

# An Introduction to G Codes

Thomas Allsup  
September 2011

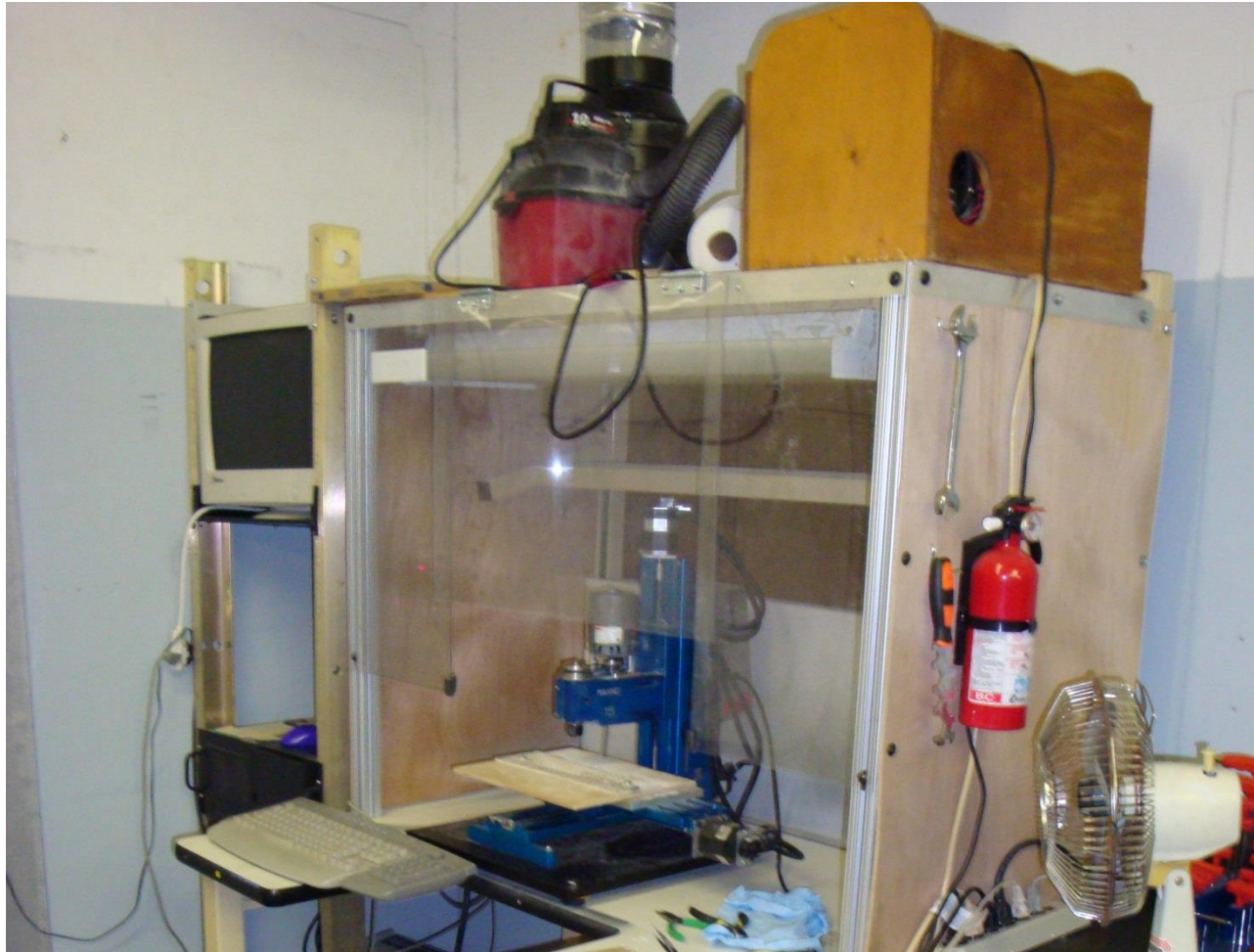
# Misunderstanding

- I hope no one is disappointed but this is not a discussion of the ***Gräfenberg Spot***.
- Be careful when you research the ***G-spot*** at work.
  - Mother will not be pleased!
- This talk is about “***G Codes***” 😊

