

USER GUIDE

The purpose of this User Guide is to guide the user on the steps to install v5a (or greater) customized post-processor (PostP) supporting Carbide Motion gcode sender for use with Vectric design software executing on a Shapeoko CNC machine.

The v5a PostP for Vectric software adds toolpath and tool# information to the STATUS line display of Carbide Motion similar to some display info added when using Carbide Create. It also adds 'SafeZ Rapid movement avoiding workholding' like Carbide Create adds to gcode. A typical v5a display line would appear as follows:

Toolpath: Profile 1, cutout, Tool# 102

Note: In the example above, the user had entered "Profile 1, cutout" as the toolpath name in the Vectric software.

Normally Vectric default Carbide Motion ATC v4 PostP only displays PostP Revision ID stamp such as: AC06FA06858F6820297A912D3D187A70, which does not mean much to the user.

Additionally, this User Guide also describes how to use the Gcode Pass Parser (GPP) program which will add tool pass information to the gcode file generated by the PostP v5a. A typical GPP produced display line would appear as:

Toolpath: Profile 1, cutout, Tool# 102, Pass 1 of 5 (or)

Toolpath: Profile 1, cutout, Tool# 102, Vector 1 of 4 Pass 1 of 5

The pass information however is only applicable to Profile and Pocket toolpaths defined in Vectric software that allows the user to define/set passes.

The GPP program is a Microsoft Windows v10 .exe executable software program. It and the v5a (or greater) Carbide Motion ATC Post-Processor for Vectric were developed and tested with Vectric Vcarve Desktop and Pro versions 11 and 12 with a Shapeoko Pro CNC, and Carbide Motion v635.

Therefore this User Guide is divided into two sections: **Section 1 Post-Processor Install** and **Section 2 GPP Operation**.

SECTION 1 POST-PROCESSOR INSTALL

You should have received the ZIP file containing the post-processors and this User Guide or you would not be here.

Step 1. Place the extracted files into a dedicated folder on your harddrive of your choice.

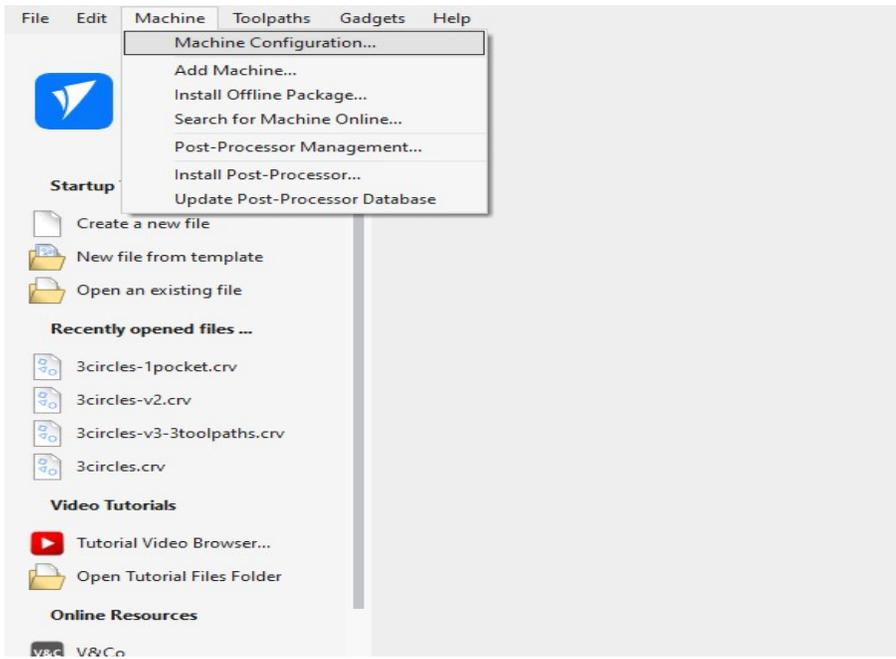
Step 2. Load your appropriate Vectric design software (Vcarve Desktop, Vcarve Pro, Aspire, etc).

Step 3. There are two options to install the post-processors: please first read the two options below and decide which you prefer to perform easiest.

