

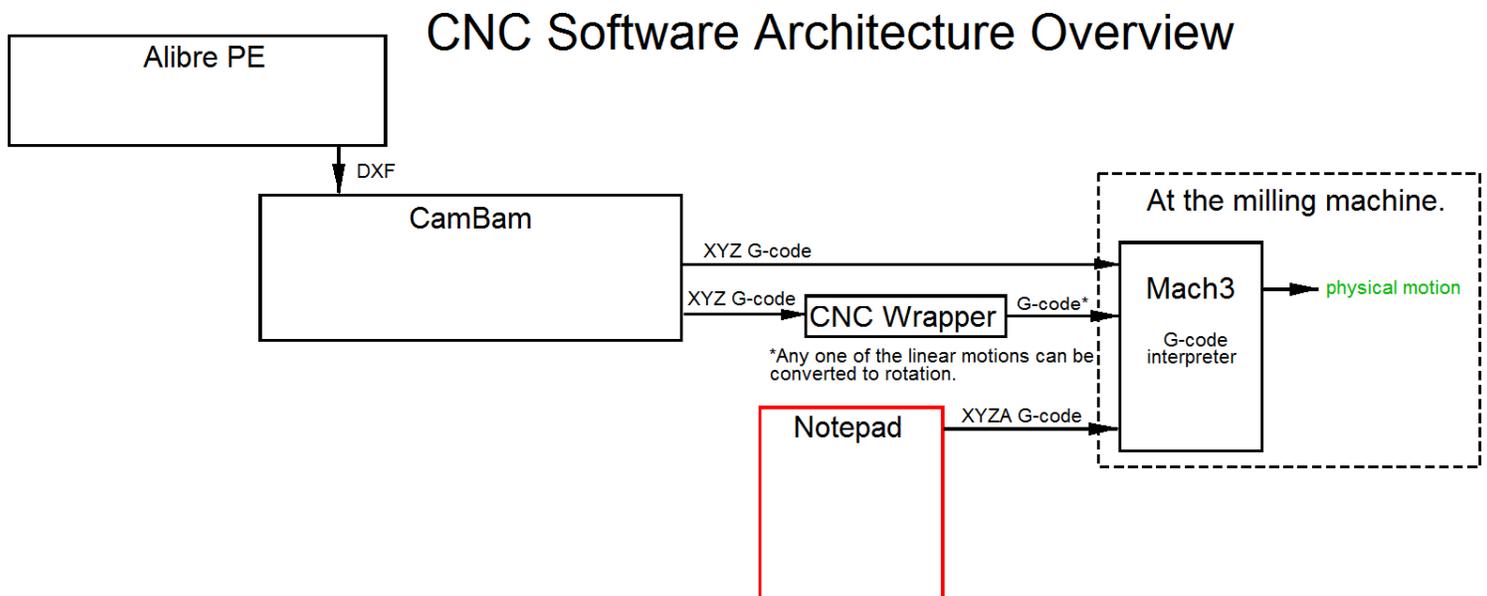
My Computer Numeric Control Software Architecture, version 1.0

By **R. G. Sparber**

Copyright protects this document.¹

Overview

Just as a brick wall must include mortar to hold it all together, a collection of software needed to perform Computer Numeric Control (CNC) must also include information on how the software interacts. When all software has been "cemented" in place, we can call the arrangement an architecture. Here is my CNC Software Architecture. I fully expect it to evolve over time.



