G Code Profile Utility Program Help

Introduction

This help file is intended to explain the behaviour of the Chestnut Pens G Code Profile Utility Program.

Contents

Program Overview

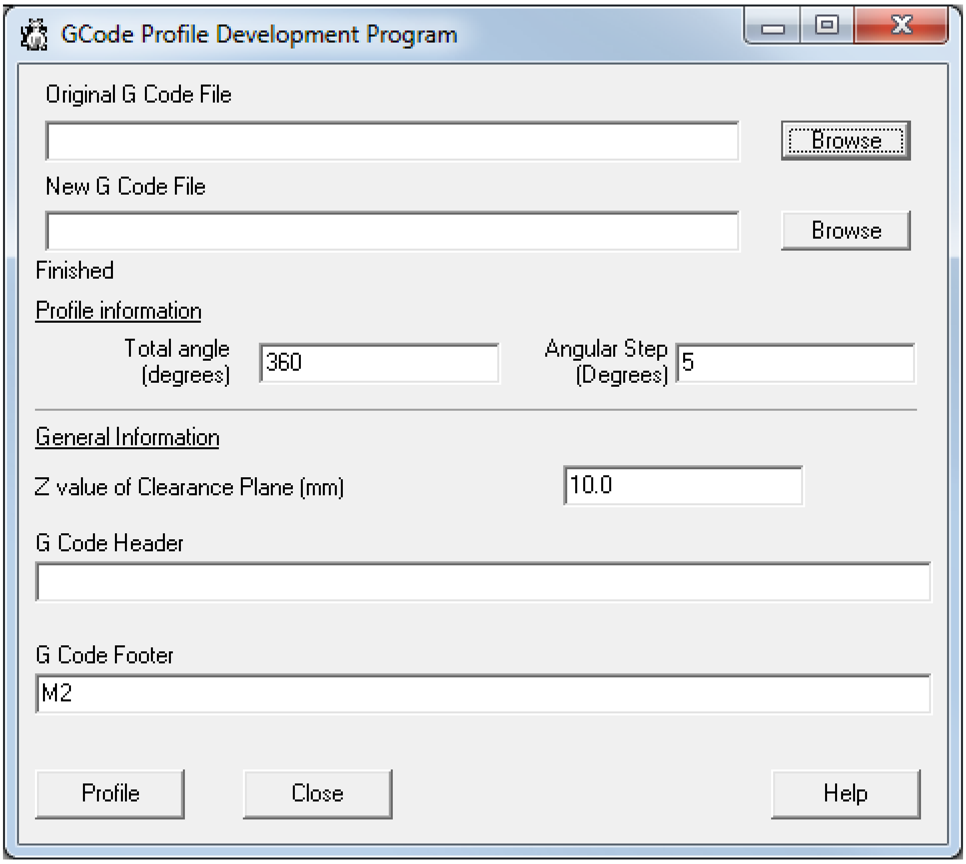
The program does the fairly simple task of opening the input toolpath file, understanding it, rotating the toolpath into the XZ plane, then running the path in forwards & reverse up and down the X axis while rotating the A axis between each pass. Finally a new toolpath file is written to disc.

The program reads in a G code profile on the +ve Y section of the XY plane, understands it and re-creates the output based on its understanding of the input code, acting in forward & reverse through the profile, rotating by the defined step angle at the end of each pass. The profile is a simple line of the toolpath that is required, and must be offset by the radius of the tool used from the work. The program assumes a ball nose cutter is used, and the controlled point is the centre of the ball.

The input profile on the XY plane is flattened so that all input Z commands are ignored, then it's rotated through 90 drgrees onto the XZ plane at Y = 0. All Y co-ordinates are converted to Z co-ordinates and all XY plane arcs are converted to XZ plane arcs to retain the controller's speed & accuracy of arc creation.