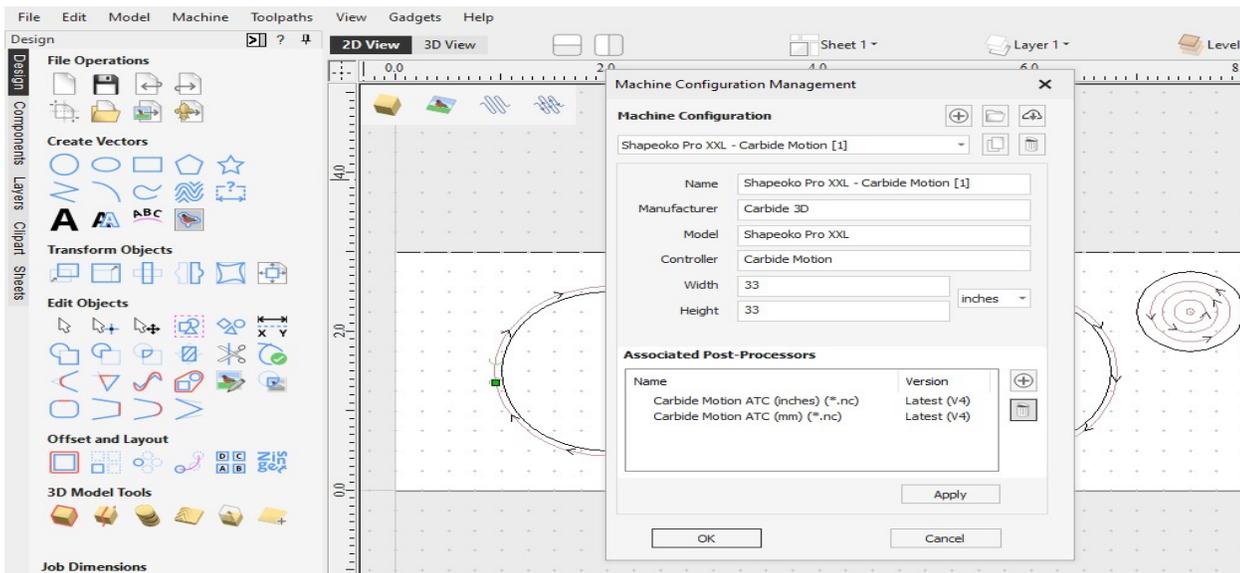
Option 1:

Step 4: Using Windows Explorer or any file management program, move or copy the two post-processor files (the two files ending in .pp) to the Vectric Users Custom Post-Processor folder named My_PostP folder. The folder path is usually as follows:
c:\ProgramData\Vectric\Vcarve Pro\V11.5\My_PostP

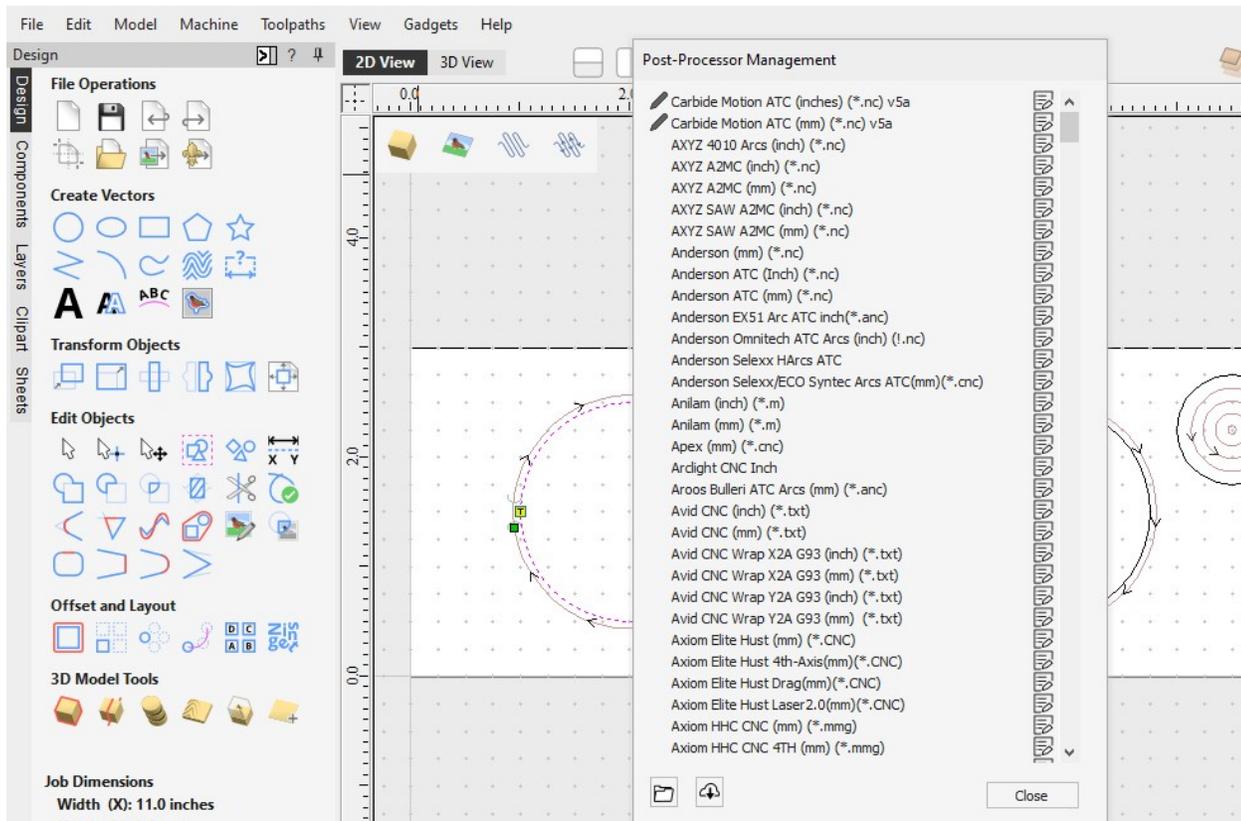
Step 5. In the Vectric software, select Machine, Machine Configuration, as shown below.