# Why Do I Know Any of This

- In 1987, I was asked to design a special cam to be driven by a step motor with very specific maximum acceleration and deceleration.
- I wrote a QuickBasic program to convert the four equations into the cam shape and the cam follower position and eventually into DXF for import into CAD.
- Upon showing the machine shop what I had, the CNC programmer asked me to change my converter program from DXF or G code.
- The rest is history.

# Max NC 15



*Purchased 2007*

# Tormach



PROPRIETARY

**anida**  
technologies

# Numerical Control (NC) Defined

- Electronics Industries Association (EIA) standard defined as “a system in which actions are controlled by direct insertion of numerical data at some point.”  
.....machines controlled by electronic systems designed to accept numerical data and other instructions usually in a coded form.

# Second Misunderstanding

- G Code is not a programming language.
- G Codes are directives.
- G Codes tell computer numerical control devices how to move and behave.

# Third Misunderstanding

- Not all G codes start with G.
  - The codes that do start with G tell the machine where to “Go”.
- Some G codes start with M.
  - The codes that start with M tell the machine how to behave.
- There is no such thing as a “N code”.
  - Lines can begin with N and a number to designate the line number.



# Sample Program

(NC PROG. NAME: C:\MAXNC\WRENCH.t)  
(SET X AND Y AT CENTER OF MATERIAL)  
(MATERIAL: ALUMINUM 1/8 THK. \* 1.83 \* 6.22)  
G90G80G40G00G20  
M3S3000F2.00  
G43H1M08  
G00X-1.9749Y1.1409  
G00Z0.1000  
G41D6  
G01Z-0.0650  
G01X-1.4456Y0.5675  
G01X2.8842Y0.8750  
G02X2.8842Y0.4425I0.0002J-0.2162  
G01X2.2945Y0.4425  
G01X2.8842Y-0.4425  
G02X2.8842Y-0.8750I-0.0000J-0.2163  
G01X2.0918Y-0.8745  
G01X-2.5126Y-0.5675

G02X-2.5126Y0.5675I0.0001J0.5675  
G00Z1.0000  
G40X-1.0937Y0.5675  
G00X-1.9749Y1.1409  
G00Z0.1000  
G41D6  
G01Z-0.1300  
G01X-1.4456Y0.5675  
G02X2.8842Y0.4425I0.0002J-0.2162  
G01X2.2945Y0.4425  
G02X2.8842Y-0.8750I-0.0000J-0.2163  
G01X2.0918Y-0.8745  
G02X-2.5126Y0.5675I0.0001J0.5675  
G01X-1.0937Y0.5675  
G00Z1.0000  
G40X-1.0937Y0.5675  
M09  
G49  
M01  
M30

# Common G Codes

- G00 – Rapid positioning in mutual directions (X,Y,Z,A,B)
- G01 – Linear Interpolation
  - Needs a feed rate command to execute (F)
- G02 – Circular Interpolation Motion CW
  - R value or I and J center point values of circle
    - Full arc or quadrant input
- G03 - Circular Interpolation Motion CCW

# Less Common G Codes

- G20 – Inch Input
- G21 – Metric Input
- Canned Cycles – A predefined string of variables which allows for a predetermined sequence of events. This programming method serves to simplify input as well as reducing program size.
  - Examples G81 Drilling, G83 Peck Drilling,
- G80 Cancel Canned Cycle

# NC System Definitions

- Cutter Compensation – The ability to adjust the cutter location with offset values in the controller. The adjustment may be necessary due to tolerance issues associated with cutter condition, material problems, or program utilization (ex. family of parts)
  - G40, G41, G42

# Common M Codes

- M00 – M01 Program Stop / Optional Program Stop
- M02/M30 – End of Program / EOP Rewind
- M03 – Spindle On CW
- M04 – Spindle On CCW
- M05 – Spindle Stop
- M06 – Tool Change
- M08 /09 Coolant On/Off

# Fourth Misunderstanding

- Lots of commands are “Modal”
- I can not find any reason why it is called “modal”
- Once you send a command, it is in effect until you send another command.

