

# **USER GUIDE**

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

Additionally also how to use the Gcode Pass Parser (GPP) which will add pass information to the gcode file generated by the PostP v5a. Therefore the User Guide is divided into two sections: Section 1 Post-Processor Install and Section 2 GPP Operation.

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.

The GPP program is a Microsoft Windows v10 software program. It and the v5a 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.

---

## **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.

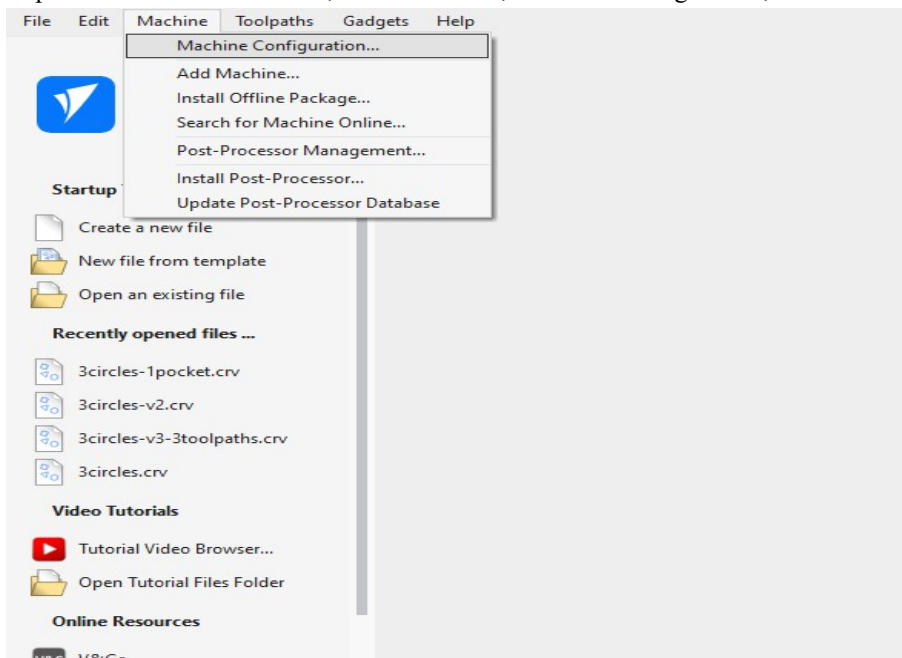
Step 2. Load your appropriate Vectric design software (Vcarve Desktop, Vcarve Pro, Aspire, etc).

