

# Setting Up CamBam 0.9 So it Correctly Feeds CNC Wrapper and Mach3

## Version 1.0

---

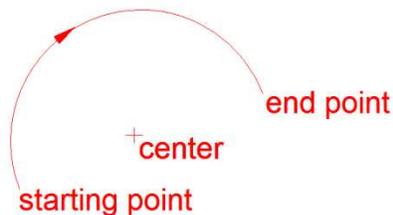
By **R. G. Sparber**

Copyright protects this document.<sup>1</sup>

If you go to the CamBam forum and search for "arc elimination", you will find the information presented in this article<sup>2</sup>. All I have done is filled in a few gaps for those needing a bit more help.

### Why Set-up CamBam for CNC Wrapper?

CNC Wrapper works by taking movement along a specified axis and converting it to rotation<sup>3</sup>. This works fine for all straight line movements. It doesn't work if an arc is specified.



The tracing of an arc uses a compact G-code structure involving a starting point, and end point, and a center. CNC Wrapper is simply not designed to "unpack" this structure.

---

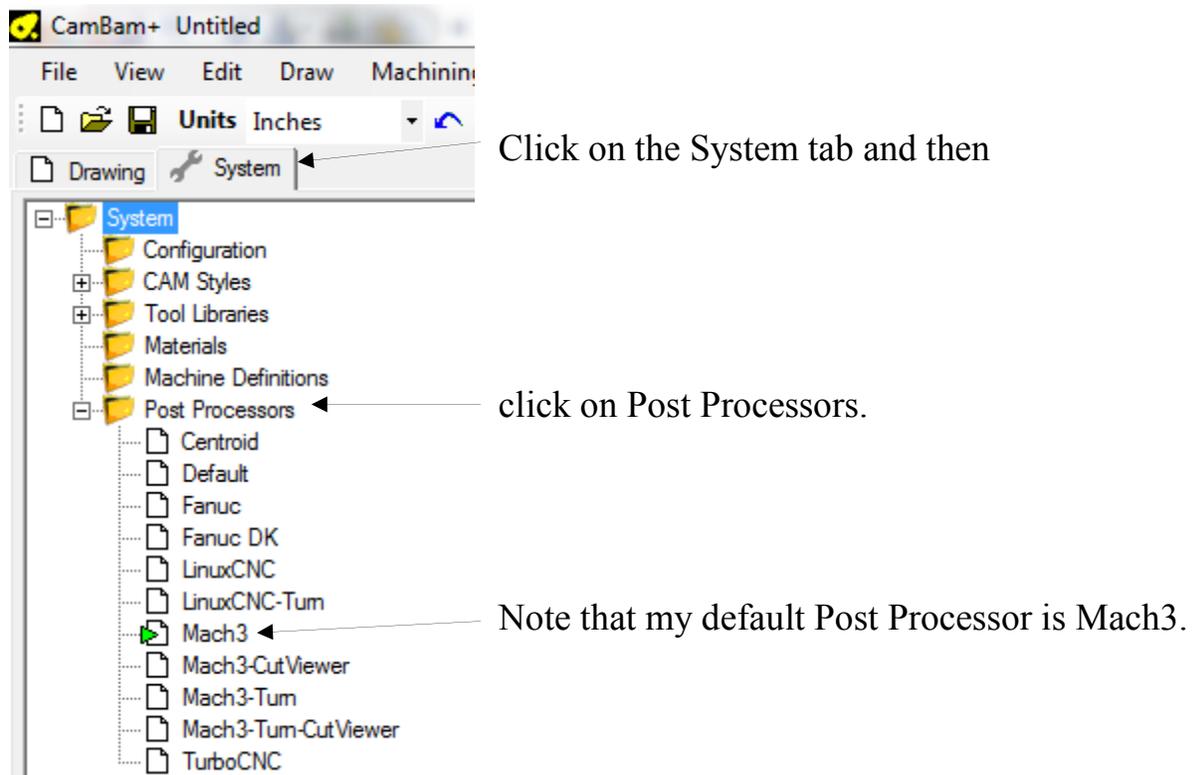
<sup>1</sup> You are free to distribute this article but not to change it.

<sup>2</sup> See <http://www.cambam.co.uk/forum/index.php?topic=4885.0> for the thread.

<sup>3</sup> See <http://www.cncwrapper.com/> for details.