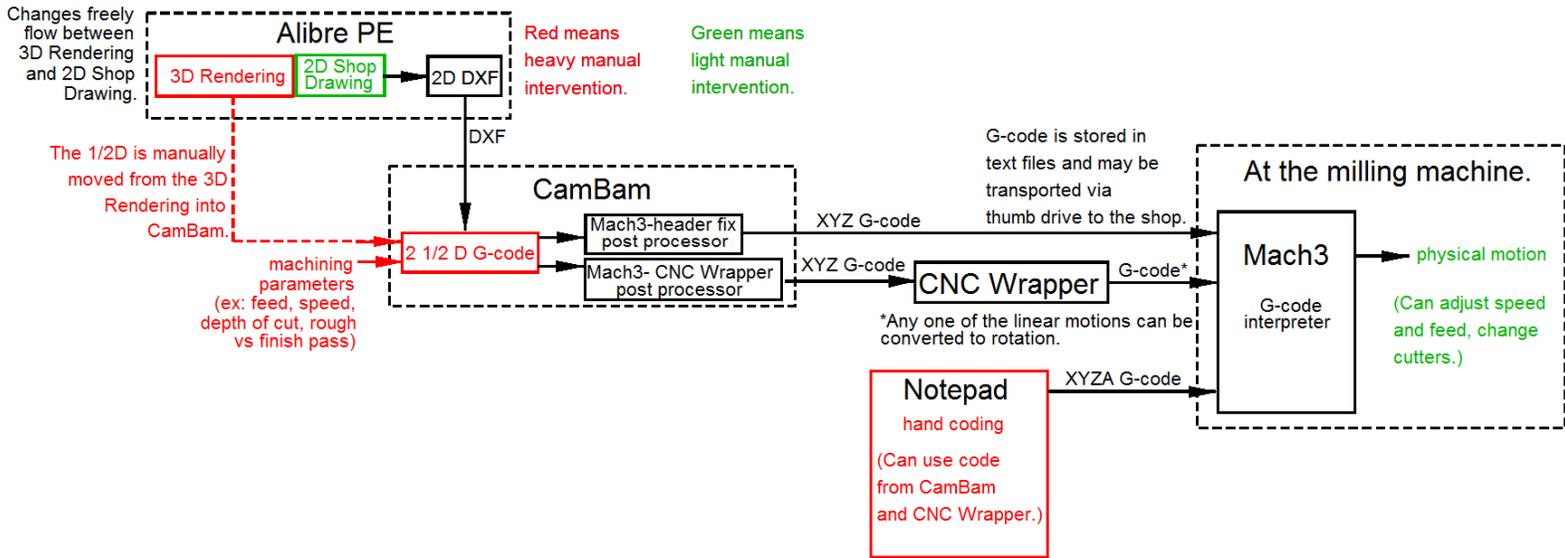
Alibre PE lets me generate 3D renderings of parts. It can output one or more *Drawing eXchange Format* (DXF) files that represents 2D views of the 3D part. **CamBam** converts each DXF into a 2 ½D G-code file. **Mach3** interprets the G-code and operates its CNC milling machine. Optionally, **CNC Wrapper** can take the G-code from CamBam and convert it into a form that permits engraving onto cylinders. **Notepad** permits me to hand generate G-code which is acceptable to Mach3.

¹ You are free to distribute this article but not to change it.

The Details

Here is the entire architecture:

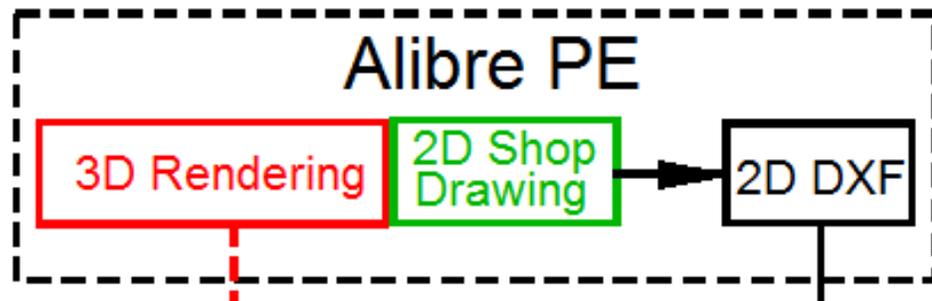
CNC Software Architecture



I will next discuss each part and how it interacts with the rest of the architecture.

Alibre PE

Changes freely
flow between
3D Rendering
and 2D Shop
Drawing.



Alibre Personal Edition (PE)² is a very capable 3D rendering program. It took me over 150 hours to learn Alibre well enough to draw anything I wanted to make.

I draw a 3D object, define 2D views of this 3D object which are stored in 2D shop drawings. Changes in the 3D object are automatically propagated to the various shop drawings. This fluid behavior eliminates errors between the two worlds and saves a lot of time.