How To Use the Program

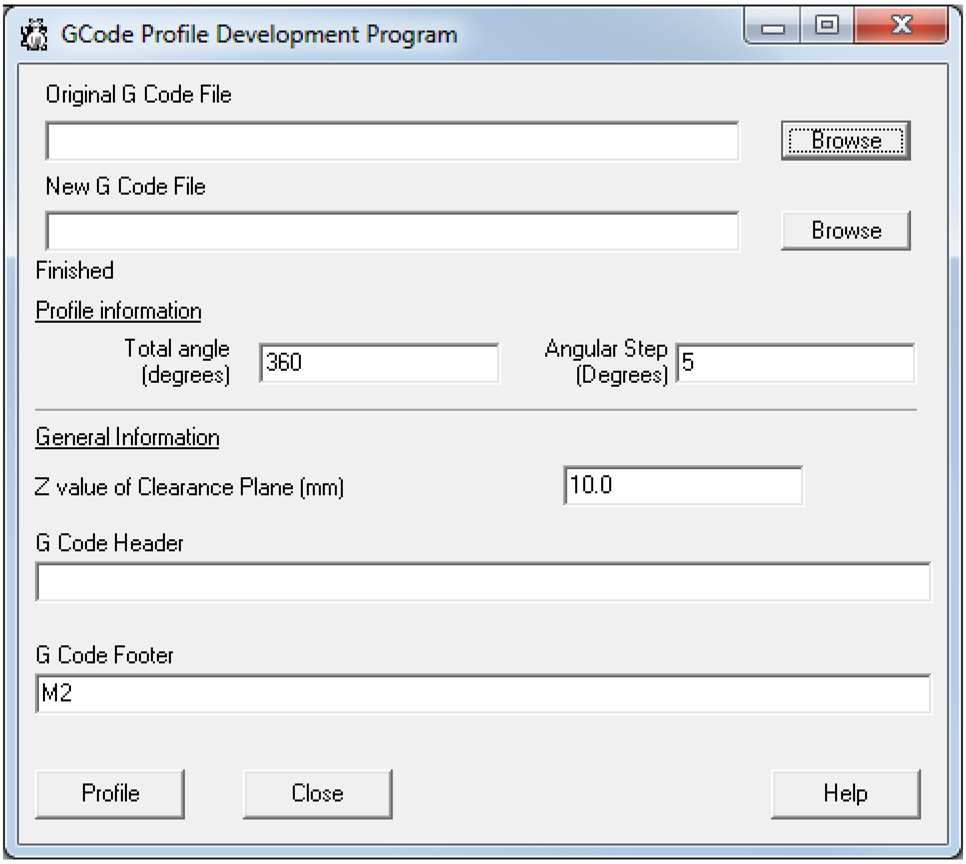
When you open the program you see the following window:



File Names
Use the 'Browse' button to the right of the 'Original G Code File' edit box, and search for the file you wish to convert. When you open a file, a suggested filename is made for the 'New G Code File', which is the old filename with ' Profile' tacked on to the end. If this is default name is not what you want for the output name, please edit it or use the lower 'Browse' button.

Note: The program will not let you overwrite any files when it runs, so please be aware of this.

Profile Information
The next job is to enter the total angle to be machined over, and the angular step between profiles. 360 degrees will provide a whole circle, and a step of 5 degrees will give a moderately good textured finish for an item 12mm diameter while using a 3mm diameter cutter.



General Information.
There are 3 parts to the general information:

Z Value of Clearance plane. At the end of each pass the tool retracts to the clearance plane before the A axis is rotated.

Header and Footer: These are standard headers and footers to be tacked on to the program. The G Code produced by the program is self contained at the header end, but does need an M2 to stop the program.

Tool setup on the CNC:
Zeroing etc. is best explained with an example.
Assume a 10mm work diameter & 6mm diameter ball nose cutter.
Rest the cutter on the top of the work, then the Z value is set as 8mm (radius of work + radius of cutter).

'G Codes' Converted

The 'G Codes' converted are as listed below:

The program is not capable of understanding all the 'G codes', the ones it understands are as below:

(Comment)
F[Feed Rate]
G0 + X, Y, Z qualifiers. A, B & C qualifiers are ignored and trigger an error notice.
G1 + X, Y, Z, F, G53 qualifiers. A, B & C qualifiers are ignored and trigger an error notice.
G2 + I, J, K, X, Y, Z, F, G53 qualifiers. Up to 360 degrees per arc. A, B & C qualifiers are ignored and trigger an error notice.
G3 + I, J, K, X, Y, Z, F, G53 qualifiers. Up to 360 degrees per arc. A, B & C qualifiers are ignored and trigger an error notice.
G4 + P qualifier
G17
G18 Triggers an error notice.
G19 Triggers an error notice.
G20, but converts all the new file to mm. Inches are old fashioned. Stop using them.
G21
G40
G41 Instruction is ignored.
G42 Instruction is ignored.
G43 Instruction is ignored.
G44 Instruction is ignored.
G49
G50
G51 + A, B, C, X, Y, Z qualifiers
G52 + A, B, C, X, Y, Z qualifiers
G54 to G59: Instruction is ignored.
G61
G64 Default Program Setting, if G61 is required, explicitly state it.
G68 Instruction is ignored and triggers an error notice.
G69 Ignored
G70, but converts all the new file to mm. Inches are old fashioned. Stop using them.
G71
G90 & G90.1: All input is converted to absolute positioning.
G91 & G91.1: All input is converted to absolute positioning. P1-C arc centre definition is assumed (Mach3 compatible).
G93 Input ignored.
G94 Input ignored.
G98 + R qualifier
G99 + R qualifier
M0 Input ignored.
M1 Input ignored.
M2 Input ignored.
M3
M4
M5
M6 Input ignored.
M7
M8
M9
M30 Input ignored.
M47 Input ignored.
M48
M49
M60 Input ignored.