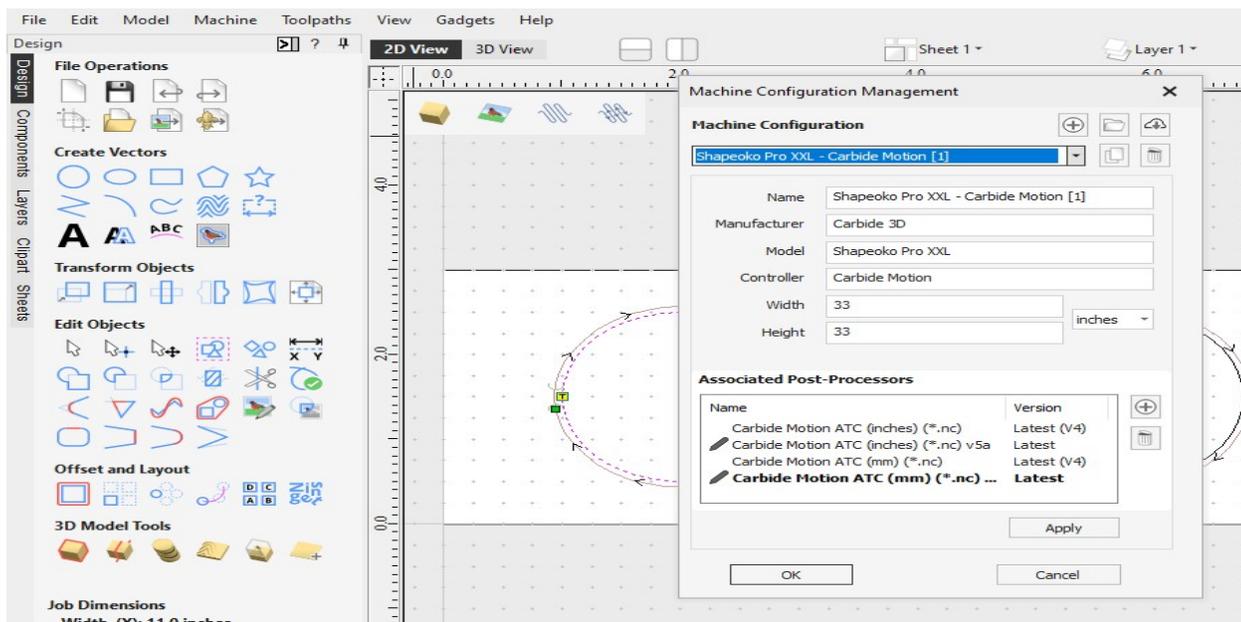
Step 6. The following below dialog box should appear, then select the + symbol to the right of the displayed associated post-processors.



Step 7. The below dialog box will appear showing the two new post-processors at the top of the listing with pencil icons beside them. Select one or both, a SELECT button will appear, press the button.