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

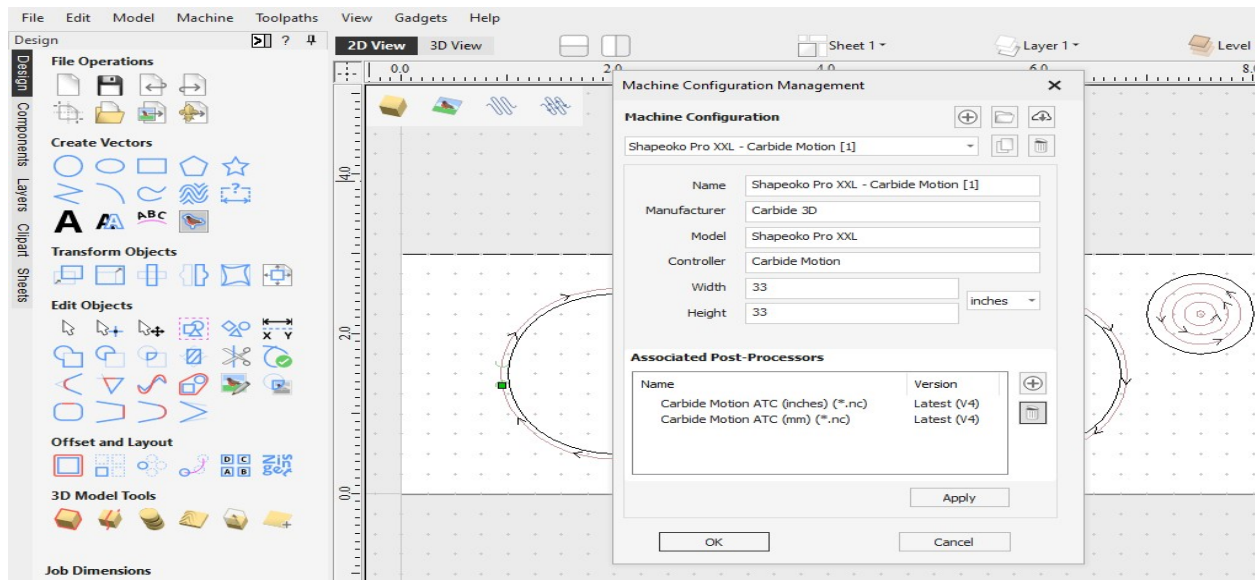
### **Option 1:**

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

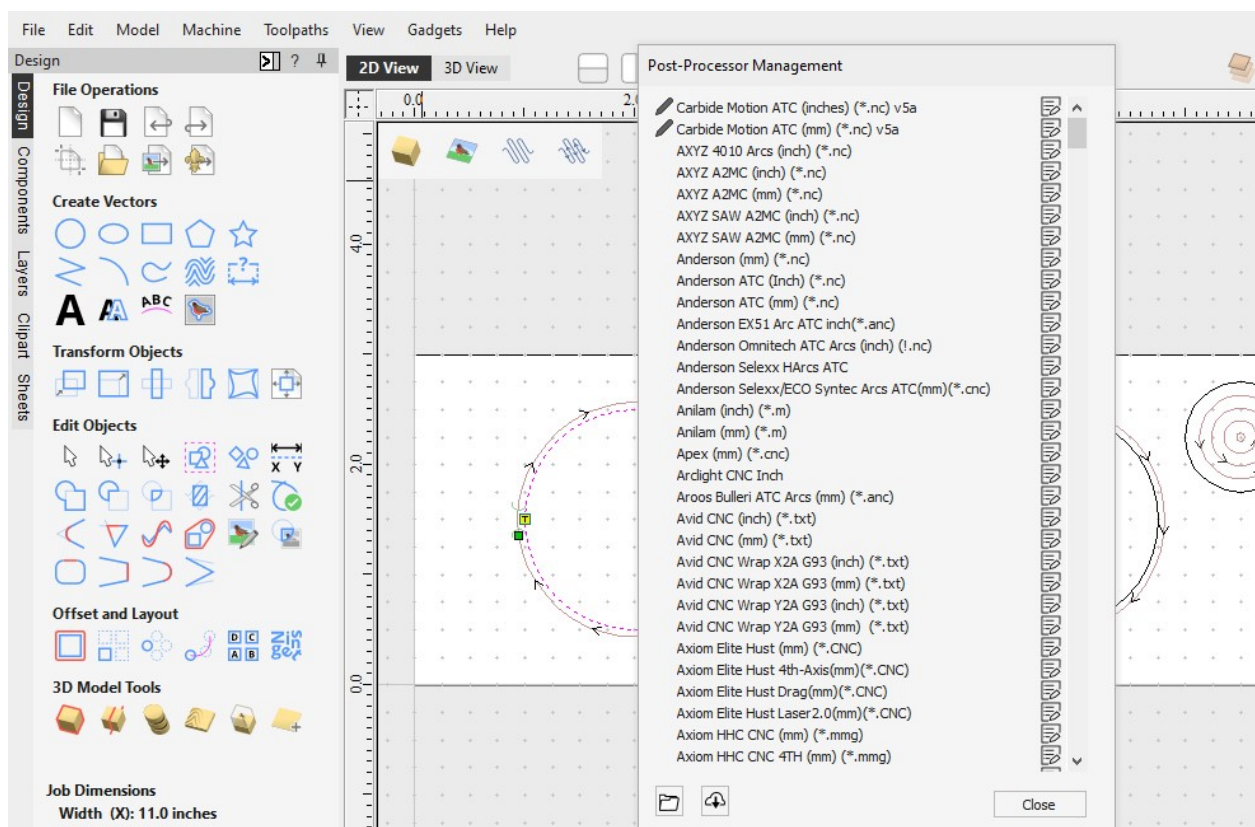
Step 5. In the Vectric software, select Machine, Machine Configuration, as shown below.



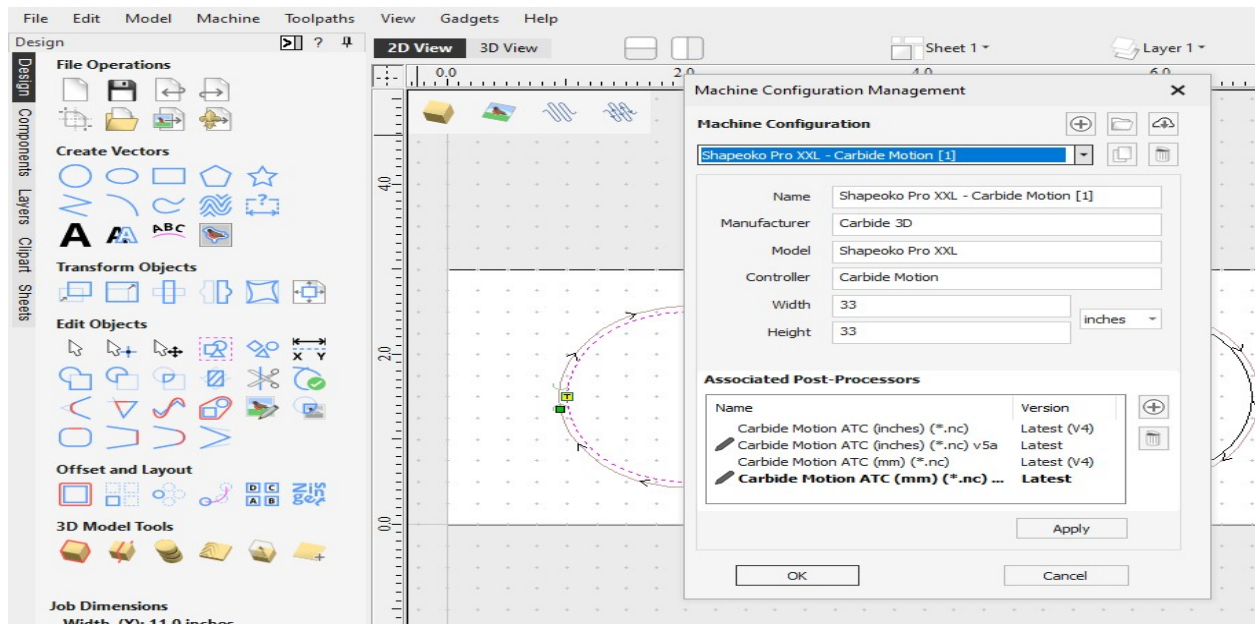
Step 6. The following below dialog box should appear, then select the + symbol to the right of the displayed associated post-processors.



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.

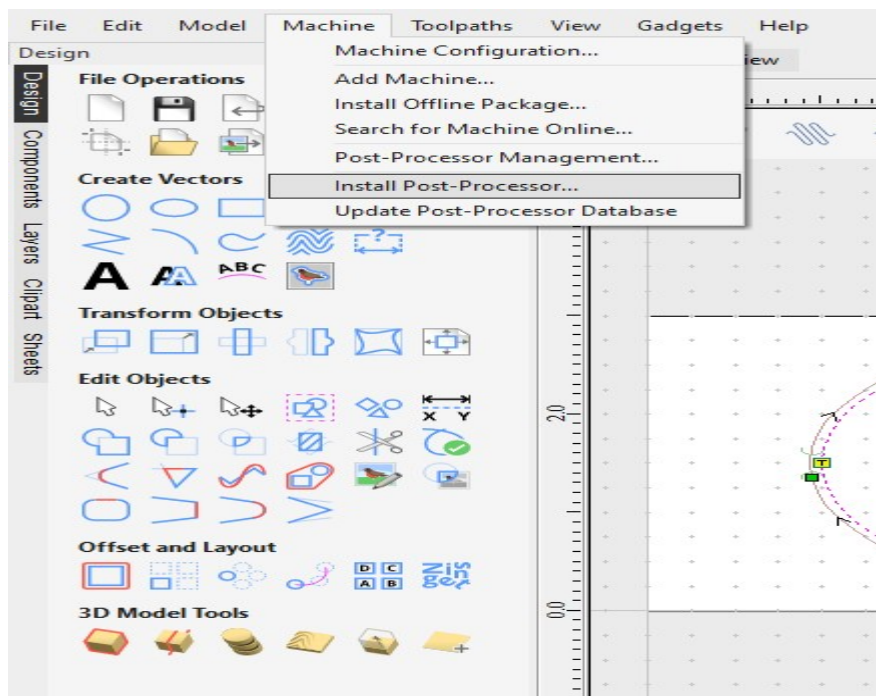


Step 8. The below dialog box will appear, double click the new (v5a) post-processor to be the default and click the APPLY button. Then click the OK button. The dialog will close, your new PostP is installed. Proceed to Step 9.

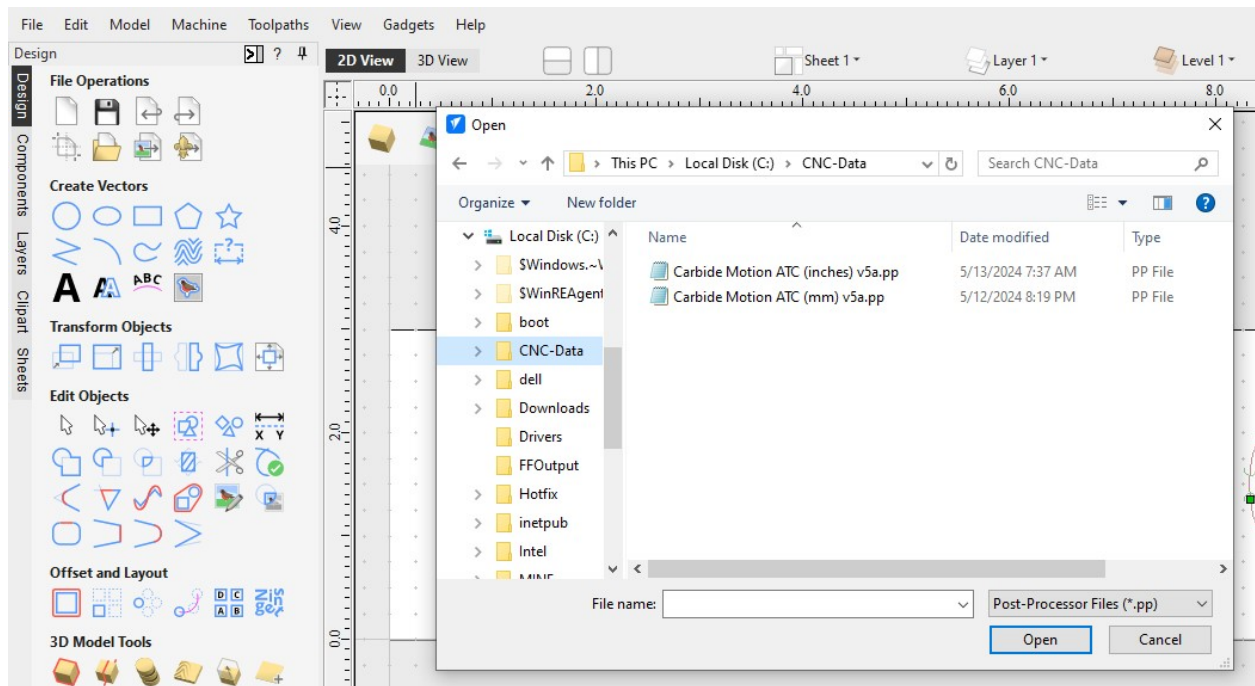


## Option 2:

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

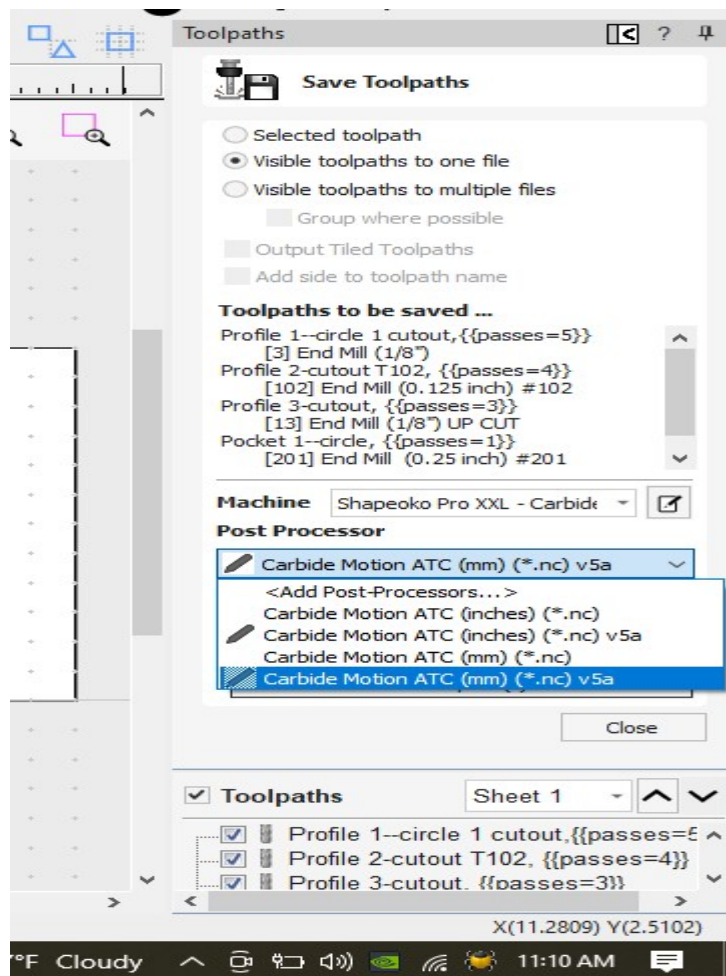


Step 5. The below dialog box will appear, navigate to the folder where you saved the new .pp files and select them, one at a time, selecting OPEN button. Vectric will move a