



Nameplate Activity

CNC | Conversational Machining

NAME: _____

Date: _____

Section: _____

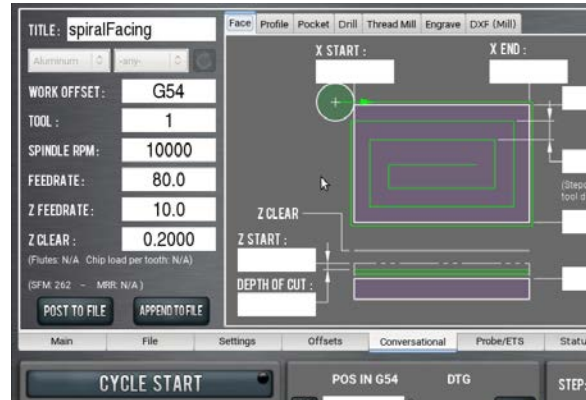
Introduction

Sometimes you just want to complete a simple task on a CNC machine, like doing any one of the following:

- Drill a couple of holes
- Make a simple pocket
- Engrave some text
- Make a profile around a boss

All these things can be accomplished quickly and easily, without having to model the part in 3D, or write G & M code by hand. The process by which this is done is called “conversational machining”. With a Tormach CNC machine and PathPilot software this is very easy to accomplish.

In this project you will create a nameplate to learn how to use conversational milling techniques to create a part.



Key Vocabulary/Concepts

- Work with Cartesian Coordinates to place geometry correctly on a part to be machined.
- Engrave text on your nameplate. This can also be called a profile, as you are just following a line with a tool.
- Create holes in your name plate to be able to hang it on a plaque or your desk.
- Create a profile to put a box around your text on your nameplate.
- Engrave
- Profile
- Contour
- Pocket
- Text
- PRZ (Part Reference Zero)

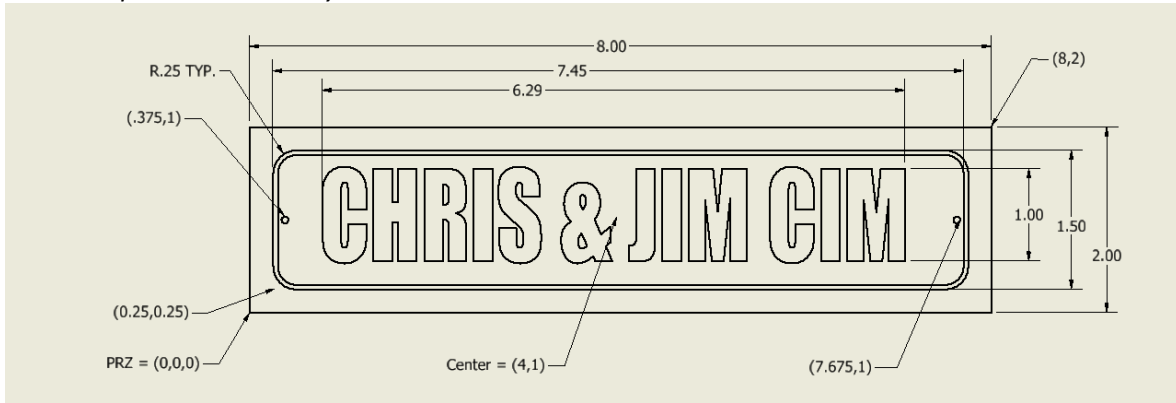
Equipment & Supplies

- CNC Machine
- Material for your nameplate
- Engraving tool
- Software that is Conversational capable such as PathPilot by Tormach

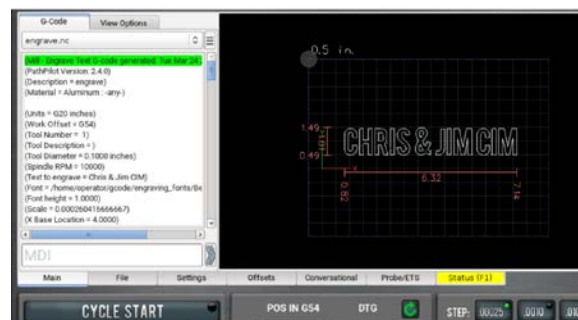


Procedure

Our example with necessary measurements:



