In this video I have shown how to convert an Autocad drawing to Gcode for CNC machining using Lazycam.

At first a rectangle is Exploded into path using ‘explode’ command. Then ‘monotext’ is used as font because other fonts would break into double lined texts when using ‘txtexp’ command to Explode the text to path, for the sake of simplicity.

The completed file is then saved as AutoCad R12 DXF format which for my case were best supported by the LazyCam software.

Then it is imported to LazyCam which produces the Gcode for CNC Machining. LazyCam has several profiles, in this case Mill is used. Then clicking on the Optimize button optimizes it’s rapids (by which path the machine moves rapidly) and then resets the origin to 0,0 position. Then there is scale adjustments, first it is told to measure in mm. By clicking again I set the value in mm.

Post Code posts the gcode in a file which is later imported into Mach3 for machining. Configuring Mach3 for CNC machine is another topic and it is generally provided by the manufacturer. Also they have a good website on this: http://www.machsupport.com/ with support forums. Here is the video: