

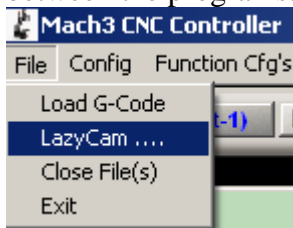
Mill – Contour / pocket a profile in a spiral fashion with CW / CCW motion, on the left or right side of the profile based on a defined tool and it's associated parameters.

- Pocket a profile in a ZigZag fashion based on a defined tool and it's associated parameters.
- Mill a profile with or without offsets based on a defined tool and it's associated parameters.
- Provide gcode for the drilling of holes into a piece and when used along with one of the Wizards will allow for canned drilling cycles.

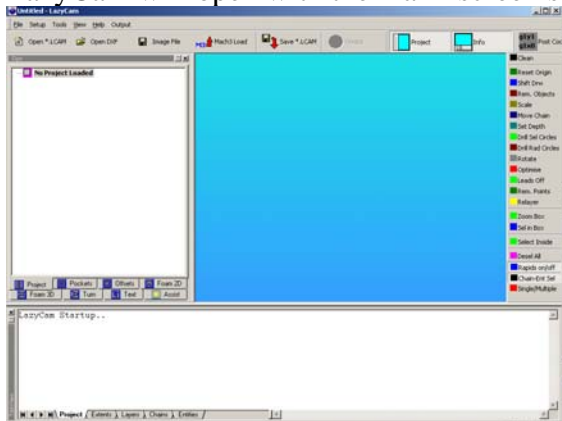
LATHE (TURN)– Provides for the creation of lathe pathing to machine a profile or generate a profile cut as defined by the tool parameters.

FOAM 2D AND 3D - ? I don't know / never used it.

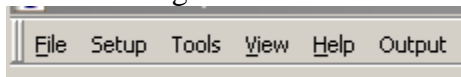
You could also create a short cut and work standalone but recommend that you invoke it from MACH since you can post directly to MACH and may want to go back and forth between the programs.



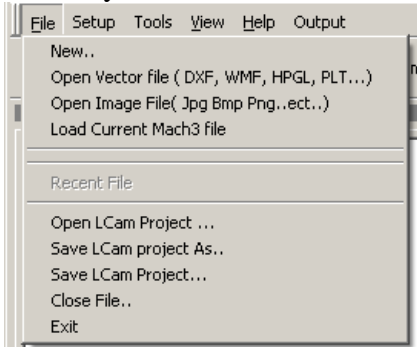
LazyCam will open with the main screen shown as below:



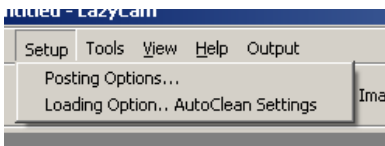
The following tabs are available at the top of the screen.



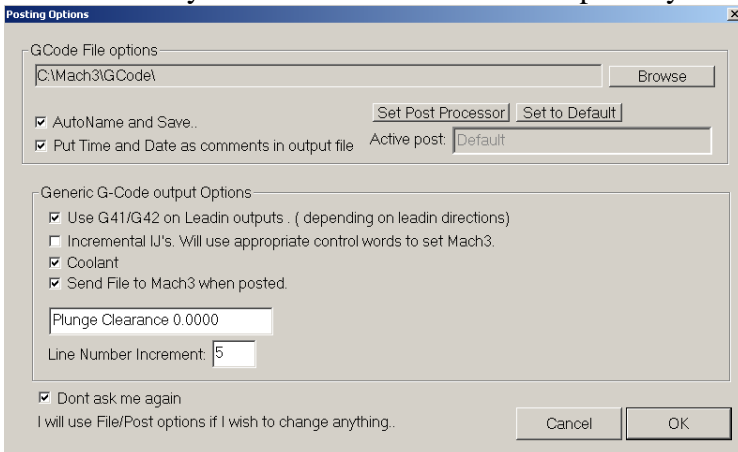
The File button provides for importing different file types, loading a gcode file which is currently in Mach3. You can also save a LazyCam project, close a file or exit LazyCam.



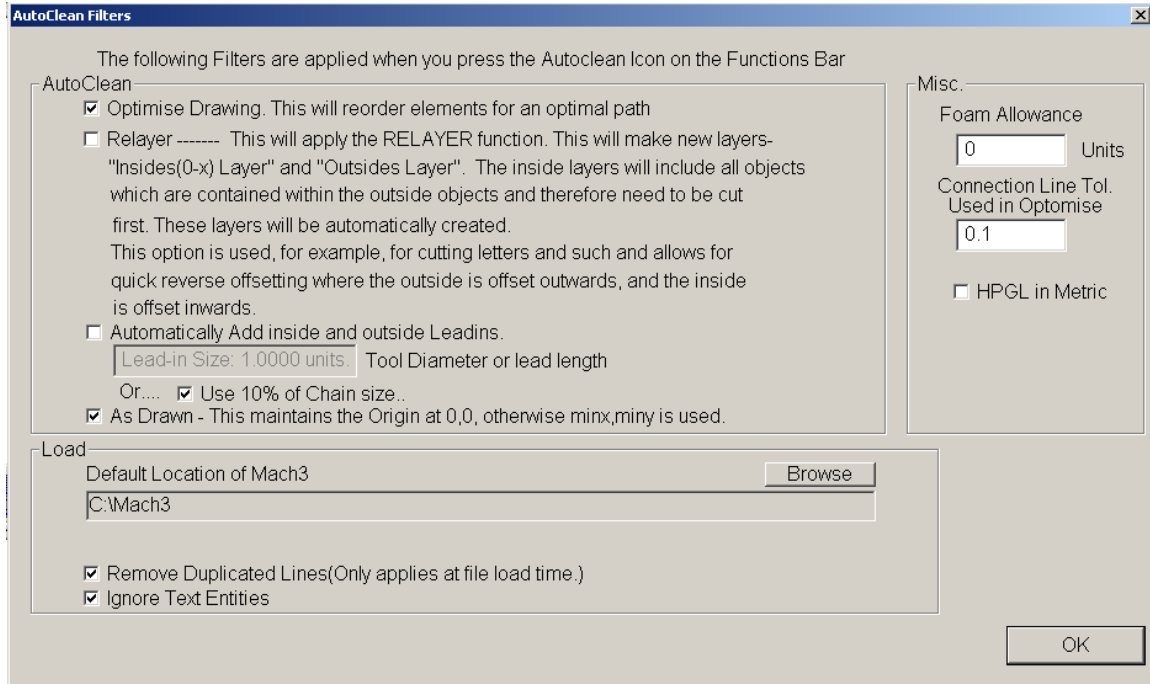
LazyCam provides for setting up posting and loading options via the Setup tab.



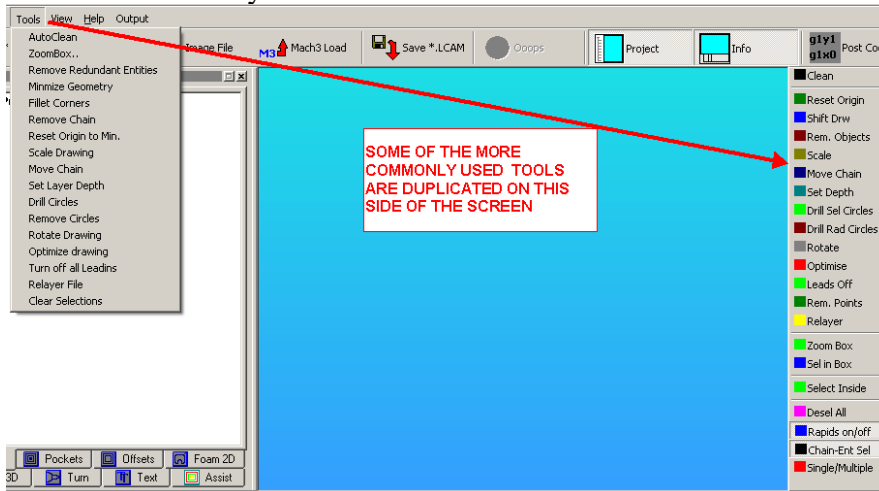
The posting options flyout provides for a number of options. You can have LC (LazyCam) autaname your file, put additional information in the beginning of the gcode to be posted, provide for coolant code, send the file to Mach, assign line numbers in increments of your choice. Just check what options you want and click OK.