copy to the My\_PostP folder automatically for you.



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 and select it).



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

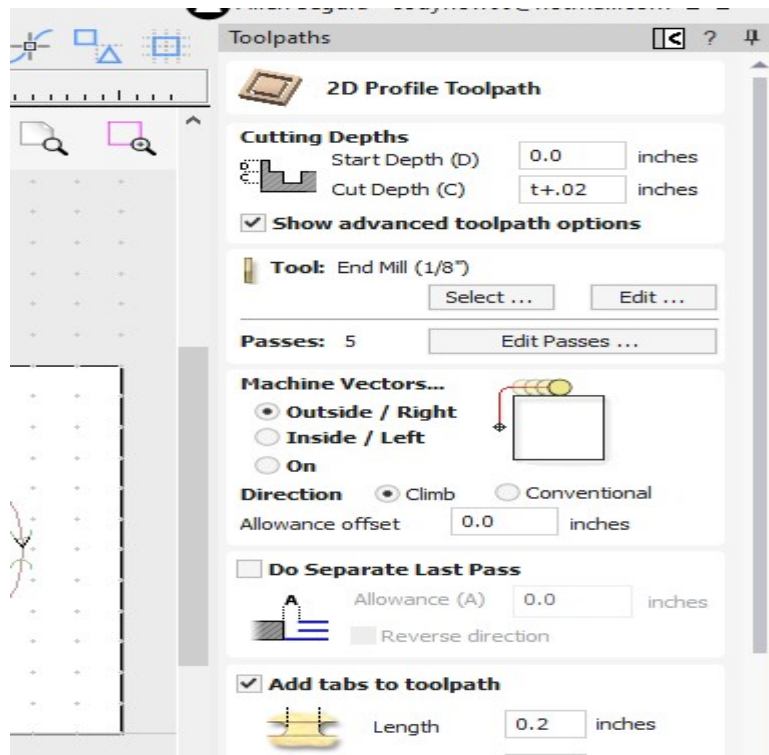


## **SECTION 2 GPP OPERATION**

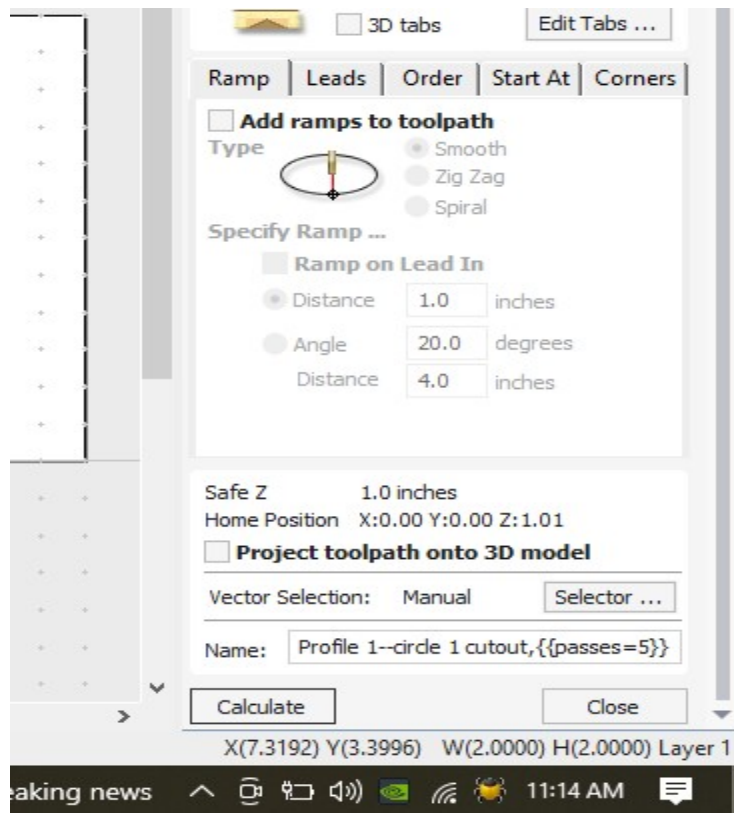
### CONDITIONS OF OPERATION:

