bot.gbr ------Sample gerber file from Eagle 5.11 using ULP:

PCB-GCODE Version 3.6.0.4

John T. Johnson

pcbgcode@pcbgcode.org

January 16, 2013

CncG.cpp -----C code for splitting up a gerber file for use with a free version of Mach3

which is limited to 500 lines of input g code.

CncG.exe-----Executable copy of the above code, compiled with Borland C++ 5.5.1

CncG.tds-----Borlands setup file.

Mill1.tap----Sample output file

Mill2.tap----Sample output file

Mill3.tap----Sample output file

ReadMe.txt---This file.

To run this program you will need an input file like bot.ggr

It will ask for the Zetch depth to be reloaded into each extra file to continue

where the last file left off.

It will ask for a Zsafe value which is the Z position where the cutter will be parked

when changing files.

Then it will ask for the full file name.

It then will make as many files as necessary with less then 500 lines

to mill your board.

Please enter the etch depth, -0.007 would be 7 mils etch down.

-0.010

Please enter the Z safe height 0.75 would be 750 mills above your Z 0.0.

0.400

Please enter the input file name.

bot.gbr

These responses would give an

etch depth of 10 mills

safe Z location of 400 mils above the board

input file of bot.gbr

create 3 files

Mill1.tap----Sample output file

Mill2.tap----Sample output file

Mill3.tap----Sample output file

Have fun.