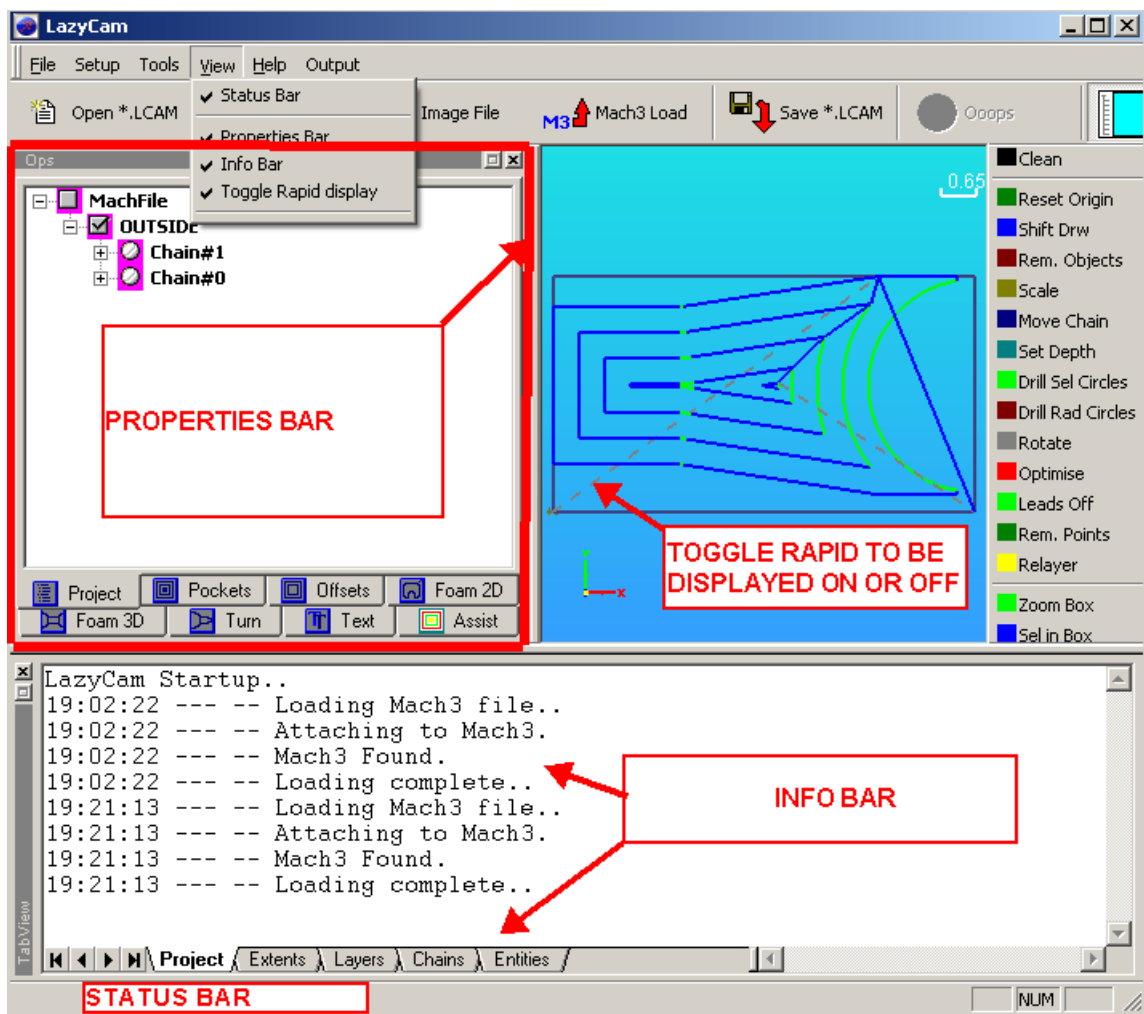
The Loading options flyout provides for some automation as it relates to the file you are importing into LC. I would suggest using a Version 12 DXF CAD generated file of your profile.



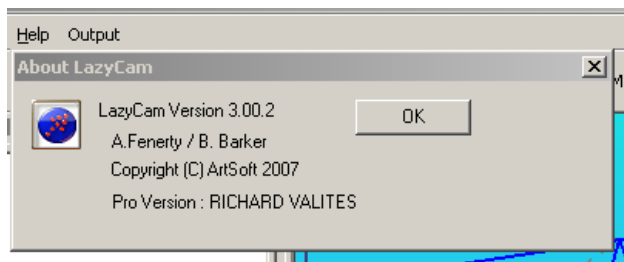
The Tools Tab allows for a number of different options to modify the profile and some of the more commonly used ones are also located on the left side of the screen.



The View Tab allows for adding or removing displayed information as highlighted below.



There is no LC help other than the thread on the web site, and what is written here, and the videos. Clicking the Help button will show you what version of LC you are using.

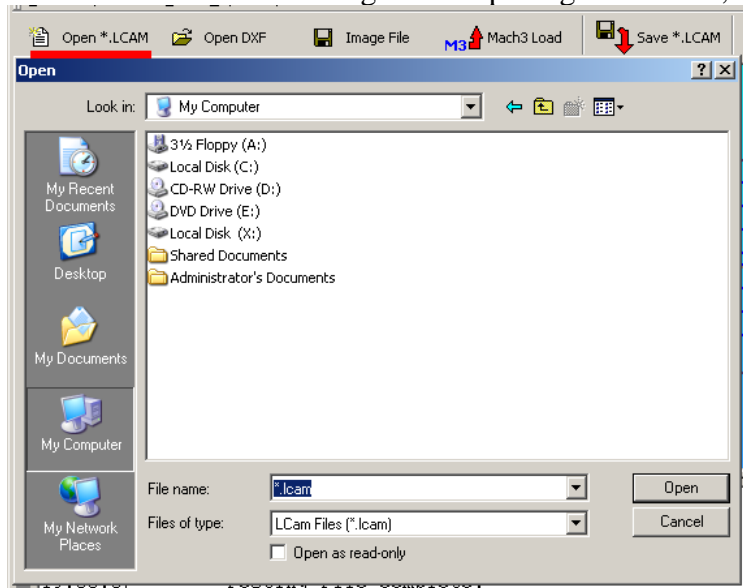


The Output tab allows for posting gcode to MACH3. Note that how and what will be posted is influenced by what you checked off in the Setup Tab.

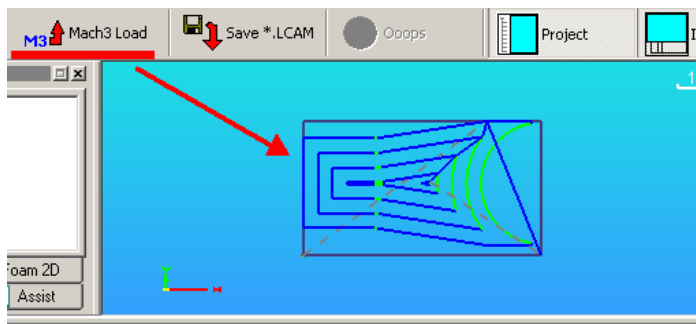


There are a number of commonly used functions which are available via buttons and duplicate some of the Tab pull down commands.

Clicking open *.LCAM will invoke the typical windows screen for locating where you saved the file and the same goes for opening a DXF file, image file.