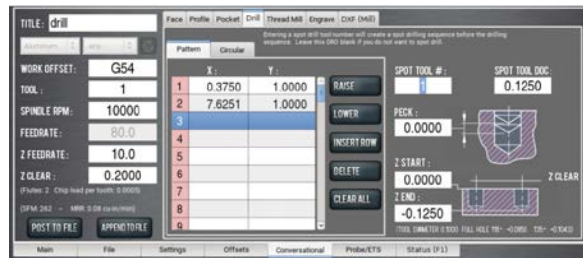
- Decide what text you are going to engrave and choose a piece of material the correct size. Be sure that you figure out the following information before you start.
 - Where PRZ is located (*use lower left*)
 - Height, width and center point of stock
 - Size of the stock
 - Height and width of text
 - Font of text
 - X,Y value of the two holes
 - The size and start point of the outline profile, and radius of the corners
- Layout your sign on a piece of graph paper. A good rule of thumb is to make your text 1" tall. This way the letters are all about 1/2" wide, and about five letters will fit on a 4" piece of stock. We used Bebas font in the example above.
- Be sure to set up your machine with the proper offset, tooling, and material. See other videos and materials to be able to do this correctly.
- Use PathPilot's conversational tab to choose "Engrave". Be sure to set the Z DEPTH OF CUT and Z START at 0. Choose CENTER and enter the center X value. Be sure to choose the right HEIGHT of text and the start point of Y (Y BASE) as well.
 - Once this information is entered, choose POST TO FILE and save the file in a folder of your choice.
 - If done correctly, it will open the *main* tab ready to machine the part. If not, the *status* tab will open and tell you what the error is. Correct it until your part is ready to go.
 - Do NOT machine yet!



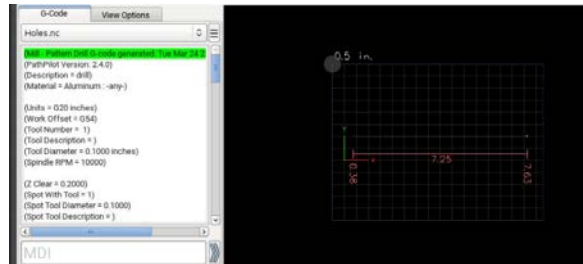
Text is all set, and code has been posted. Note the X & Y values of the text.



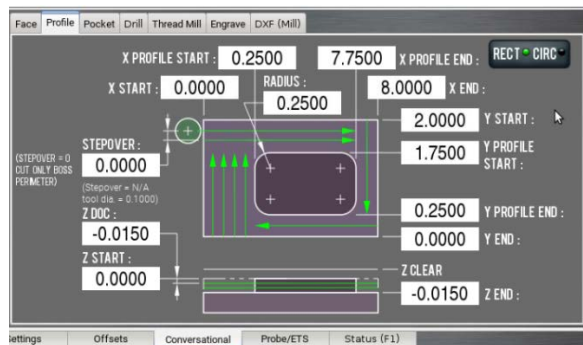
- Next, go back to the *Conversational* tab and choose *Drill*. Enter the X and Y coordinates for the two holes, tell it the *TOOL #*, and *TOOL DOC* (Depth of Cut). *Z START* should be 0, and *Z END* should be a little larger than the thickness of your stock.