N[Line No] Input ignored.
O[Label] Input ignored.
P[Time]
R[Distance]
S[Spindle Speed]
T[Tool No]

The output is specified to 4 decimal places - if that is insufficient, contact me to get the output precision increased.

Limitations

Beyond the limitations already specified, it must be noted that there WILL be a problem if the XZ or YZ planes are specified.

If the input file already contains A, B or C axis movements, there will be an error report and the output file may contain unexpected problems. The A, B or C axis instructions are all ignored.

The Canned cycles (G12, G13, G73, G80-89) are all removed from the program.

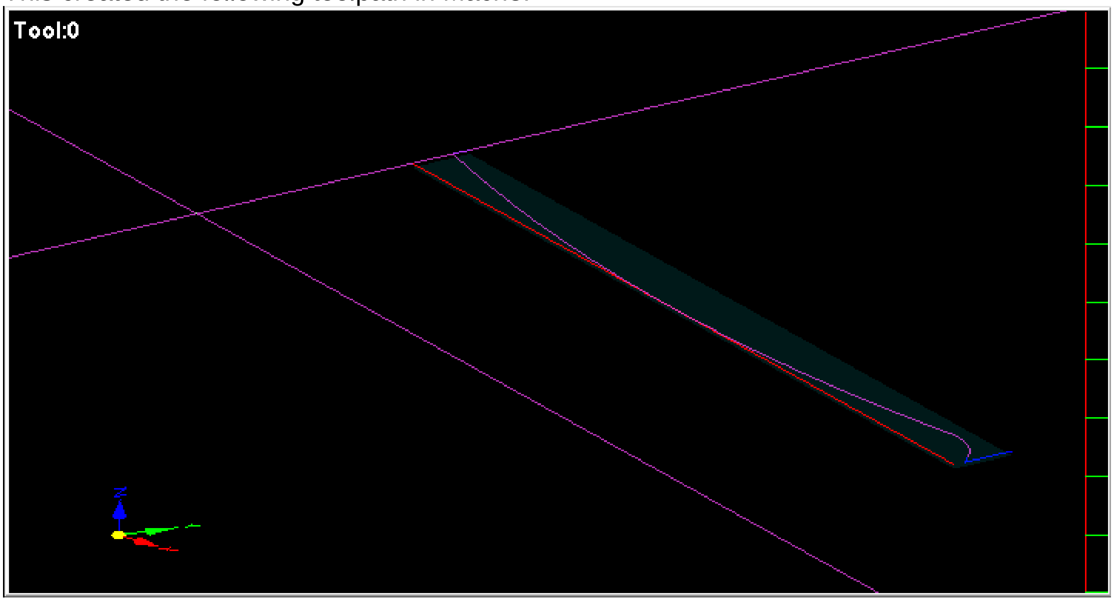
I think I understand the meaning of the G53 qualifier in the G1/G2/G3 codes, but it would be better not to include it if you can, because I may be wrong.