Clicking the Mach3 Load button will load the gcode which is currently in MACH3 if one was loaded.



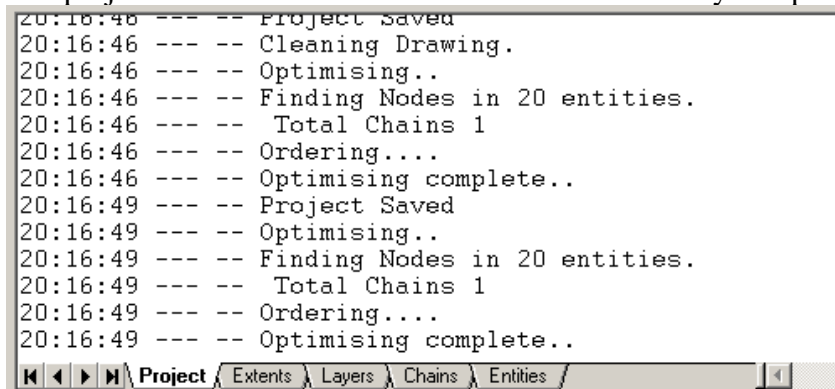
Clicking the Save *.LCAM button will save you current project as it currently is and can be reloaded using the other button or File tab.

The Oops button will undo you last command, the Project and Info Button turn the associated displayed information on or off.

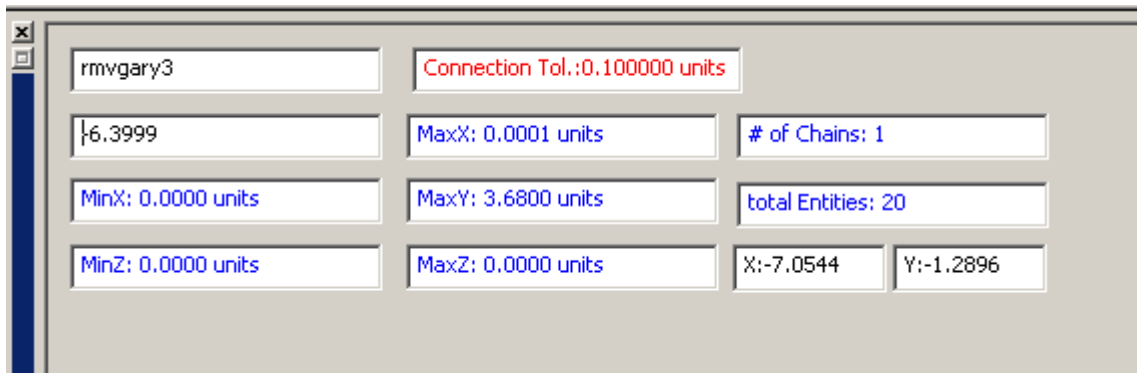


The Info box on the screen provides detailed information relative to your work and is broken down by category via 5 tabs. Clicking on the table will show / provide additional dialogue for relative information or input.

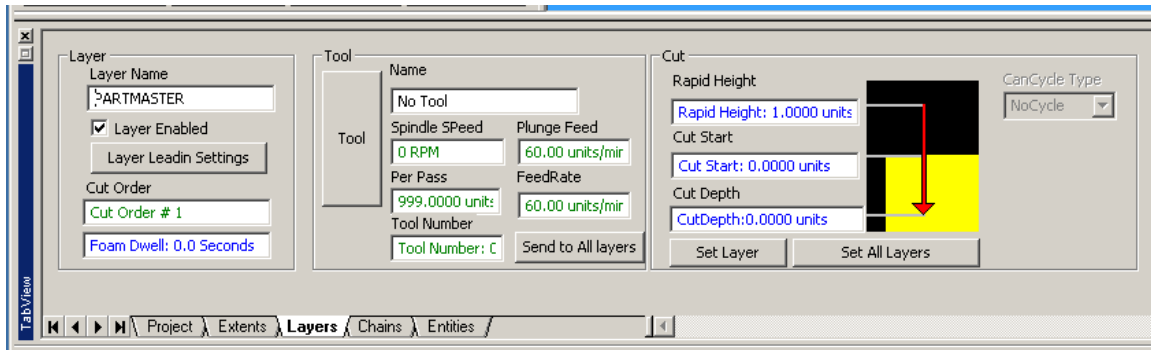
The project tab should be monitored to see LC's activity and provides command status.



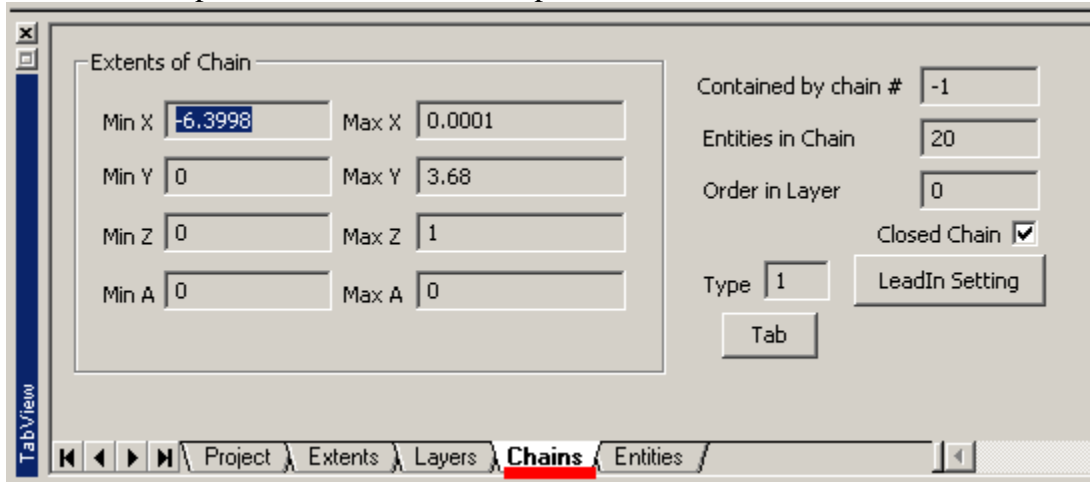
The extents tab will show you min / max values for the profile, how many chains, entities, etc there are. One important one is the connection tolerance as this relates to how the drawing of your profile was done and how LC will connect the lines etc.



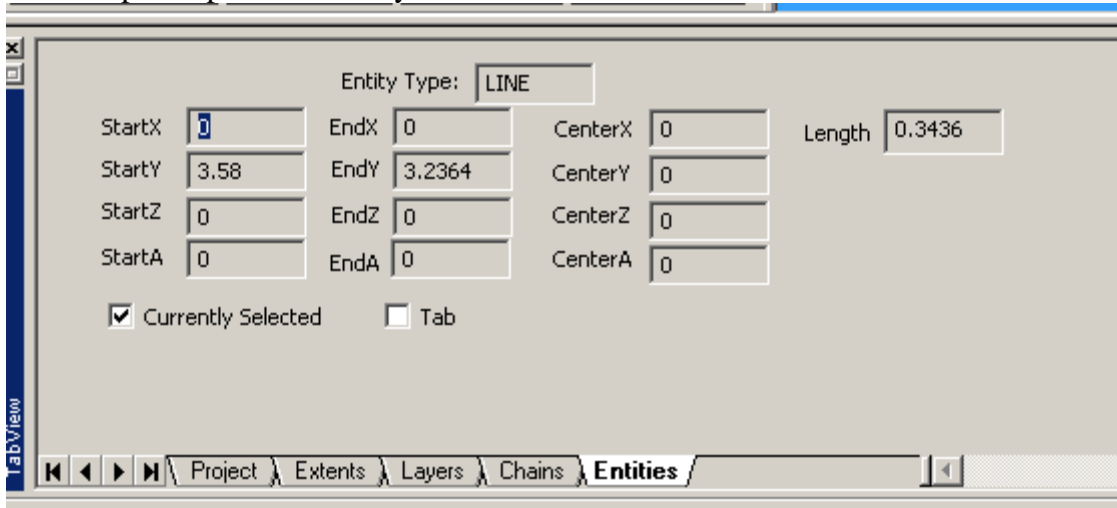
The Layer tab allows for tool creation and picking a tool and how it should be applied to a layer. Then you also have choice of how the tool will travel when cutting.