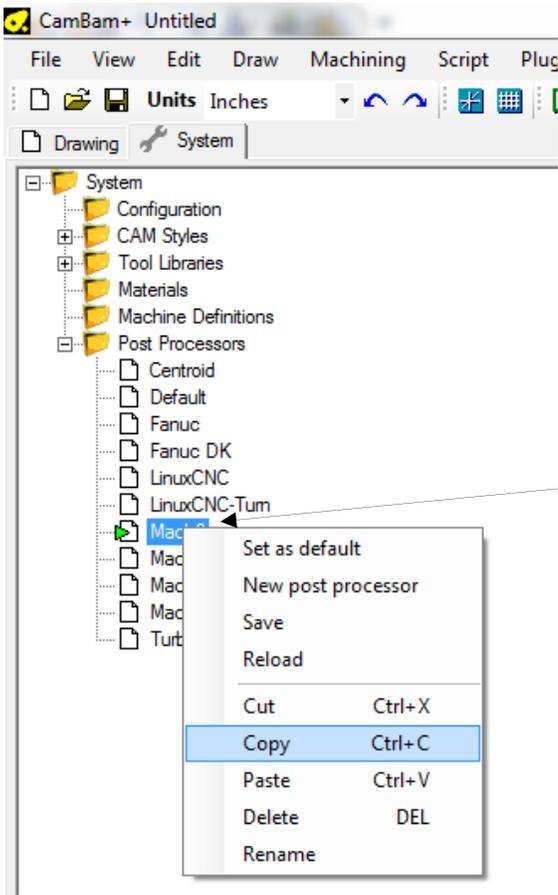
Instead, we must configure CamBam<sup>4</sup> to generate tiny line segments that closely approximate each arc as it creates the G-code file. This is done by defining a new Post Processor that will convert all arcs into line segments.

Here is how it is done:



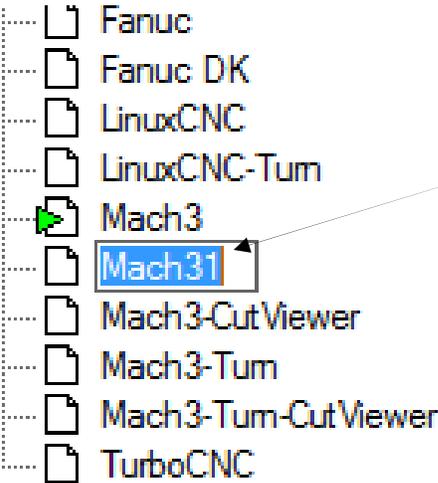
---

<sup>4</sup> See <http://www.cambam.info/>

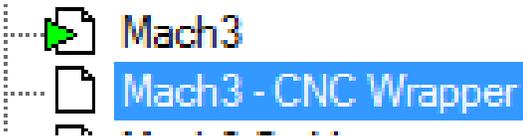


Click on Mach3 followed by a right click which should bring up a menu containing Copy and Paste.

Click Copy. Then right click again and this time click Paste<sup>5</sup>.



You should now see Mach31. Without touching the mouse, just type "**Mach 3- CNC Wrapper**" followed by Enter.



The result should be a copy of Mach3 with a new name.

<sup>5</sup> Sure, you can use the shortcuts too...

The screenshot shows a software interface with a file tree on the left and a table of G codes below it. The file tree includes folders like Mach3, Mach3 - CNC Wrapper (selected), Mach3-CutViewer, Mach3-Turn, Mach3-Turn-CutViewer, and TurboCNC. The table below is titled '(Main)' and contains various G codes and their corresponding values or macros.

(Main)	
End Cut	
Footer	{\$clearance}
Header	{\$comment} {\$cbfile.name} {\$date} {\$sendcomment}
MOP	{\$comment} {\$mop.name} {\$sendcomment}
Notes	
Post File	{\$comment} Made using CamBam - http://www.cambam.co.uk {\$
Post Processor Macros	
Start Cut	
Canned Cycles	
Drill	{\$g81} {\$x} {\$y} {\$z} {\$r} {\$f}
Drill Dwell	{\$g82} {\$x} {\$y} {\$z} {\$p} {\$r} {\$f}
Drill Peck	{\$g83} {\$x} {\$y} {\$z} {\$p} {\$q} {\$r} {\$f}
G Codes	
Arc Center Absolute	G90.1
Arc Center Incremental	G91.1
Canned Cycle End	G80
Canned Cycle Start	G98
Cutter Comp Left	G41
Cutter Comp Off	G40
Cutter Comp Right	G42
Distance Absolute	G90
Distance Incremental	G91
G0	G0
G1	G1
G2	G2
G3	G3
G81	G81
G82	G82

