A text box in the center reads: "Inkscape is a professional vector graphics editor for Windows, Mac OS X and Linux. It's free and open source." Below this text is a "Download" button with a downward arrow icon and the text "Current stable version: 0.48.5".

# Download gcodetools

Gcodetools is an extension 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 'extensions'

# 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

Badass Level  
Adjuster



Work Light  
On/Off



Laser  
Enable/Disable



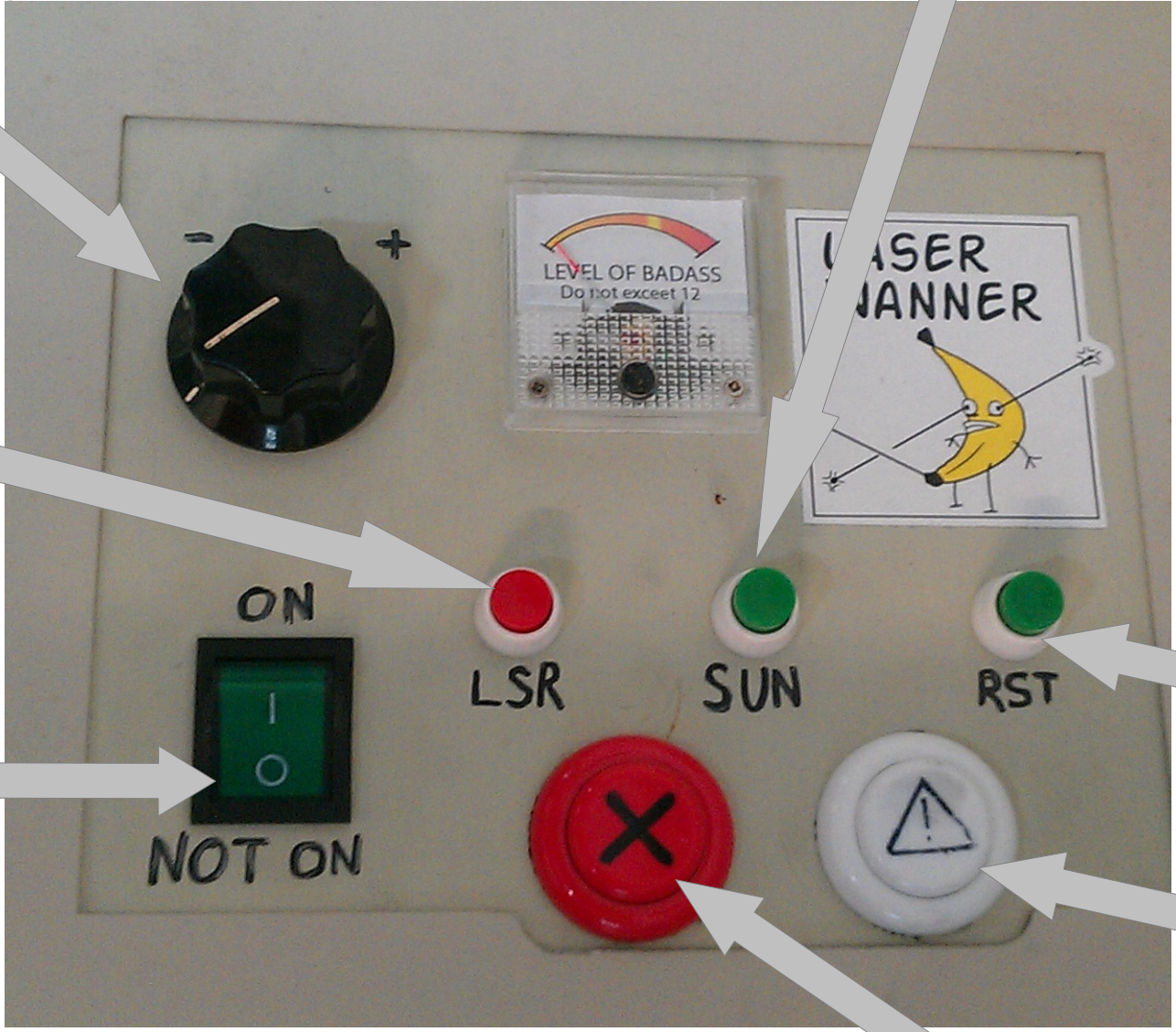
Doesn't work

Big Green  
Power Switch



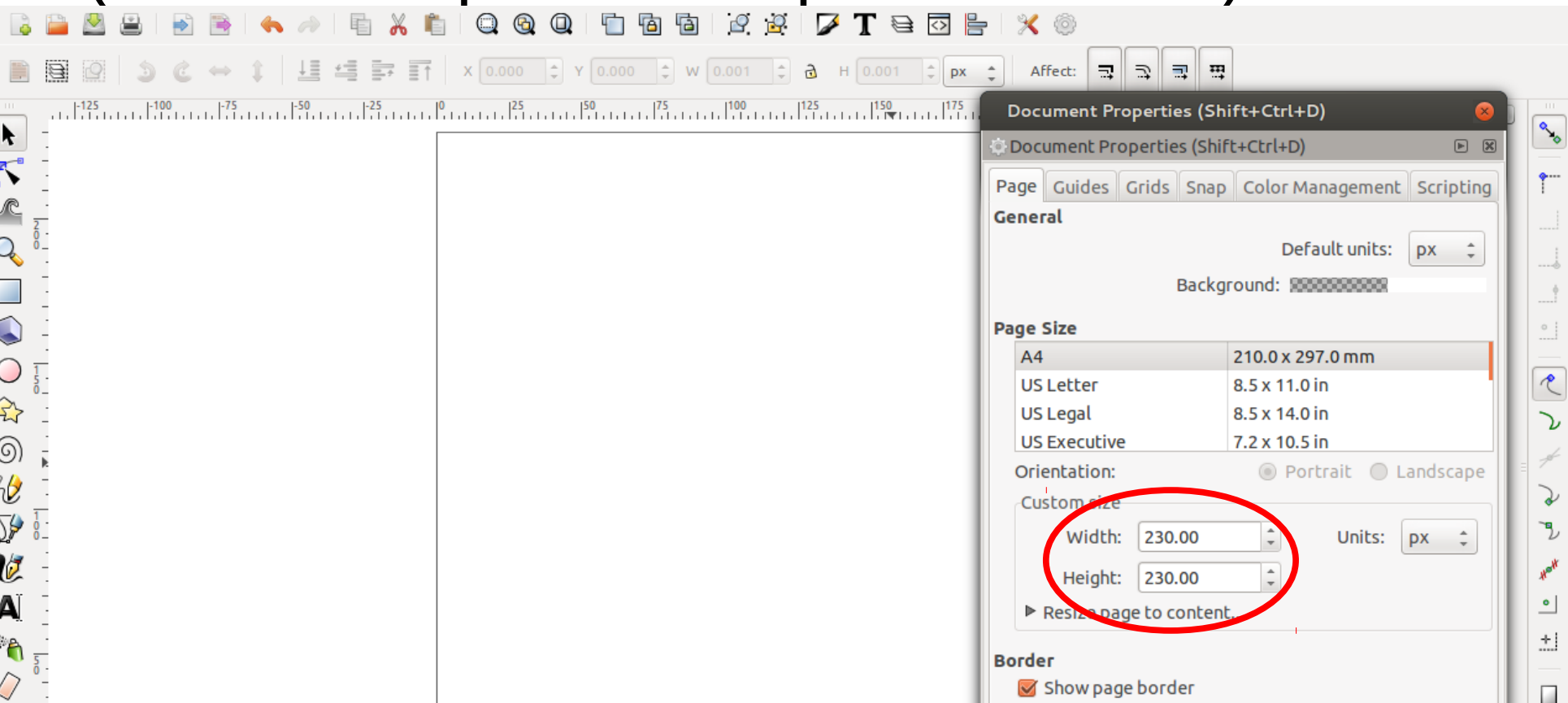
Test  
Fire

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)