The Chain tab provides information on a particular chain.

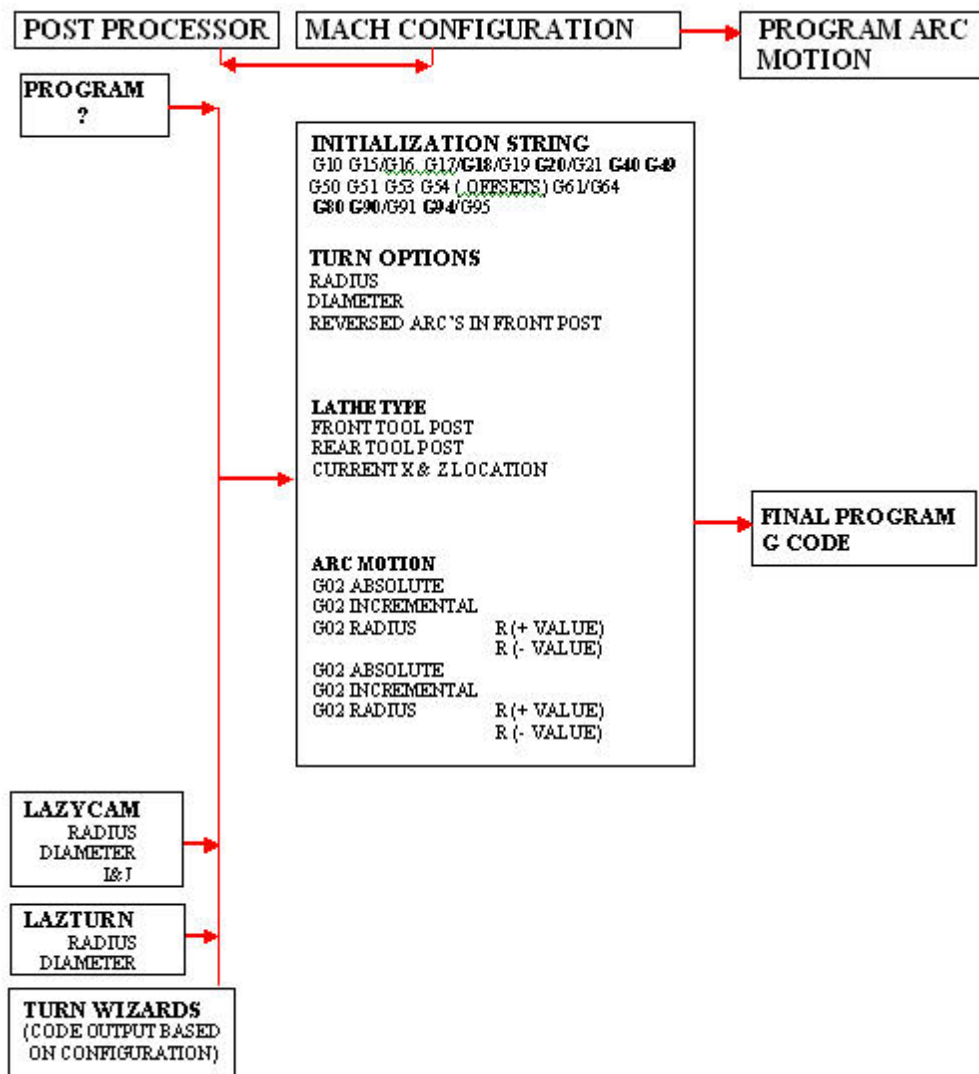


You can pick a particular entity and see it's related information.



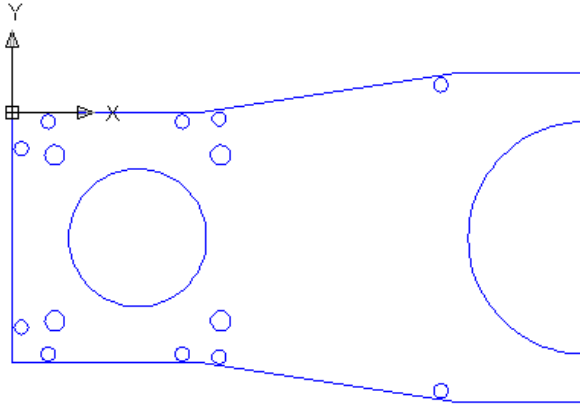
So much for the Tabs, Buttons, and screens and sorry for no definitions for each command as I am not going to write a book! Just follow the example which is next.

How you have Mach3 Mill or Lathe configured will affect posting of the gcode. Here is just some logic to look at that relates to Lathe but makes the point.

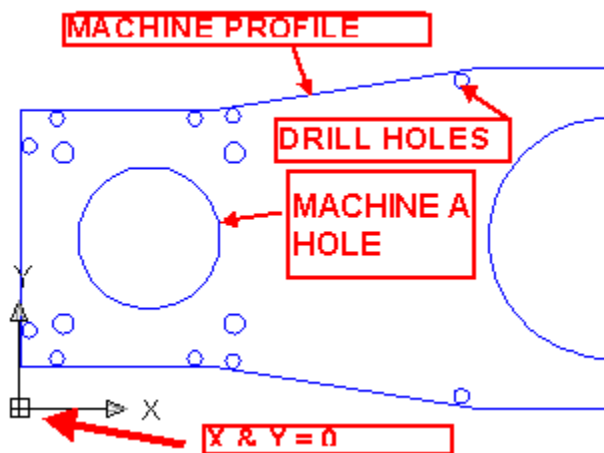


MILLING A PROFILE

Lets start with the CAD drawing of the profile. This particular one is a plate that needs to be machined out.. Additionally it has a big hole which needs to be machine out and there are holes that need to be drilled. I am going to do this step by step as an example. It will show you how to contour a piece and drill the holes. It's probably represents much of what a typical user of LC would want to do.



First of all I don't like the idea of having the origin of the part as shown above. On the mill I prefer to work from the lower left corner of the piece since all gcode and coordinates will be positive. Here is the updated CAD.



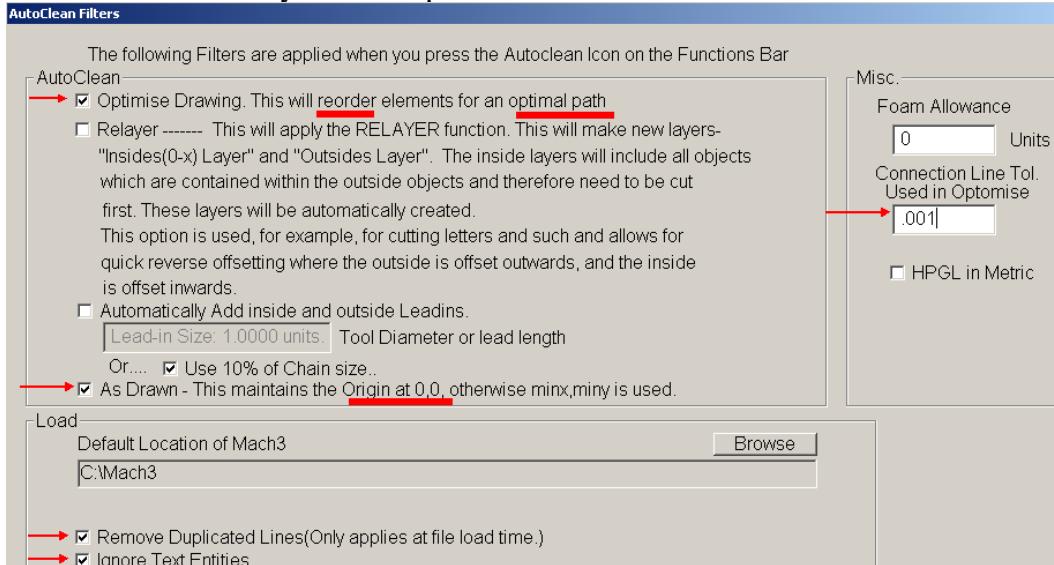
Now if you are wondering about DXF files and how to draw them I would suggest that you take a look at the LAZYTURNS MANUAL in members docs as it has an appendix on dxf files and drawing problems which although relate to the lathe are relevant.