# Modal Example

G00X0Y0

G00Z0

G01Z-.1

G01X1Y0

G01X1Y1

G01X0Y1

G01X0Y0

G00Z1

G00X0Y0

Z0

G01Z-.1

X1

Y1

X0

Y0

G00Z1

***Both columns have exactly the  
same movement!***

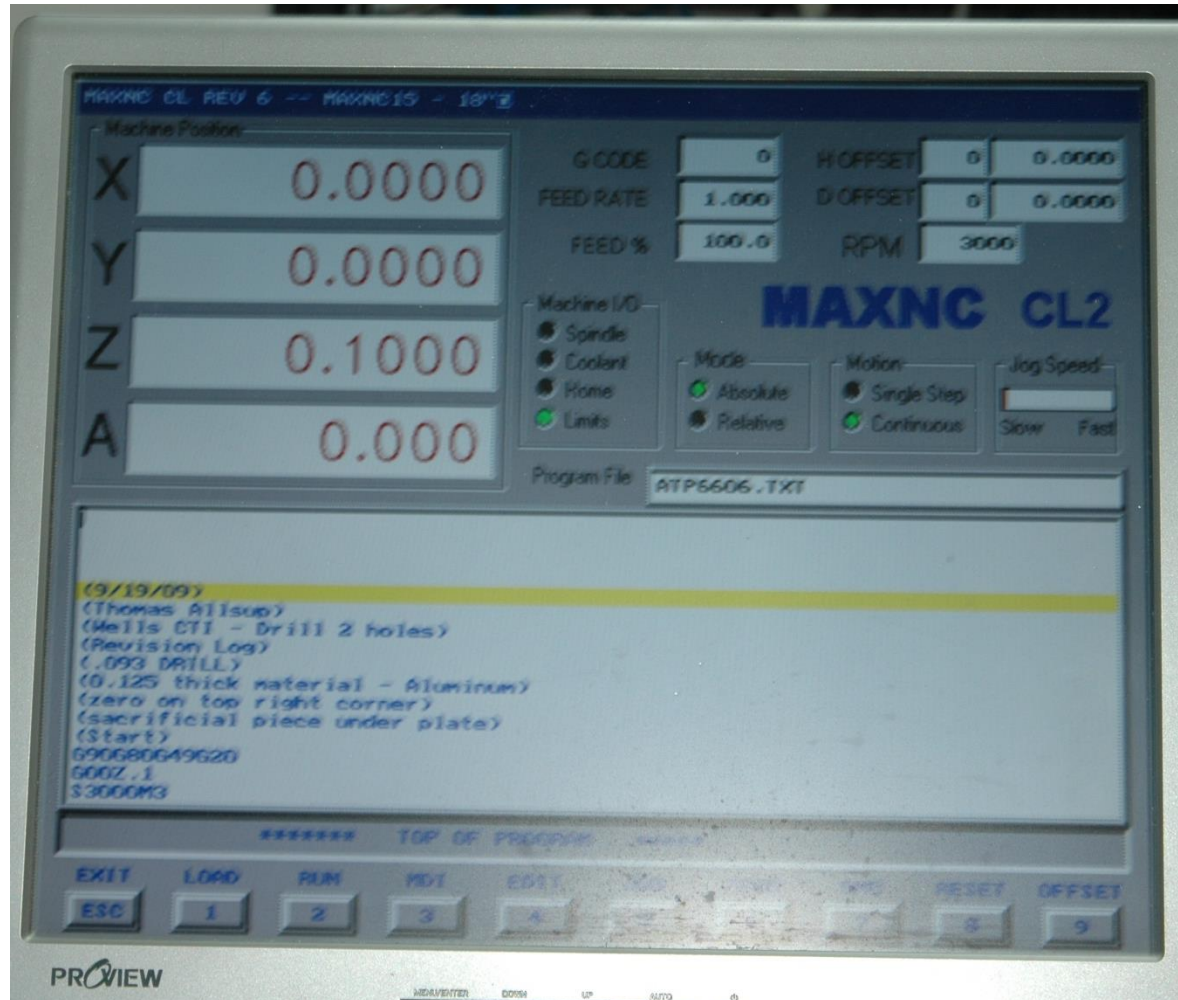
# What Software Do I Need?