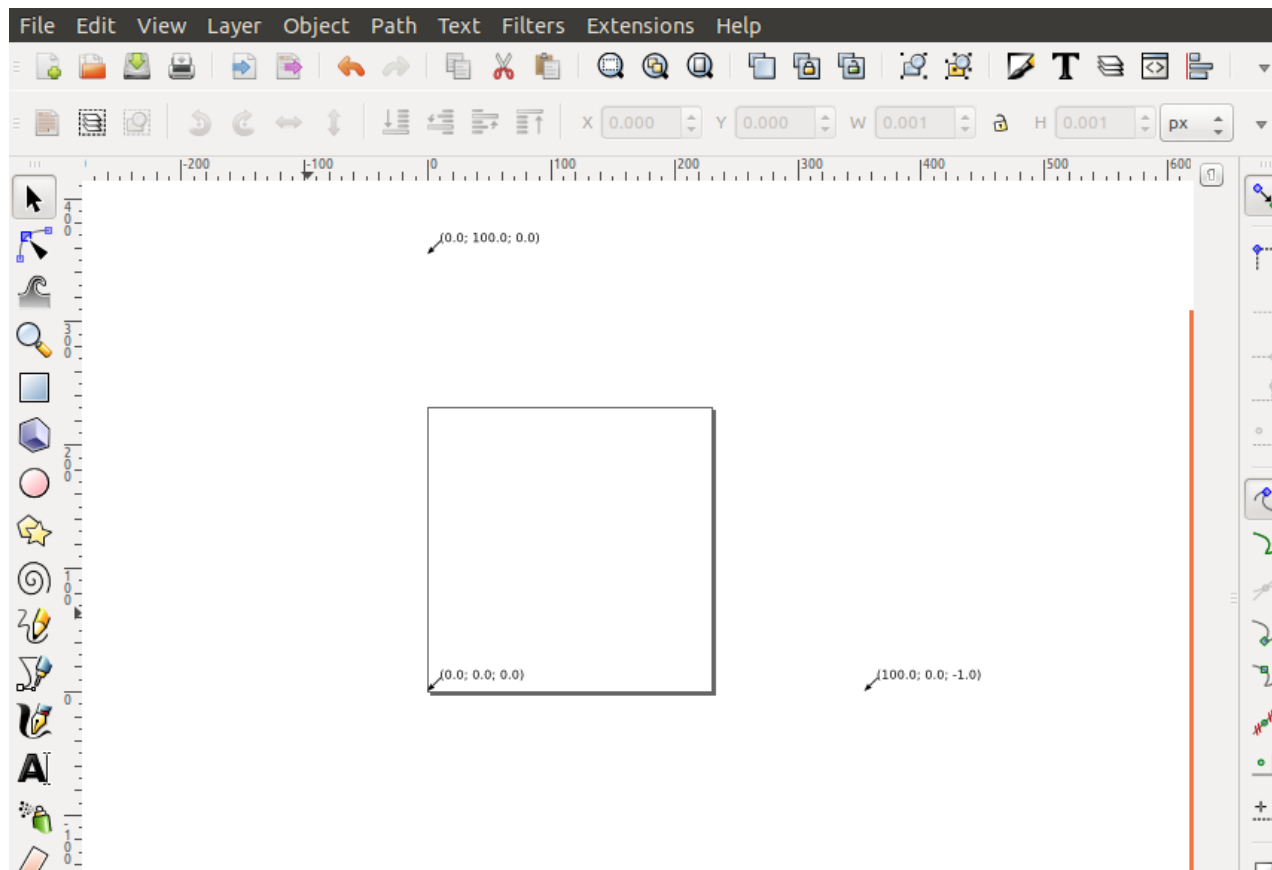


# Lets cut a part: Rectangle

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”