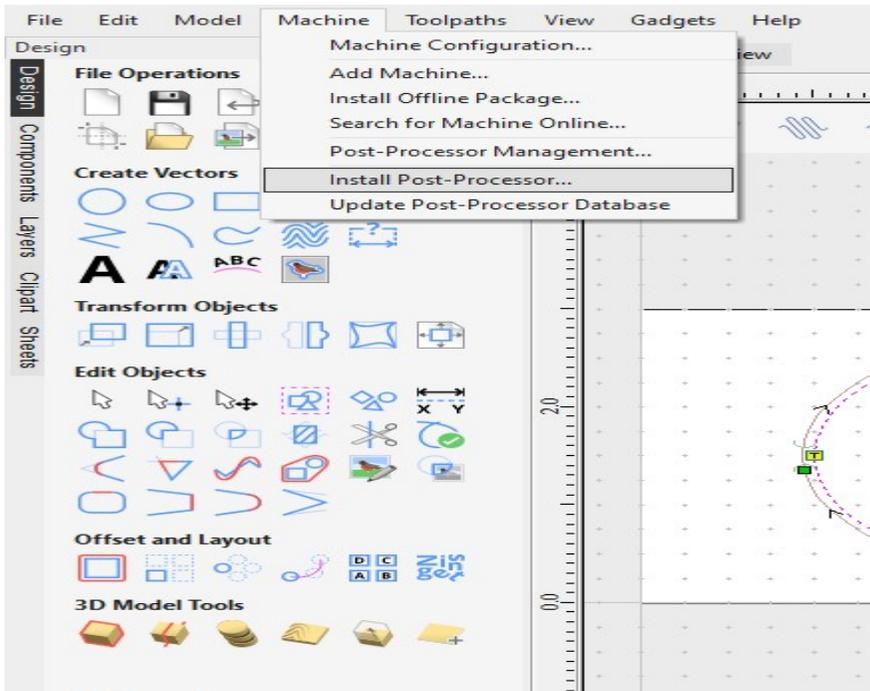


Step 8. The below dialog box will appear, double click the new (v5a) post-processor to be the default and click the APPLY button. Then click the OK button. The dialog will close, your new PostP is installed. Proceed to Step 9.

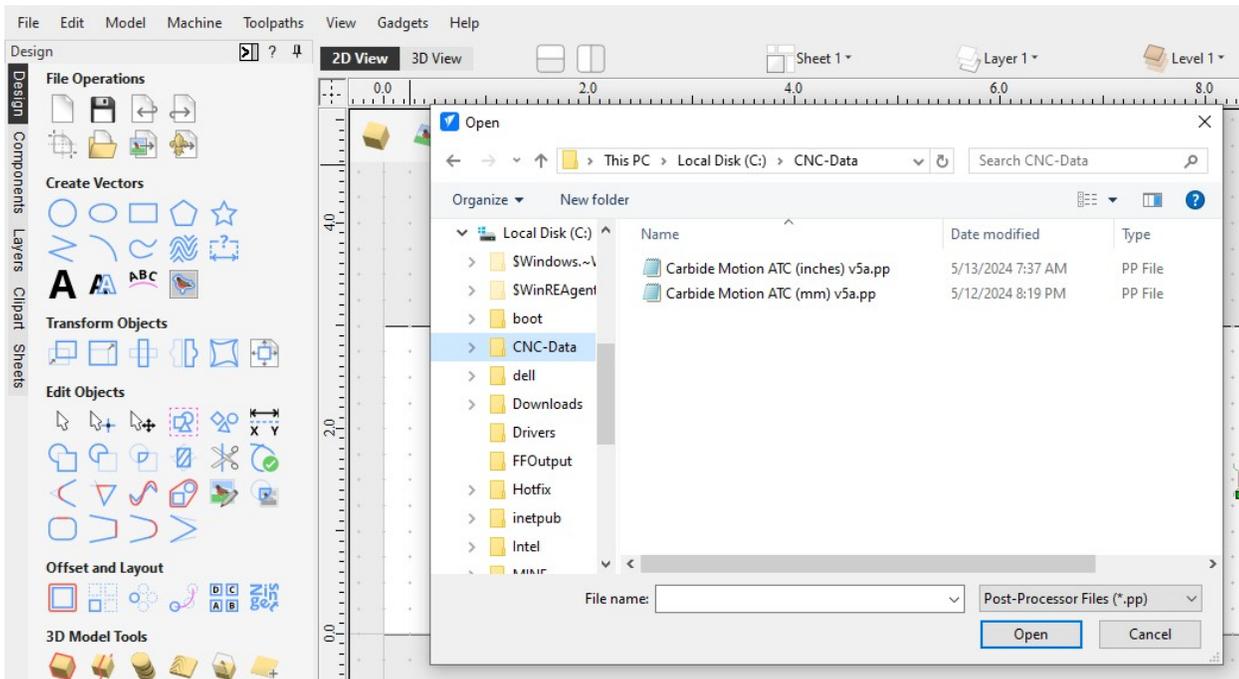


Option 2:

Step 4. In the Vectric software, select Machine, Install Post-Processor, as shown in below figure.