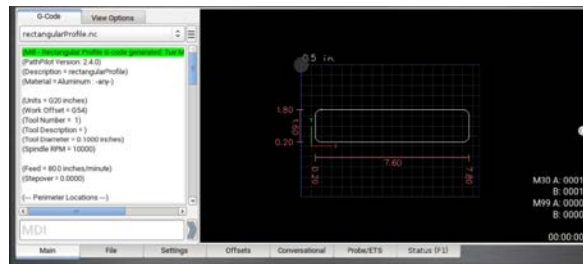
- Once this information is entered, choose *POST TO FILE* and save the file in a folder of your choice.
 - If done correctly, it will open the *main* tab ready to machine the part. If not, the *status* tab will open and tell you what the error is. Correct it until your part is ready to go.
 - Do NOT machine yet!



- Next, go back to the *Conversational* tab and choose *Profile*. Be sure to enter all the shown information from the sketch that you made earlier.
- Be cause we only want to trace the profile, make sure *STEPOVER* is 0, and be sure to get the X and Y start and end values correctly.

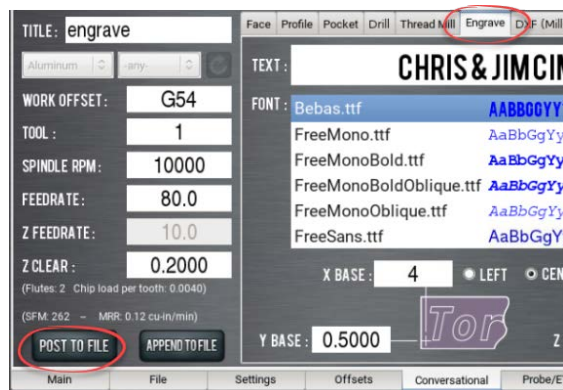


- Once this information is entered, choose *POST TO FILE* and save the file in the same folder as the other two files
 - If done correctly, it will open the *main* tab ready to machine the part. If not, the *status* tab will open and tell you what the error is. Correct it until your part is ready to go.
 - Do NOT machine yet!

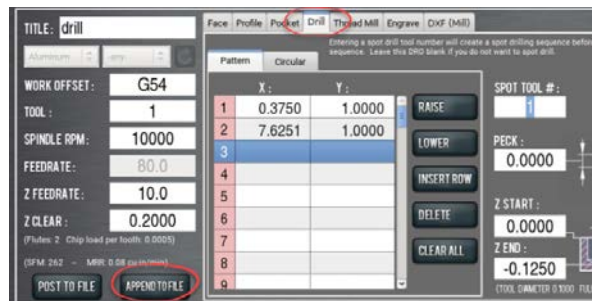


Be sure to see if the X & Y values of the rectangle are correct before you procede.

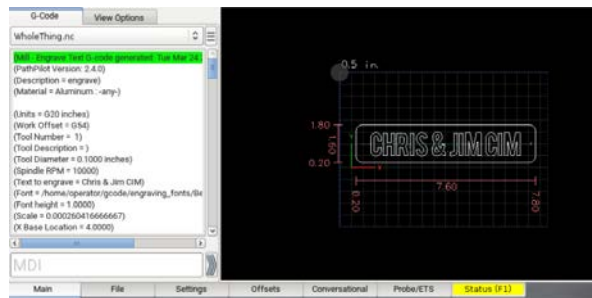
- Once all three files are saved in a folder, it is time to put them all together. To do this, start with the *Engrave* tab, where the text information is.
- Choose Post to File and save it with the name "WholeThing"



- Now choose the Drill tab and click on *Append to File* and be sure to choose the file named "WholeThing". This will add the Drill cycle code to the file after the Text code.



- Do the same thing with the Profile Tab. And *Append To File* as well. This adds all three pieces of code together.
- After the last append, you should see all three operations in the window.



- Once you check your machine set up, and you have your stock held securely in the machine, you are ready to run your code. Check with your instructor before running the machine.



-

Conclusion

- What was the hardest part of this project for you? Please explain in a few sentences.
- There are many G & M codes in your program that you may not have seen before. What do the following NC codes mean? G17? G54? G30?

Going Beyond

- Edit the NC code to make the machine do all three parts of the code without the pause in between operations. Be sure to check with your instructor before you run the machine.