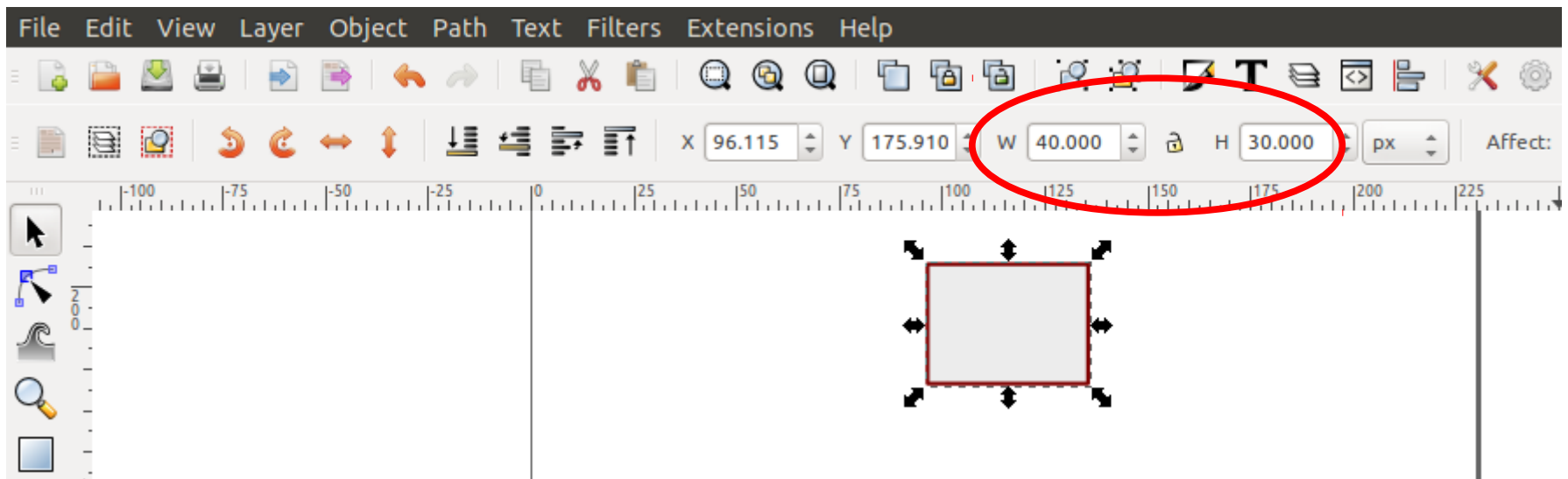
It should appear somewhere above the canvas

7. Fill it out like this:

name	Laser Cutter
id	laser
diameter	1
feed	500.0
shape	10
penetration angle	90.0
penetration feed	1000.0
passing feed	800
depth step	1.0
in trajectory	(None)
out trajectory	(None)
gcode before path	M3
gcode after path	M5
sog	(None)
spinlde rpm	(None)
CW or CCW	(None)
tool change gcode	(None)
4th axis meaning	(None)
4th axis offset	0.0
4th axis scale	1.0
fine feed	800

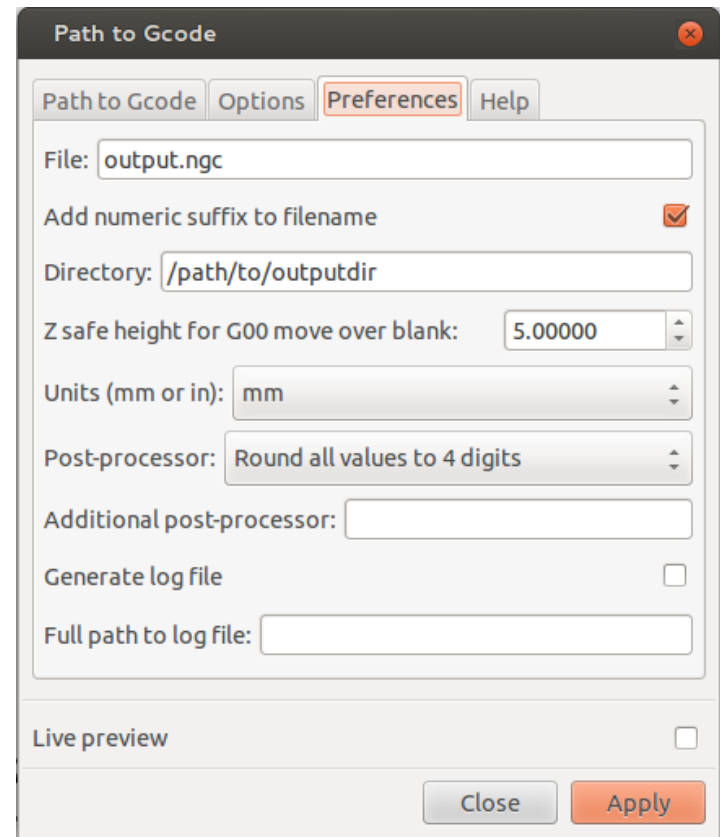
# Lets cut a part: Rectangle

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](http://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.