So I save the file in CAD as version 12 (other version work but recommend 12)

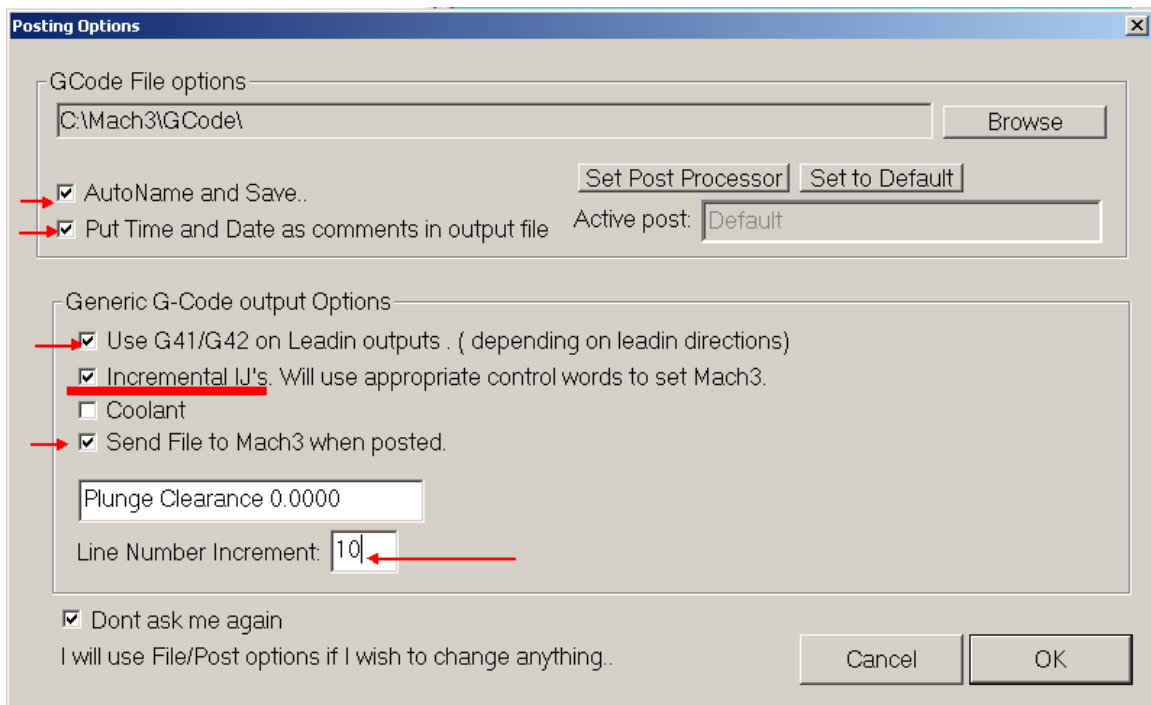


LC SETUP:

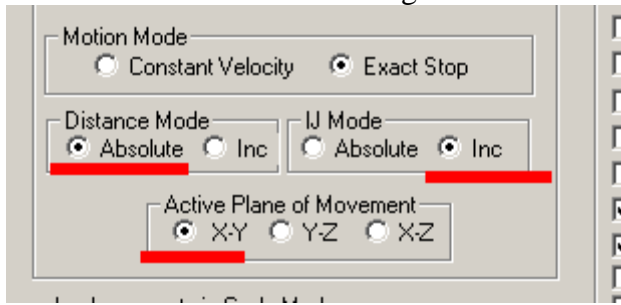
Note that I want to optimize the drawing and not base it on how it may have been drawn. Using a small connection line tolerance (I draw accurately!) and want to leave the drawing as drawn with it's origin at X& Y = 0,0. There is no text in the drawing and should not contain any extra / duplicated lines.



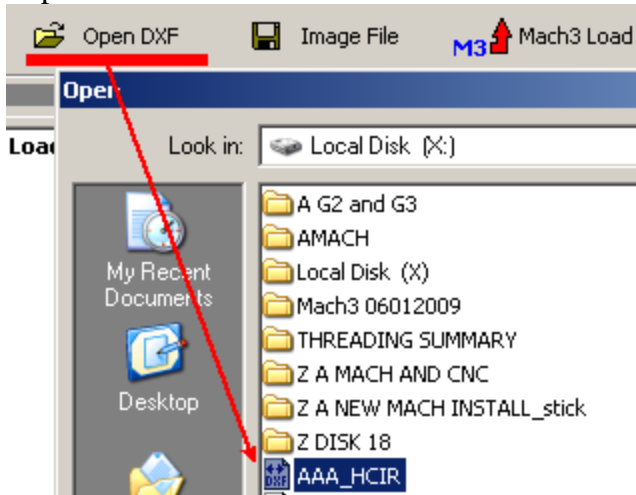
Here are posting selections. We will try incremental IJ's



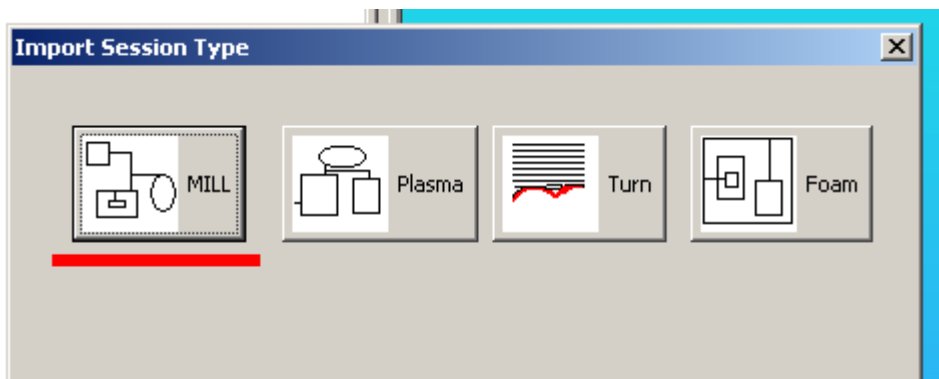
Here is the MACH3 Mill configuration.



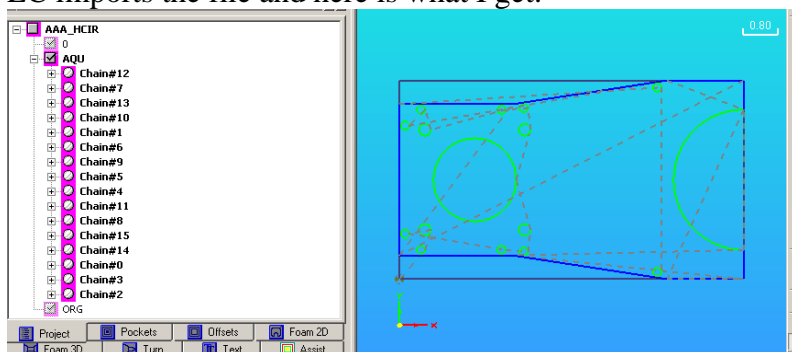
Import the DXF file into LC.



Tell mach that it will be a mill session. Click the MILL box.