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. The words “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 to avoid a lot more program code. A typical toolpath name field entry might look like this: Profile 1—circle 1 cutout, `{{passes=5}}`.

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



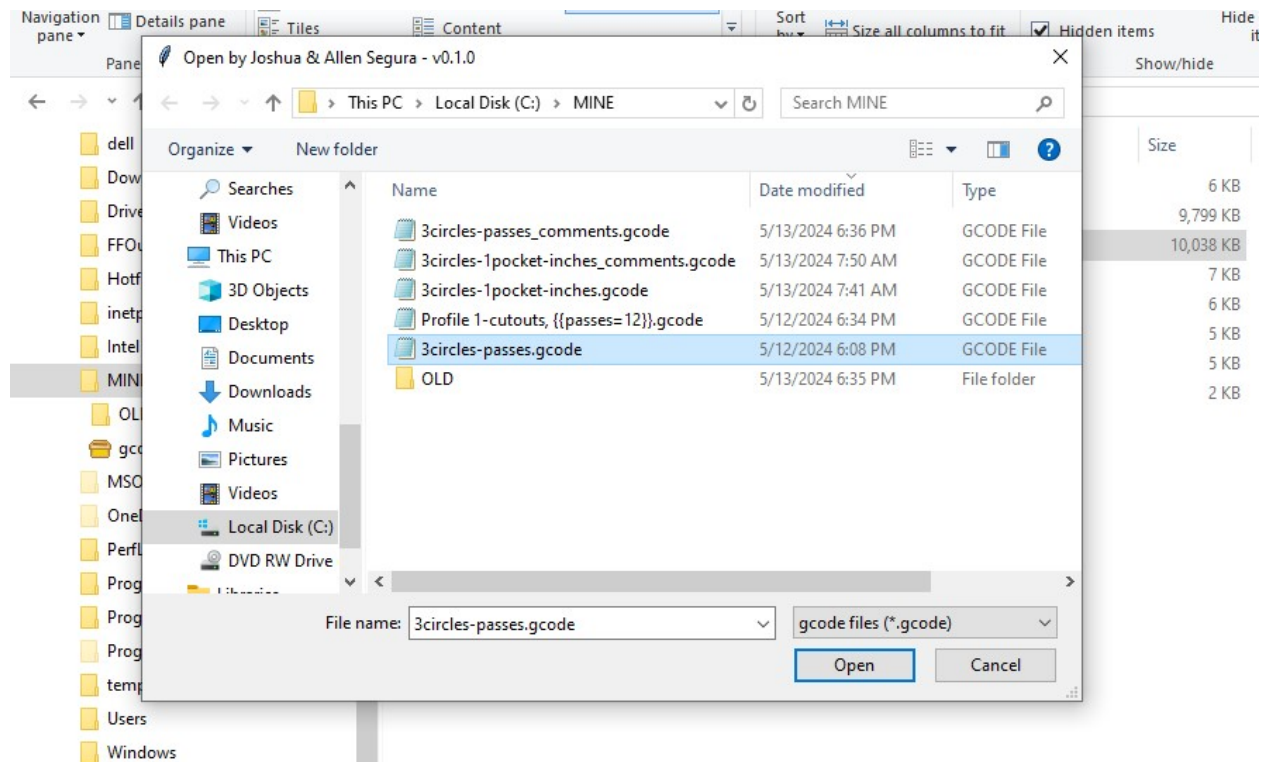
Therefore the appropriate toolpath name field entry would appear as below.



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. (maybe in version 2 we will fix that, but it takes a lot more coding effort so we need to see the interest for it)
5. The GPP program must reside in the folder with the gcode to be utilized.

## 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-passes\_comments.gcode) is the new gcode file ready to cut. If desired you can open it with a program like windows Notepad to look at the results.

Enjoy!!

**Next page, some example Carbide Motion screenshots.**