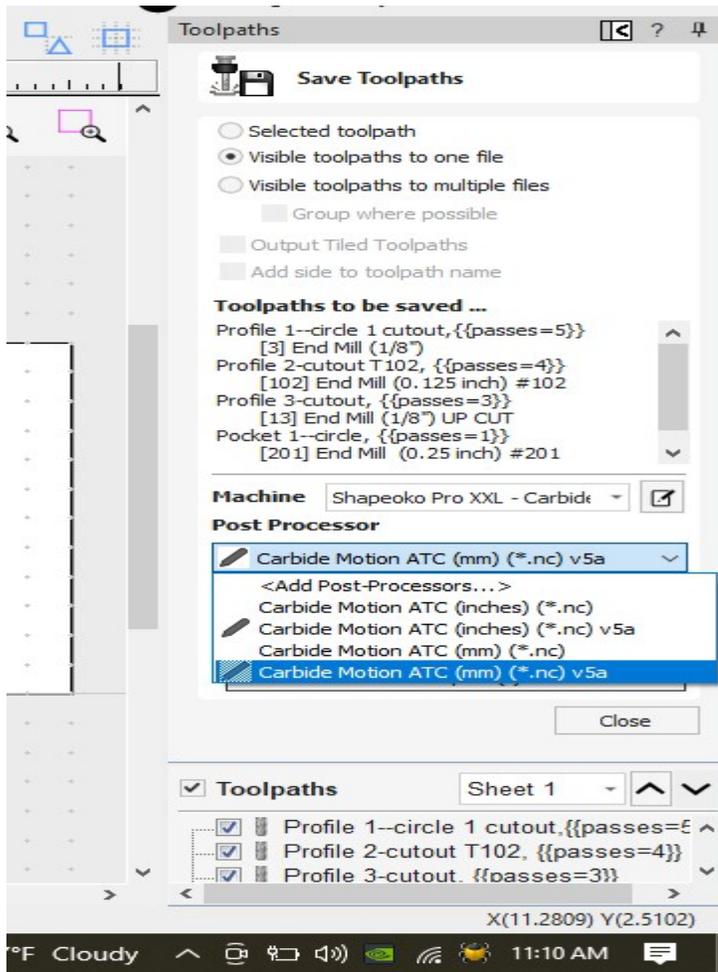


Step 5. The below dialog box will appear, navigate to the folder where you saved the new .pp files and select them, one at a time, selecting OPEN button. Vectric will move a copy to the My_PostP folder automatically for you.



Step 6. Repeat Steps 5 to 8 of Option 1 above. Then proceed to Step 9.

Step 9. Your new post-processors are installed and ready to be selected when you generate a toolpath. See below example. (If not shown, select 'add processors...' from pulldown arrow and select it).



Step 10. This completes the post-processor install SECTION 1.

SECTION 2 GPP OPERATION

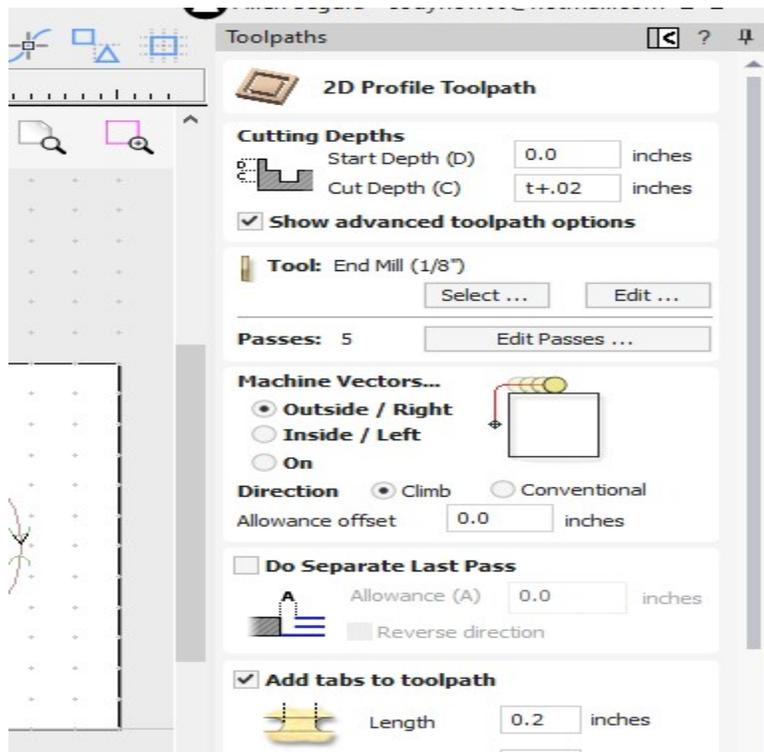
CONDITIONS OF OPERATION:

1. Must have version 5a or greater of my post-processor described above already installed.
2. The GPP program is intended to work with Profile and Pocket toolpaths only which define passes. The word “Profile” or “Pocket” must appear in the toolpath name the user enters in the Vectric toolpath dialog toolpath name field when generating a new toolpath (note: Vectric software defaults to starting the toolpath name with the appropriate word normally).
3. The User (you) must manually append to the toolpath name when entering the name in the toolpath name field the total number of passes which were selected in the toolpath dialog box in the format `{{passes=x}}`, where x is the total number of passes. This is a handshake needed between the two software programs. A typical toolpath name field entry might look like this: Profile 1—circle 1 cutout, `{{passes=5}}`.