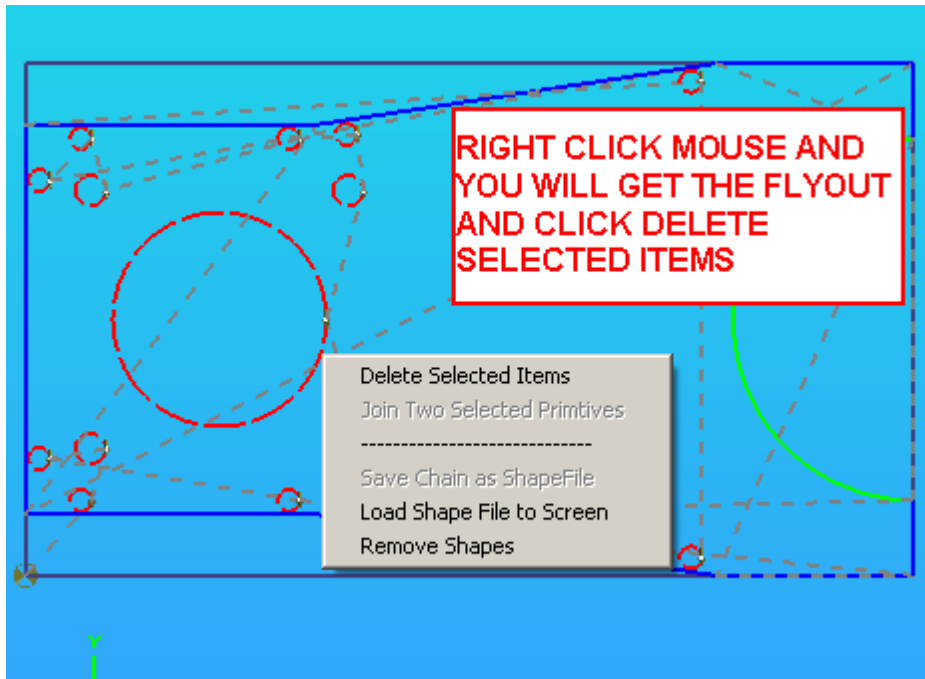
LC imports the file and here is what I get.



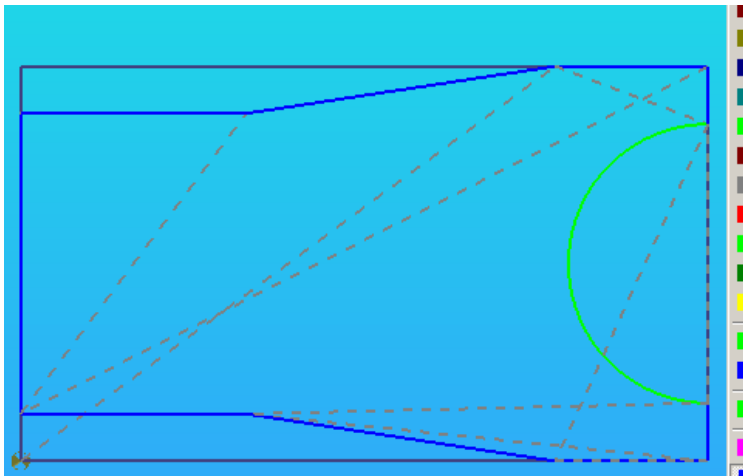
I only want to machine the outside plate so I am going to delete the entities I don't want. Click the Single / Multiple tool, then click each entity you don't want as shown below.



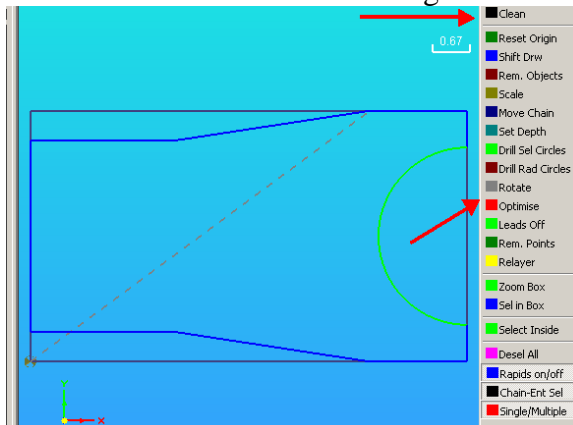
Place the mouse on one of the entities, right click the mouse and you will get the fly out with the menu item to delete the selected items, click on it and when you get a prompt say yes.



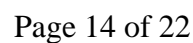
LC deletes the items as shown below.



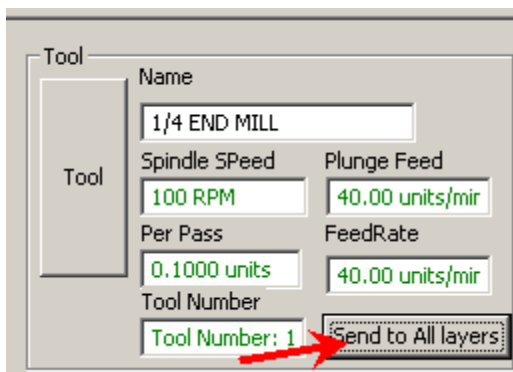
We still need to do some cleaning so click clean and the optimize tools.



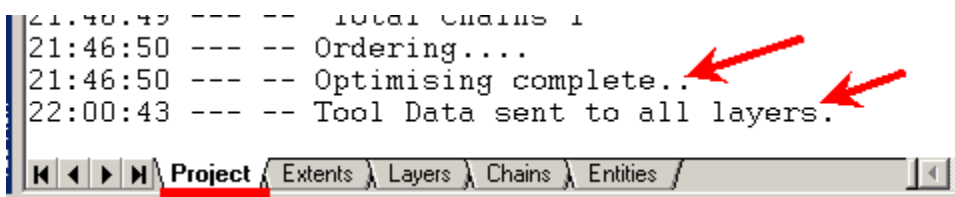
Click the layers tab, click tool, fill in the information, click create, give the tool a name click ok , and then again ok data flyout as shown below.



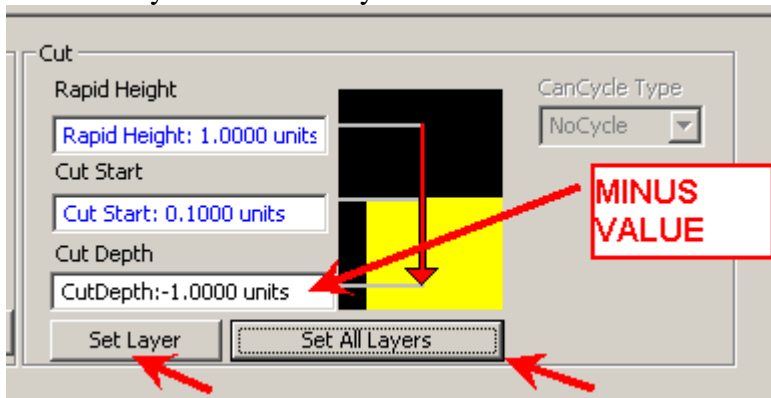
Notice the data has changed in the layers tab, click Send to all layers box.



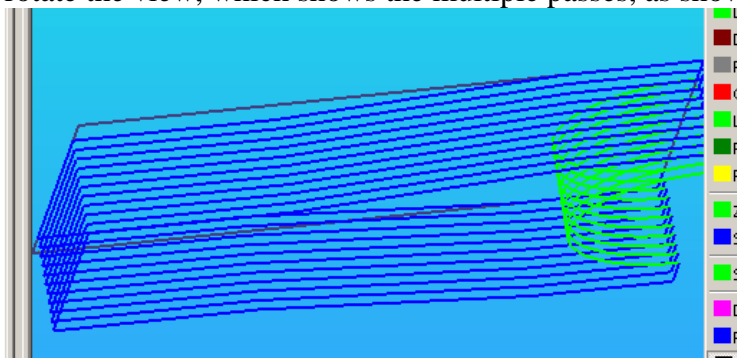
Click the project tab and notice that the optimizing that we did before is complete and that the tool data was sent to all the layers.