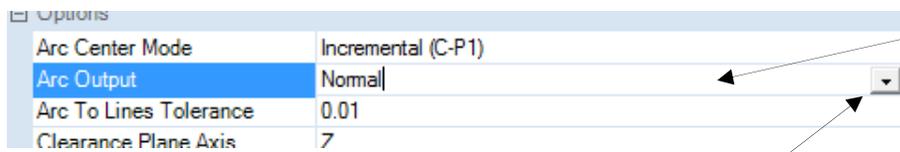
Below the table is a yellow box titled 'Arc Output' with the text 'Controls how arcs are output to goode'.

With Mach3- CNC Wrapper selected, notice the window below it.

Scroll this window down until you see **Options**. If there is a "+" in front, click it and you should then see a "-".

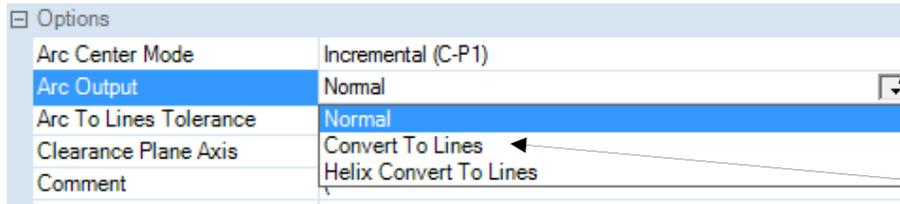
Basic	
Spindle CW	M3
Spindle Off	M5
Stop	M0
Moves	
Arc CCW	{G3} {F} {X} {Y} {Z} {I} {J} {K}
Arc CW	{G2} {F} {X} {Y} {Z} {I} {J} {K}
Feed Move	{G1} {F} {X} {Y} {Z} {A} {B} {C}
Rapid	{G0} {F} {X} {Y} {Z} {A} {B} {C}
Options	
Arc Center Mode	Incremental (C-P1)
Arc Output	Normal
Arc To Lines Tolerance	0.01
Clearance Plane Axis	Z
Comment	(
End Comment	)
End Of Line	\r\n
Invert Arcs	False
Maximum Arc Radius	100000
Minimum Arc Length	0.0001
Number Format	0.0###
Rapid Down To Clearance	True
Suppress Parser Errors	False
Upper Case	False
Post Build	
Post-Build Command	
Post-Build Command Args	
Tools	
Tool Change	{Clearance}
Tool Table Item	{Comment} T{Tool.index} : {Tool.diameter} {Sendcomment}

The second entry is Arc Output. The default is "Normal".



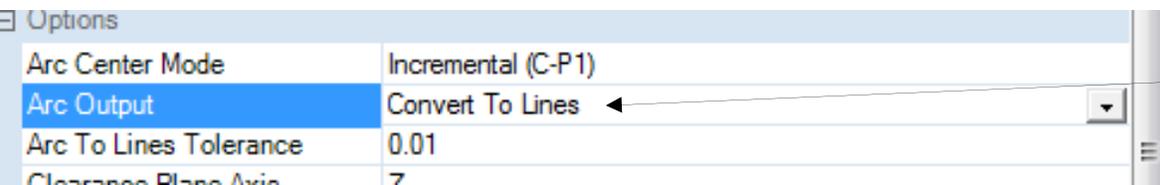
Click anywhere inside the box containing the word "Normal" and

a grey box with a down arrow will appear.



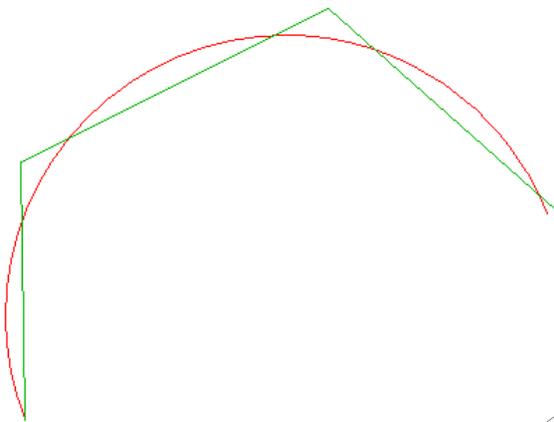
click on the down arrow and another drop down menu will appear.