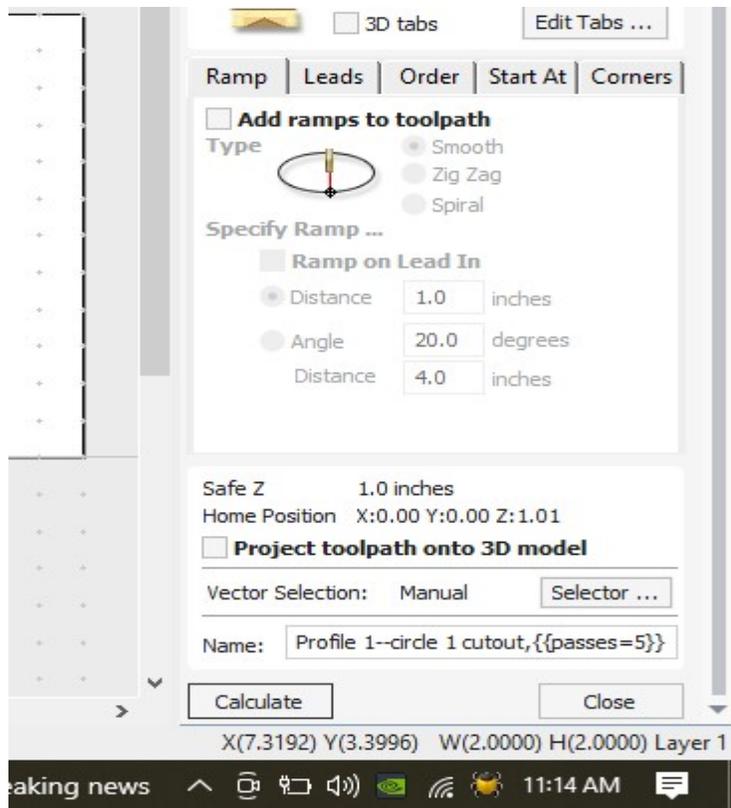
CAUTION

Do not enter symbols () in the filename as Carbide Motion interprets these symbols as commands and will issue line error messages when you try to load the gcode file. V5b of my post processor automatically corrects it for Carbide Motion.

In the example toolpath dialog box below, you can see that the number of passes assigned to the tool is 5.

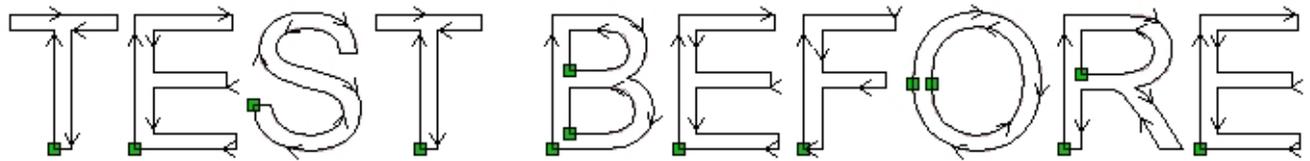


Therefore the appropriate toolpath name field entry would appear as below.