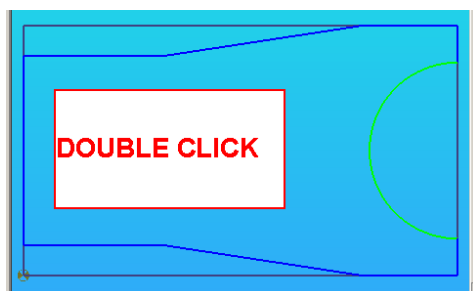
We defined a tool to cut with but now we need to tell LC how we want the tool to travel. This is done by providing the cut data as shown below. Note that we will be doing 10 cuts since our tool per pass was set for .1" and we are going to cut to a depth of 1". Click set layer and set all layers.



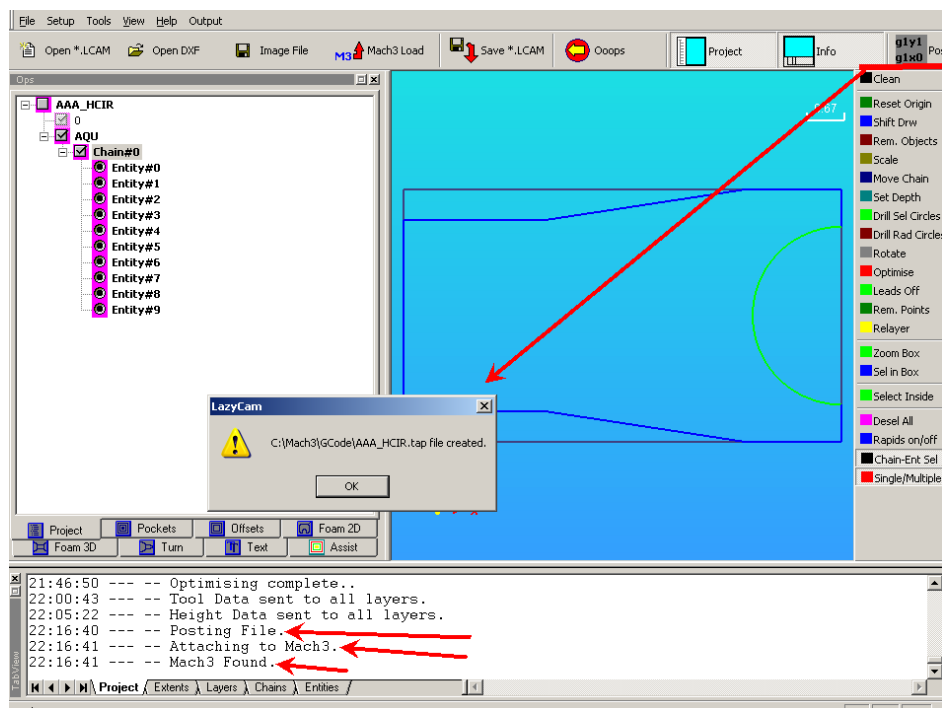
Move your mouse into the graphics area, then hold the scroll wheel and move the mouse to rotate the view, which shows the multiple passes, as shown below.



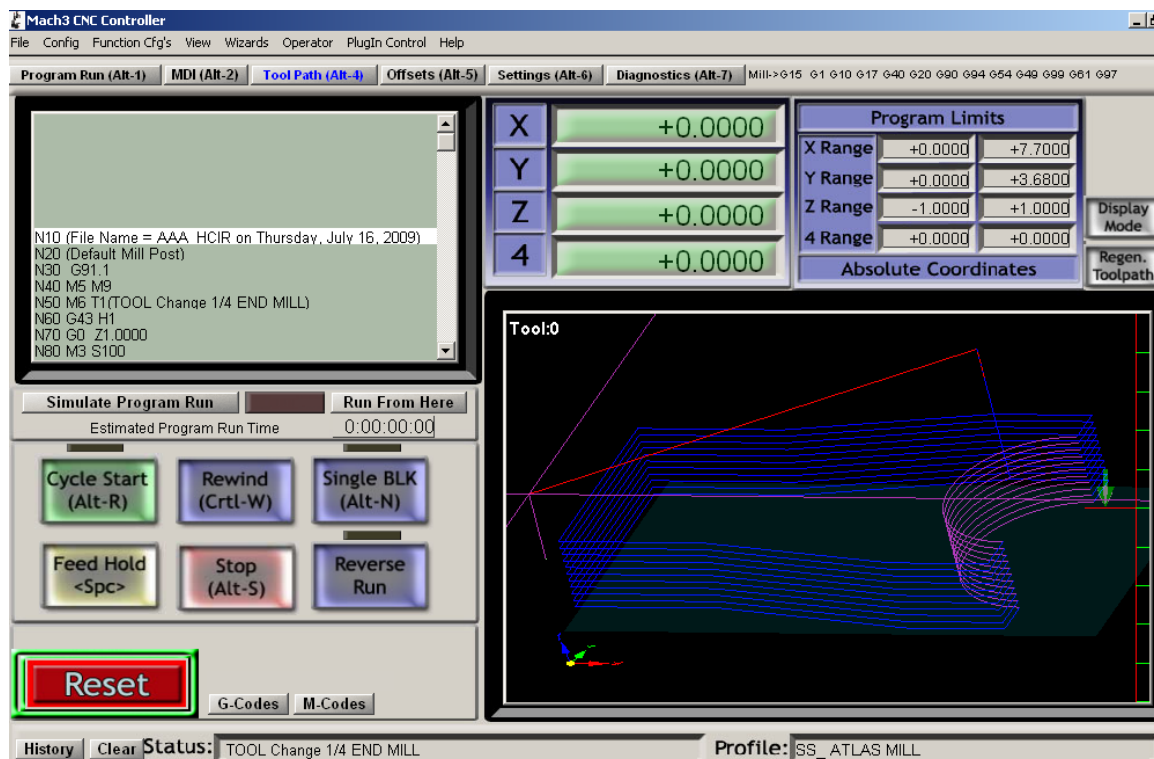
Double clicking in the graphics area returns the graphic to plan view.



Click POST code button and the code is sent to MACH3, click ok in the posting box.



Minimize LazyCam window and then you will see the posted code in MACH3 Mill.

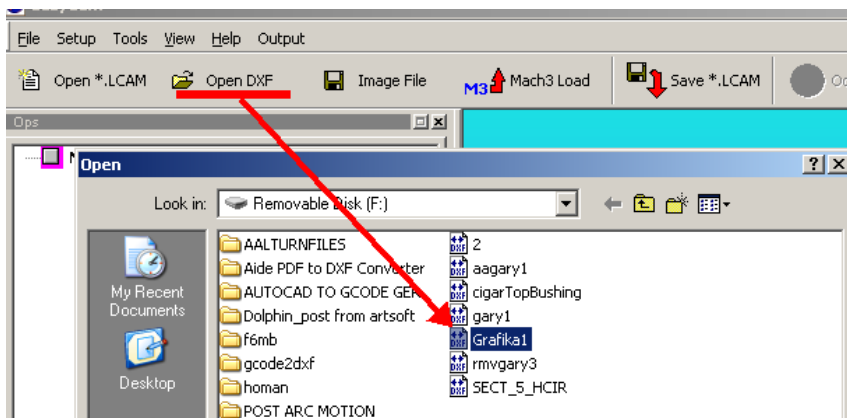


Now wasn't that easy. Only one problem with the above, it will cut the profile out exactly as drawn and is not offset to outside of the profile. Now you can get tricky and scale the X & Y in Mach3 but we will go back redo the it LC.