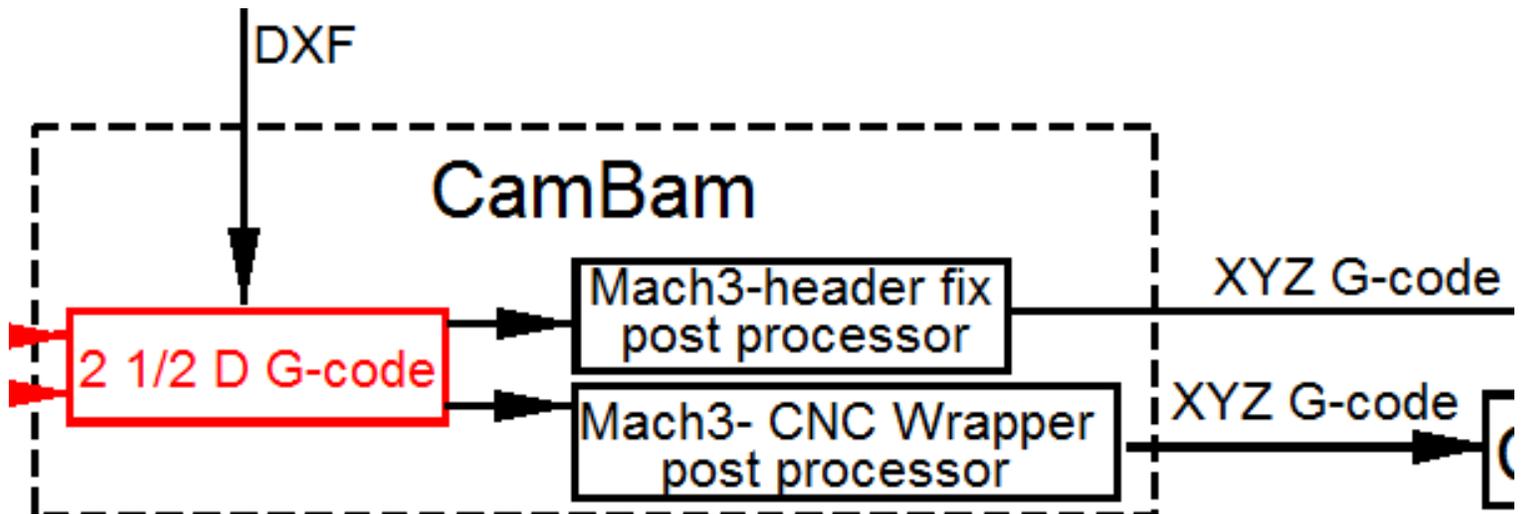
Once you generate a DXF from the shop drawing, the gate closes. Any changes to the design beyond the DXF has no effect on the 3D rendering. If you have not yet fed the DXF to CamBam and need to make a change, it is best to update the 3D rendering which will automatically update the 2D shop drawing. It is then a simple matter to export the data to a new DXF file.

For the sake of good documentation, the 3D rendering should be manually updated so all modified G-code is represented correctly.

See section 1 of <http://rick.sparber.org/AGP.pdf> for details.