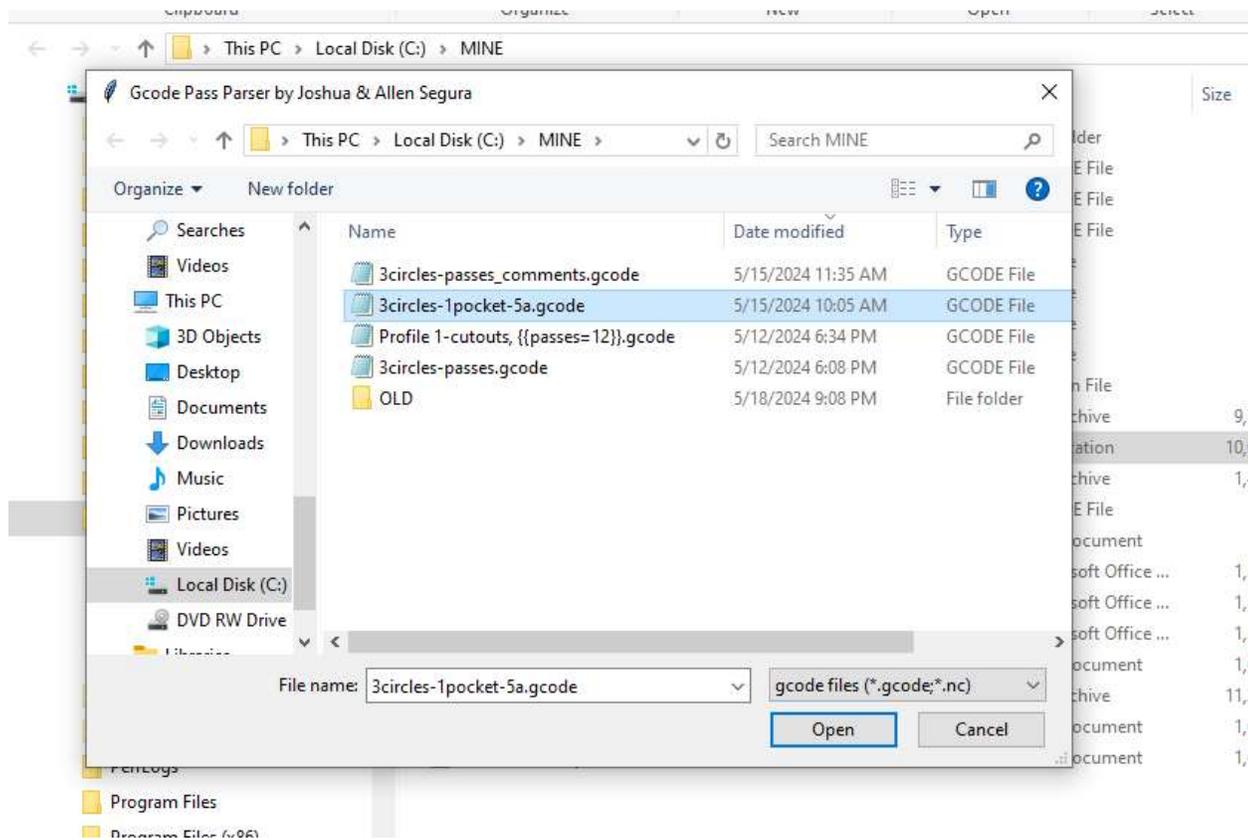
4. For the program to be most successful, all closed vectors should be their own toolpath. For example, if four (4) coaster size circles are being cutout, each circle should be in one toolpath, for a total of four toolpaths, not all four circles in a single toolpath.
5. However, the program does have the ability to handle multiple vectors in a single toolpath, but to do so the user must enter `{{vectors=x}}` just prior to the `{{passes=x}}` entry. So for the 4 coasters example the toolpath name entry would be "Profile 1—circles cutout, `{{vectors=4}}`, `{{passes=5}}`".
6. When entering the full basic toolpath filename, best to keep the basic name to under 50 characters to keep from running off the right edge of the Status line display (excluding the vectors/passes character count)..
7. NOTE: the GPP program currently does not function on toolpaths containing ramps since they are multiple axis moves, not Z axis only, the program will ignore them.
8. The GPP program must reside in the folder with the gcode to be utilized.

9. When doing TEXT as a Profile toolpath, each letter should be considered as a vector. As an example, if the text word was “SOS” then $\{\{vectors=3\}\}$ would be entered. However, some letters require more Z plunges to be cut. So the real vector count would equal the total number of Z plunges to cut the letters. (Note: letter A is generally 2 plunges; letter B three plunges, etc---sounds complicated but simple to estimate thinking about it.) (Another example, if text was “BOO” vector count would be $3+1+1=5$ $\{\{vectors=5\}\}$ would be entered.) The program will find all the letter vectors, Z plunges, so if you do miscalculate and enter too few, say 10 instead of 14, the only issue is at the end of the toolpath the display would end with “Vector 14 of 10” display. **{Another easier method is to use Vectric, use the 2D view with Toolpaths selected and count the small green squares for the Z-plunges; or the 3D view and count the blue lines representing Z-plunges} In the 2D example figure below, the count would be 14.**



OPERATION STEPS:

Step 1. In the gcode folder, double click on the GPP .exe file. Dialog box as shown below will appear. Click on the file to be converted, name appears in the File-name field. Then click the OPEN button.



Step 2. The program executes, saving a new file appended with the word “comments”, and automatically closes, job complete. In the example above (3circles-1pocket-5a_comments.gcode) would be the new gcode file ready to cut. If desired you can open it with a program like windows Notepad to look at the results. (Note: in the screenshot above you can see that the GPP program previously was performed on file 3circles-passes.gcode.). (Note: If a previous ‘comments’ file of same name already exists, the GPP program will overwrite it automatically.)