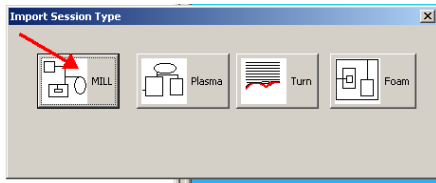
MILLING SOME TEXT

Here is a recent posting on how to mill some text, quick and dirty but available as a crude example:

1. Open the DXF file.



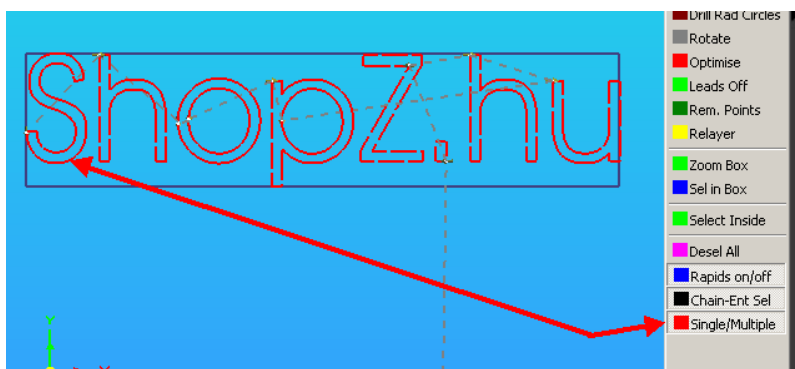
2, Click Mill



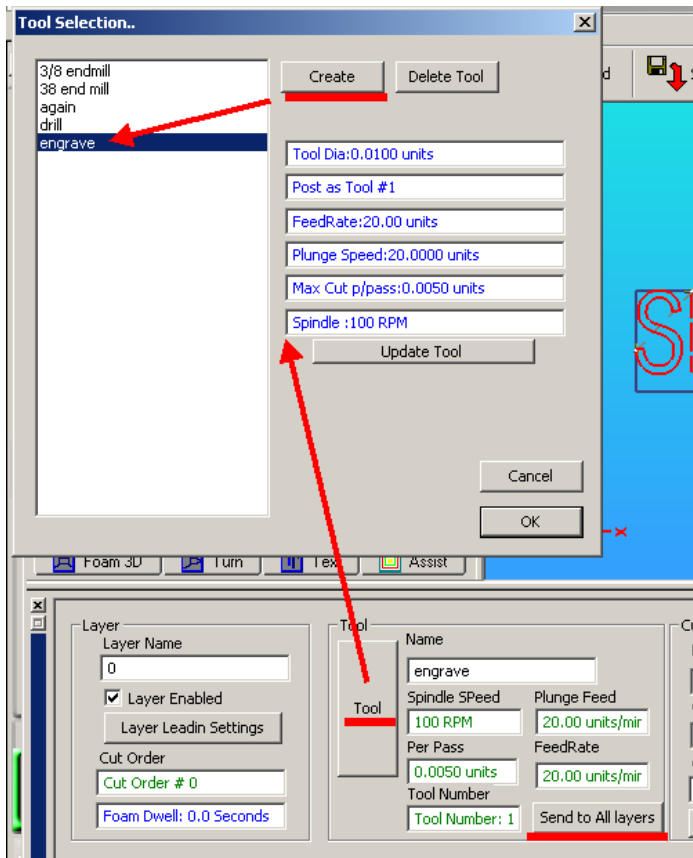
3. Click Clean and then Optimize. Then also click Single / Multiple.



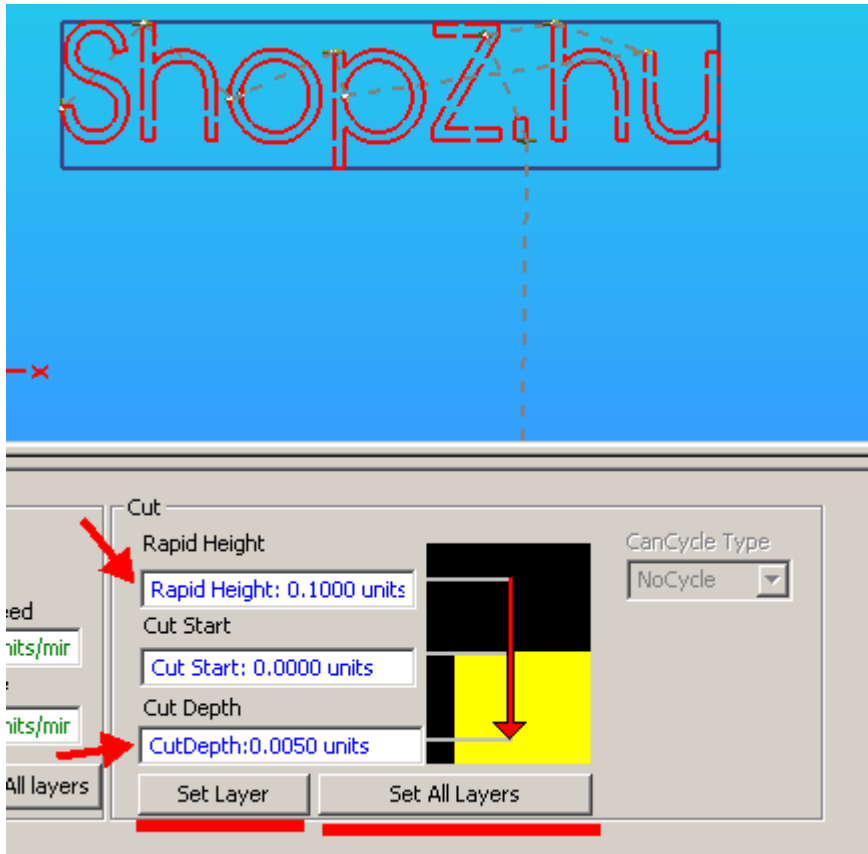
4. Now go and select each of the text parts.



5. Open the Layers Tab and click Tool, fill in the data for the tool, click Create, then click OK. Select the tool and click OK, then click Send to All Layers. Notice I set the per pass cut depth to .005”.



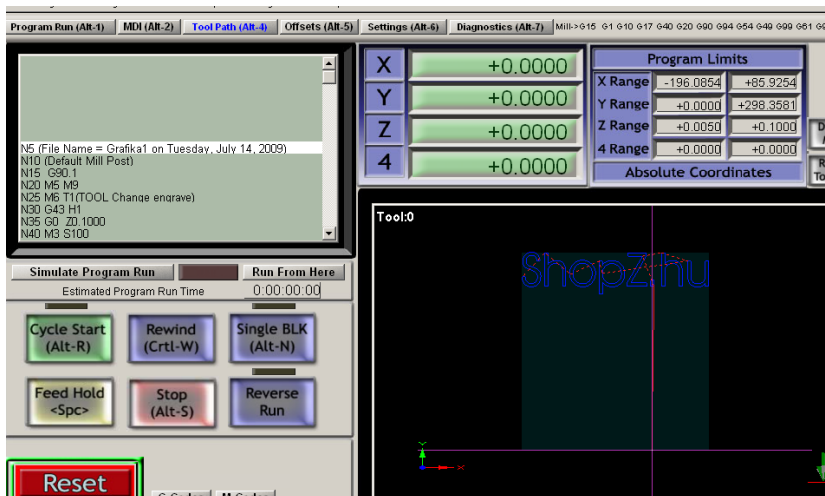
6. Now fill the data for how you want to cut the text. In this case it's a single pass. The cut depth pass should be negative and not positive.



7. Click Post Code (note that there are some settings which you can select for posting code under the Setup Tab). It creates the file so click OK.



8. Go out of LazyCam or minimize, you may also want to save the file and Lazycam as an .LCAM for later retrieval. Here is the file posted into MACH.



9. You may need to have a license for LazyCam as since the number of lines are restricted without a license.