Click on "Convert To Lines".

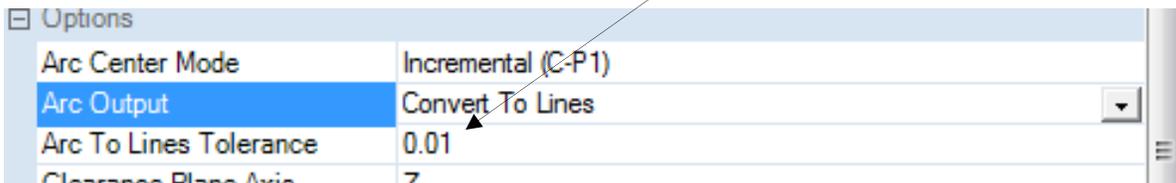


You should then see this.

The final step is to decide how closely our line segments must approximate each arc.



The original design specified an arc (shown in red) but CNC Wrapper cannot deal with them. Instead our new post processor will approximate the arc with line segments. We get to specify how closely these line segments will follow the arc. More accuracy - more line segments so larger file. You get to decide this trade off.



Arc Output

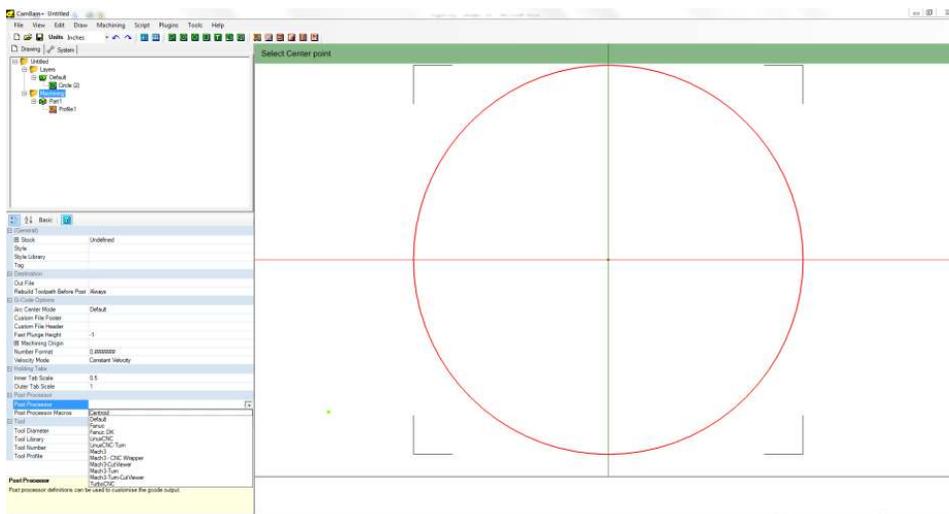
Convert To Lines

Arc To Lines Tolerance

0.001

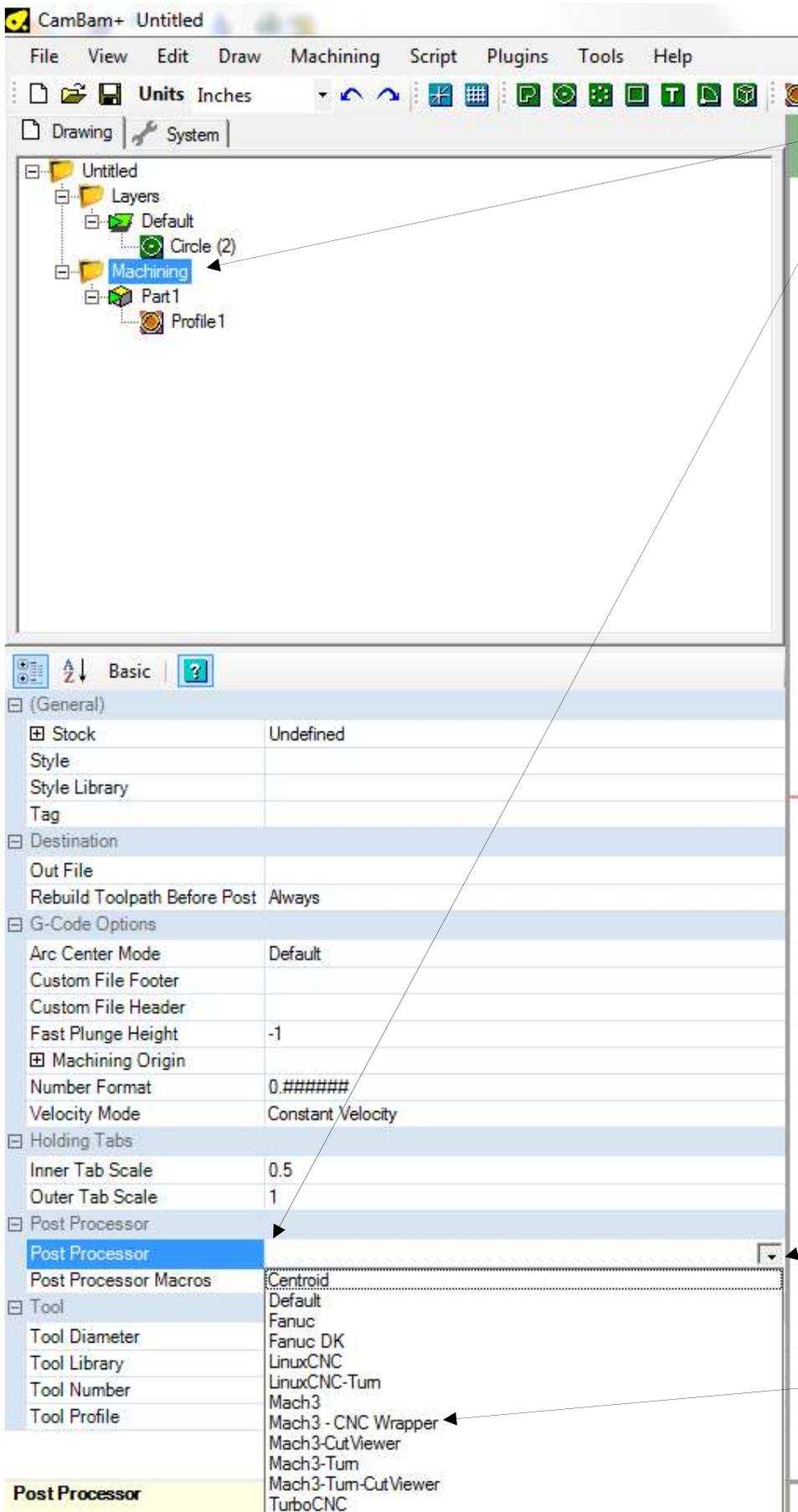
Click in the box with the dimension and you can modify the value. I chose to make the Arc to Lines Tolerance 0.001.

Keep in mind the size of the text to be engraved. If the letters are going to be 0.1 inches tall, then 0.001 inches of error should be reasonable. If the letters are going to be 4 inches tall, then maybe 0.01 inches of error is fine.



To show you how this works, I have drawn a circle of diameter 0.01 inches.

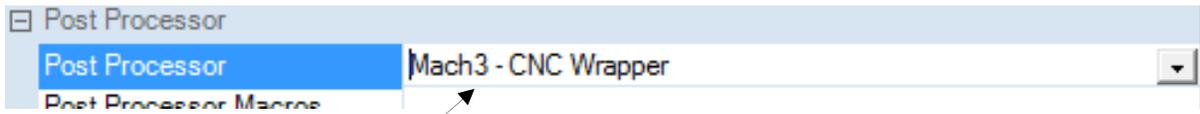
If using arcs, the G-code needed to draw this circle would be a series of arcs connected end to end. But since we will be using our new post processor, the G-code will only contain lines.