² Alibre PE has become <http://www.geomagic.com/en/products/design/overview>

CamBam 0.9



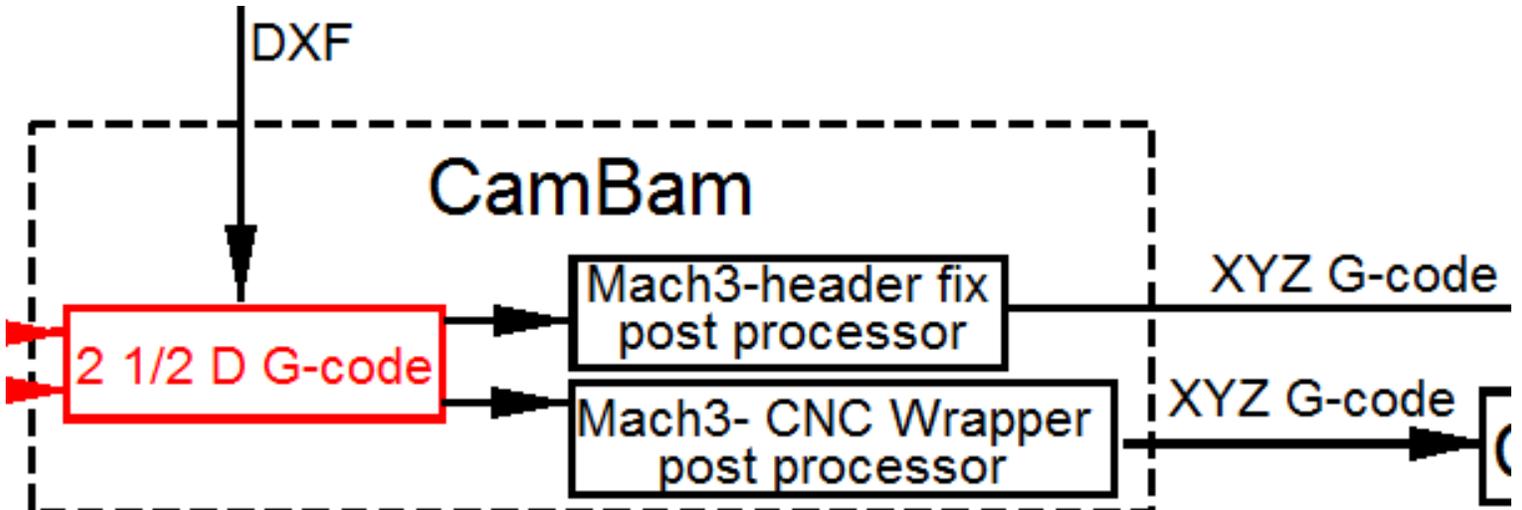
Although CamBam does do a lot of automatic G-code generation, it also requires a lot of manual intervention. A working knowledge of manual machining is essential.

One DXF provides a single 2D view of the part. This view consists of one or more lines. The user must define, for example, which cutter to use, how deep to cut, the RPM of the cutter, the feed rate, and whether this is a rough or finish cut. Tools such as G-Wizard can help the user optimize parameters but there is no substitute for manual machining knowledge.

Of equal importance but beyond the scope of this article is fixturing. How you secure the stock so CNC machining can safely and accurately occur is critical.

The DXF contains 2D information. CamBam outputs G-code able to guide a milling machine into making 2 ½ D parts. So where does that ½ D come from? It is supplied by manually populating fields associated with each line defined in the DXF.

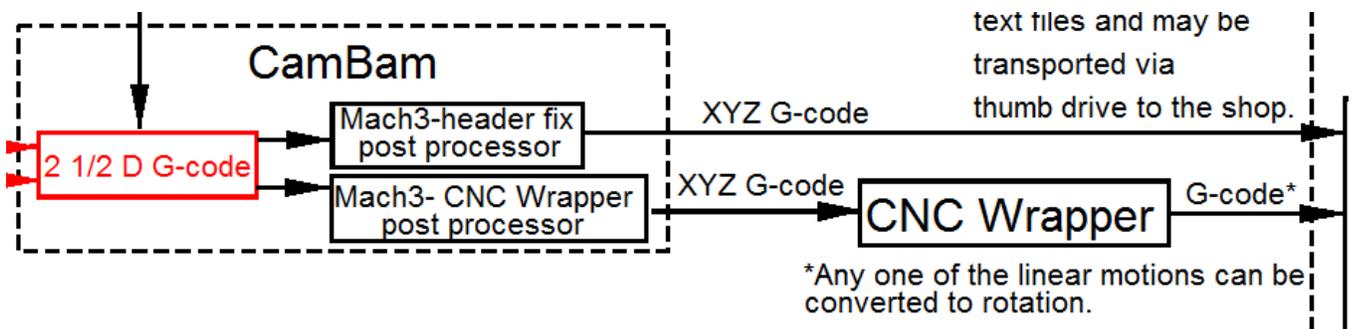
See section 2 of <http://rick.sparber.org/AGP.pdf> for details.