Enjoy!!

Next page, some example Carbide Motion STATUS Line screenshots.

Run Job

File: 3circles-1pocket-5a_comments.gcode
 Current Line: 13 of 226
 Progress: 5.75%
 Time Remaining: TextLabel
 Status: Toolpath: Profile 1--circle 1 cutout, Tool #3, Pass 1 of 5
 G0Z5.080

FEEDRATE +10%

RESET FEEDRATE

FEEDRATE -10%

Position

X: 0.948
 Y: 1.350
 Z: 0.971
 Vel: 39.4
 Override: 100%
 (Inch)



START

PAUSE

STOP

STOP

Build: 635



10:10 AM
 5/15/2024

Run Job

File: 3circles-1pocket-5a_comments.gcode
 Current Line: 105 of 226
 Progress: 46.46%
 Time Remaining: TextLabel
 Status: Toolpath: Profile 2-cutout T102, Tool #102, Pass 2 of 4
 G2X149.857Y11.382I-26.987J0.000

FEEDRATE +10%

RESET FEEDRATE

FEEDRATE -10%

Position

X: 6.717
 Y: 1.058
 Z: -0.135
 Vel: 16.7
 Override: 100%
 (Inch)



START

PAUSE

STOP

STOP

Build: 635



10:12 AM
 5/15/2024

Typical v5b PostP Gcode File Example

Beginning of file screenshot:

```
michael-dolphins-plaque_v2-oak_comments.gcode - Notepad
File Edit Format View Help
(Material Size: 18.000in-x, 5.500in-y, 0.750in-z)
(VECTRIC POSTP: Carbide Motion ATC inches * .nc v5b)
G20
G90
M6 T28
(Move to safe Z to avoid workholding)
G53G0Z-0.197
(Toolpath: V-Carve 1--dolphins; Tool #28)
G0X0.0000Y0.0000
S10000M3
G0X2.5284Y2.3415
G0Z1.0000
G0Z0.2000
G1Z-0.0561F20.0
G1X2.5272Y2.3522Z-0.0552F40.0
G1X2.5248Y2.3609Z-0.0544
G1X2.5196Y2.3733Z-0.0528
G1X2.5129Y2.3853Z-0.0516
G1X2.4929Y2.4170Z-0.0511
G1X2.4893Y2.4230Z-0.0514
G1X2.4858Y2.4268Z-0.0477
G1X2.4806Y2.4305Z-0.0406
G1X2.4770Y2.4317Z-0.0381
G1X2.4731Y2.4321Z-0.0375
G1X2.4322Y2.4294Z-0.0380
G1X2.4176Y2.4279Z-0.0380
G1X2.4069Y2.4262Z-0.0382
G1X2.3938Y2.4230Z-0.0394
G1X2.3819Y2.4197Z-0.0415
```

End of file screenshot:

```
*michael-dolphins-plaque_v2-oak.gcode - Notepad
File Edit Format View Help
(Toolpath: Toolpaths COMPLETE)
G0Z1.0100
G0X0.0000Y0.0000
(Create Date / Time: Tuesday June 04 2024 / 05:33 PM)
(StockMin: 0.000in-x, 0.000in-y, -0.750in-z)
(StockMax: 18.000in-x, 5.500in-y, 0.000in-z)
(Z-Origin: Material Surface, Safe-Z: 1.000in)
(XY-Origin: Bottom Left Corner)
(X-Origin-Position: 0.000in; Y-Origin-Position: 0.000in)
(Toolpaths Output: V-Carve 1--dolphins)
(Toolpaths Output: V-Carve 2--Michael)
(Toolpaths Output: V-Carve 3--Boone)
(Toolpaths Output: V-Carve 4--629)
(Toolpaths Output: Profile 1-cutout, {{passes=14}})
(Toolpaths Output: Profile 2-roundover, {{passes=1}})
(Tools Used: 28 = V-Bit (30.0° - 1/8"))
(Tools Used: 102 = End Mill (0.125 inch) #102)
(Tools Used: 72 = 1/4 Radius Form Tool (27/40"))
(File Notes: .Sub dolphins for customer Michael Smith)
(File Notes: Invoice #12345)
(Toolpath Notes: Roundover using IDC 1/4" roundover bit)
(Vectric PostP ID: 0F062BE3122283025DC3749C3B1B6FD8)
(Design software: VCarve Pro 12.007)
(Toolpath Filename: michael-dolphins-plaque_v2-oak.gcode)
(File Location: C:\Users\Owner\CNC_Data\PROJECTS-mine\michael-dolphins-plaque_v2-oak.gcode)
(Toolpath: Toolpaths COMPLETE)
M2
|
```