Example

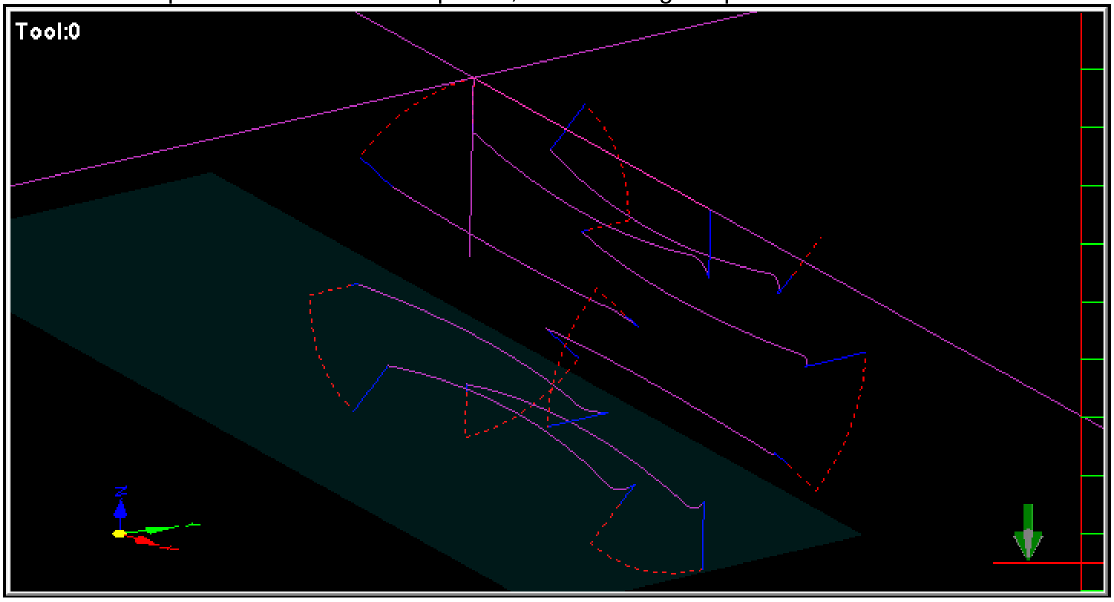
The following G Code was used:

```
(TEST)
F800
G1 X21.5984 Y7.5
G1 Y6.2069
G1 X21.3533 Y6.4704
G3 X19.9509 Y6.9177 I20.2549 J5.4489
G2 X0.0 Y7.0345 I10.2494 J53.7895
G1 Y7.5
M2
```

This created the following toolpath in Mach3:

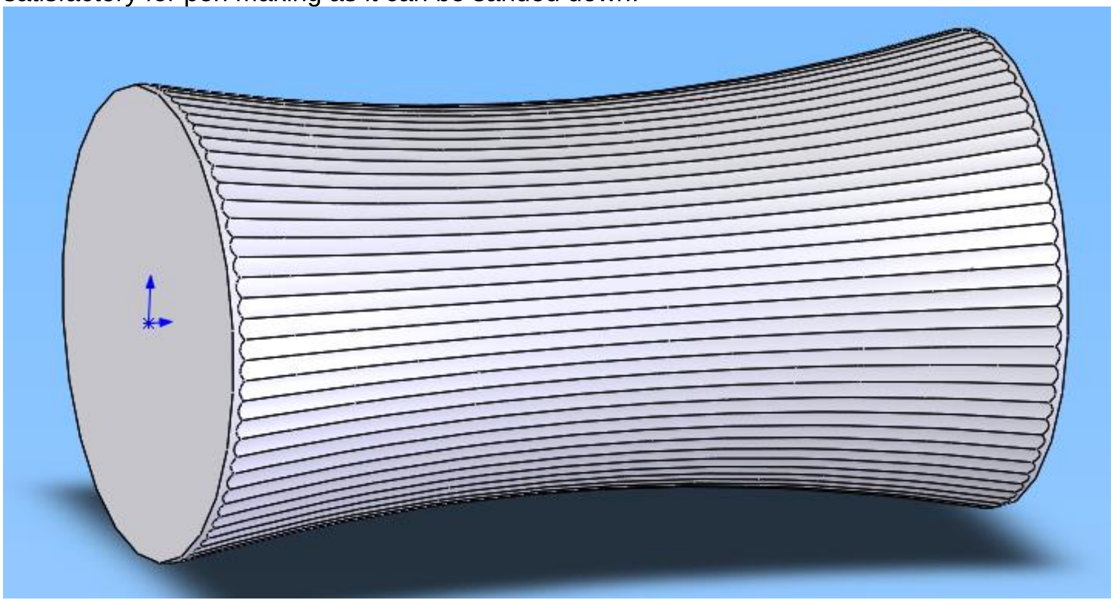


When the toolpath was turned into a profile, the following toolpath resulted:



You will see that all Y values are converted to Z, which is why it is important that the toolpath is followed by a ball nose cutter.

When the above path was done every 5 degrees, the following machined shape resulted (envisaged in SolidWorks). The width of each toolpath witness mark is 0.48mm. This is satisfactory for pen making as it can be sanded down:



Problems/ Suggestions

If you have problems with the program, or suggestions, please send them in an e-mail to:
richardandtracy@yahoo.com
Mark the subject line with something sensible to do with the program, otherwise they'll get deleted unread.