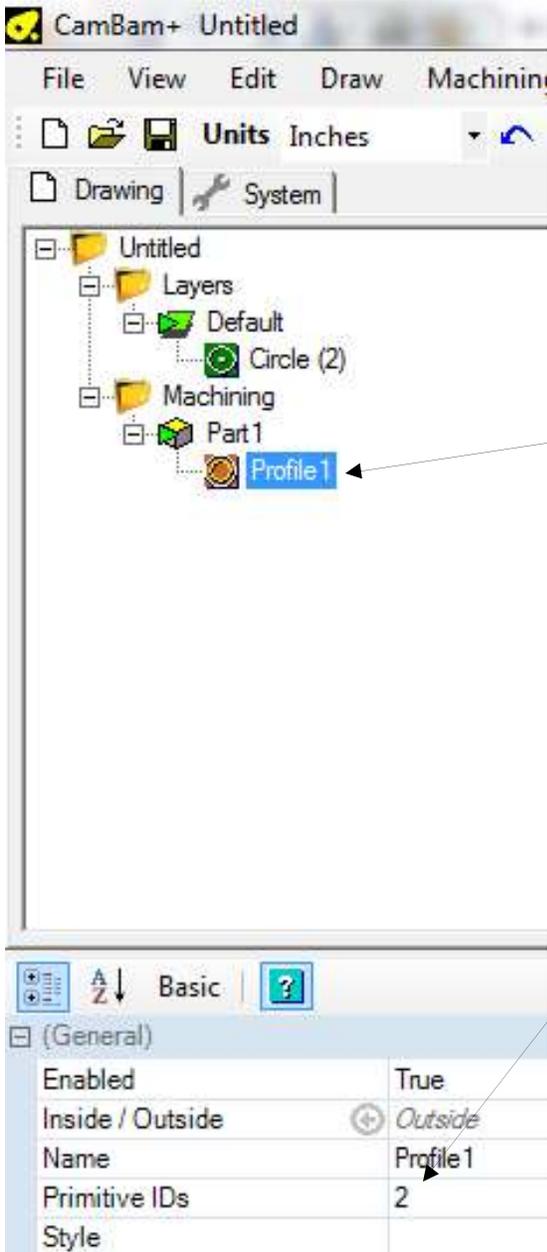
I click on Machining and then find the Post Processor entry in the window below it.

Click on the down arrow in the gray box.

Select our new Mach3- CNC Wrapper post processor.



You should then see this.



I have defined a profile that follows my circle. The circle has an ID of 2.

I then right clicked on Profile 1 and selected "produce gcode". This gave me:

( Made using CamBam - <http://www.cambam.co.uk> )

( Untitled 3/18/2016 5:24:32 PM )

( T0 : 0.0 )

G20 G90 G91.1 G64 G40 ←note 1

G0 Z0.125

( T0 : 0.0 )

T0 M6

G43 H0

( Profile1 )

G17

S0

G0 X0.015 Y-0.01 ← start of straight line segment approximation of a circle

G1 F1.0 Z0.0

Note that there are no G2 or G3 commands which specify arcs.

G1 F3.0 X0.0106 Y-0.009

G1 X0.0072 Y-0.0062

G1 X0.0052 Y-0.0022

G1 X0.005 Y0.0

G1 X0.004 Y0.003

G1 X0.0014 Y0.0048

G1 X-0.0018 Y0.0047

G1 X-0.0025 Y0.0043

G1 X-0.0046 Y0.002

G1 X-0.0049 Y-0.0012

G1 X-0.0032 Y-0.0039

G1 X-0.0025 Y-0.0043

G1 X0.0006 Y-0.005

G1 X0.0035 Y-0.0036

G1 X0.0049 Y-0.0008

G1 X0.005 Y0.0

G1 X0.006 Y0.0044

G1 X0.0088 Y0.0078

G1 X0.0128 Y0.0098

G1 X0.015 Y0.01

G0 Z0.125

M30

We got relatively few line segments because I set the error at 0.001 inches and drew a circle only 0.01 inches in diameter.

Note 1: My configuration of CamBam puts G90 and G91.1 on the same line. Mach3 ignores the G90 and only accepts G91.1. This is harmless if your Mach3 defaults are set for Absolute Distance Mode and Incremental IJ. If not, then you will need to break this line to get:

```
G20 G90  
G91.1 G64 G40
```

Then Mach3 will accept G90 on its line and G91.1 on a separate line.

## Acknowledgements

Thanks to the members of the CamBam forum, kvom, lloydsp, Bubba, dh42, and jeff\_Birt, for explaining how to configure CamBam so it generates CNC Wrapper compatible G-code.

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](mailto:Rgsparber.ha@gmail.com)

[Rick.Sparber.org](http://Rick.Sparber.org)