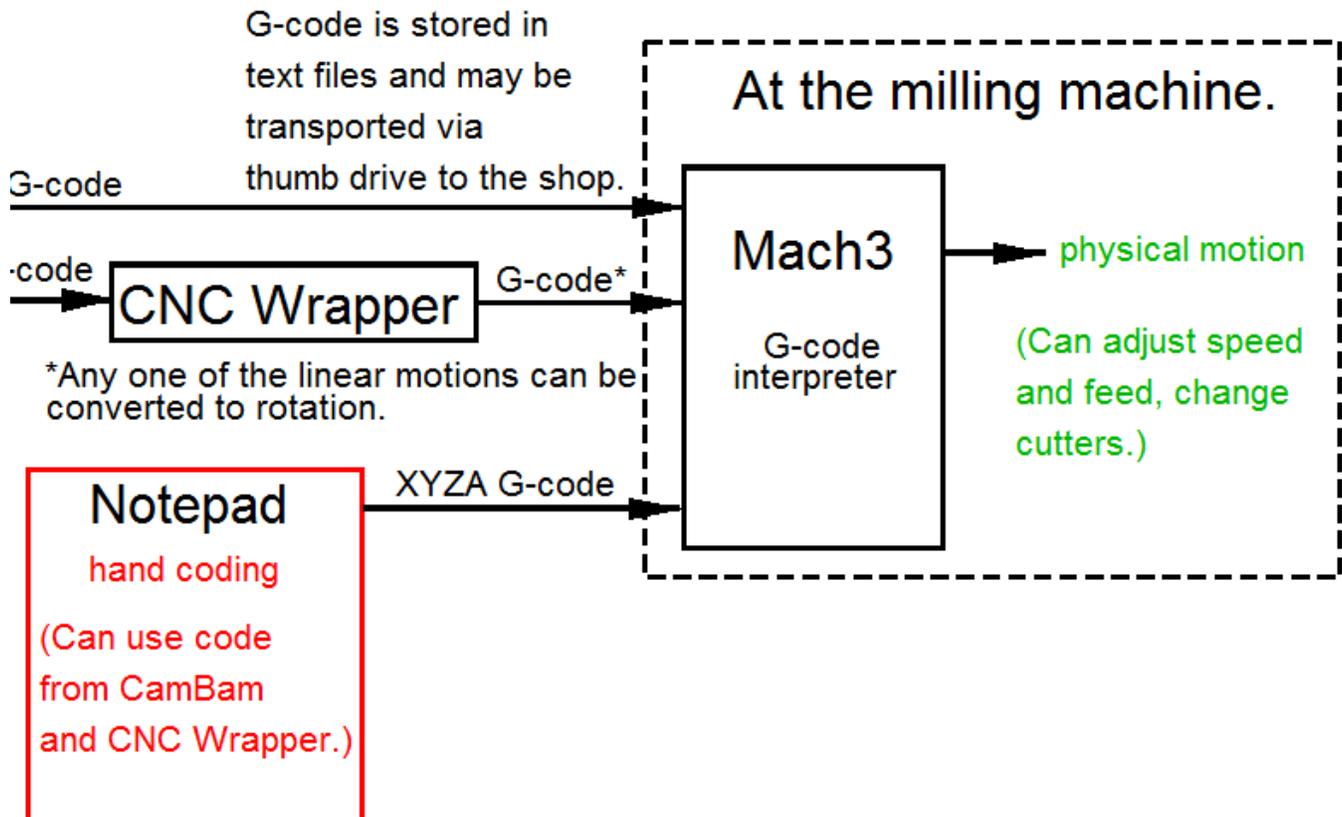
After the 2 1/2 D G-code is generated, I can select one of two custom Post Processors.

If I plan to control motion along the X, Y, and/or Z axes, then I will call on the "Mach3- header fix" post processor. See <http://rick.sparber.org/ABCM.pdf> for details.

If I plan to control motion involving engraving on a cylinder, I will call on the "Mach3- CNC Wrapper" post processor. See <http://rick.sparber.org/ENCY.pdf> for details. This post processor converts any arc commands in the G-code to lines.



Notepad

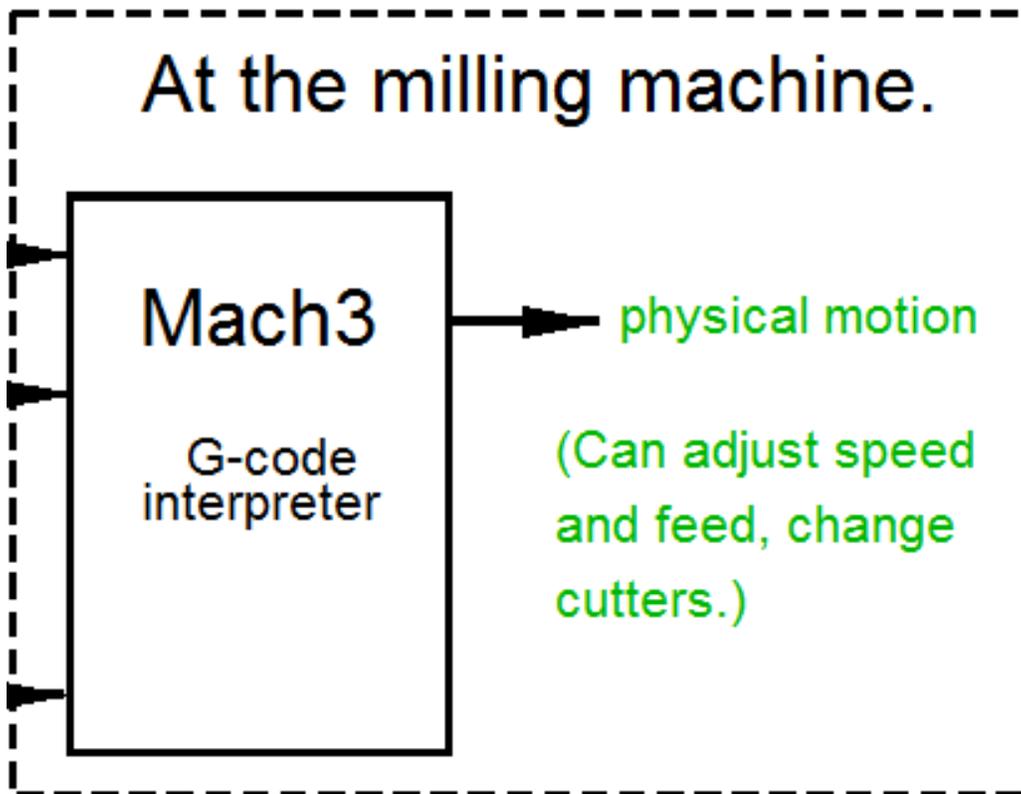


It is common to watch the CNC mill operate and see a need to change something. Sure you could go back to Alibre and make the change. Then re-input all of that data into CamBam to generate new G-code. Maybe you are much more careful than me and do not inject new bugs. Or maybe you are as careful as me and only inject a few new bugs. Either way, it can become one step forward and two steps back.

Often, it is far less risky to directly edit the G-code. To do that you will need a simple text editor. Notepad fills that need for me although CNC Wrapper has a fancier one built in. Don't forget to update the 3D rendering to document the change.

I also have times when I must hand code based on bits of code from CamBam. One example of this can be found at <http://rick.sparber.org/234D.pdf>.

Mach3



Mach3 accepts text files containing G-code and interprets it to physically operate the CNC mill. Mach3 also accepts limited manual intervention so parameters set in CamBam can be adjusted.

For details on my set up, see <http://rick.sparber.org/CNCHW.pdf>.

The PC that runs my milling machine is not connected to the internet and has no virus protection. All G-code files are moved to it via thumb drive.

Acknowledgments

Thanks to the countless people that have helped me get this far in my CNC journey. I know that I am barely crawling now but aim to someday walk.

I welcome your comments and questions.

If you wish to be contacted each time I publish an article, email me with just "Article Alias" in the subject line.

Rick Sparber

Rgsparber.ha@gmail.com

Rick.Sparber.org