## *Part 1*

- The CNC comes with a control program that reads the G code text file and instructs the CNC what to do.
- The control program does not care where the G code test file comes from.
- Normally the control program ships with the CNC.



# Control Program - DOS



**Note the use of the DOS "screen capture".**

# Control Program – Mach 3

Mach3 CNC Controller

File Config View Wizards Operator PlugIn Control Help

OpenG Recent Change G-Code ReLoad Close

Mill->G15 G0 G17 G40 G20 G90 G94 G54 G49 G99 G64 G97

X +0.0000 Ref All

Y +0.0000

Z +13.4253

Jog ON/OFF Ctrl-Alt-J

Jog Step - Num-\* 0.0010 Step

Jog Mode Num-/

Cont. Step Slow Joq Rate 100.0%

Shift to Ignore Slow Joq Rate

(ATP-6603 REV -)  
(WELLS CTI LATCH)  
(CUTTER 1/8" .125")  
(MATERIAL 3/8" THICK)  
(SACRIFICIAL MATERIAL UND

Cycle Start Alt-R

Hi / Low HIGH LOW

Max Speed 5140 RPM

S 5000

Spindle CW F5

CHANGE TOOL T 0 OFFSET ON

Empty

Z-off +0.0000

D +0.0000

Tool:0

Regenerate Toolpath Jog Follow Display Mode

MDI:

Z:\Anida Technologies\Jemison\WellsCTI\ATP-6603.txt

PCNC1100M3-II R3.12 SRev 3.4c

Machine OK

RESET

Clear FINISH CUTTING - READY FOR NEXT

G-Codes M-Codes Simple Run Ctl-Alt-1 Comp Run Alt-1 MDI Alt-2 Toolpath Alt-3 Offsets Alt-5 Settings Alt-6 Diagnostics Alt-7

X/Y Probe Ctl-Alt-8 Z Probe Alt-9

©2008 Tormach® LLC. All rights reserved.

start Mach3 CNC Controller EG 2:50 PM

# What Software Do I Want?

## *Part 2*

- Freeware

- Heeks CAD/CAM
- G-Simple
- FreeMill

***All of these programs create G-code text files, may have their own modelers built in and most allow you to import 2D and 3D geometry.***

- Commercial

- Dolphin CAD/CAM
- CamBam
- Vizion (ArtCam)
- OneCNC
- VirtualGibbs
- **MasterCAM**
- BOBCAD-CAM
- RAM
- MeshCAM
- VisualMill
- TurboCADCAM
- DeskCNC
- SheetCAM
- OneCNC
- SprutCam
- EdgeCAM
- ArtCAM

# What Software Do I Really Need?

## *Part 3*

- CNC machining software has great value.
- The code generated can be analyzed, optimized, and with some packages even linked back to your 3D model.
- However, since the control program doesn't care what or **WHO** created the G-code txt file, can we do it ourselves?

# What Software Do I Really Need?



- If you are going to hand code your own G code then you are mainly going to be using drills and flat end mills.
  - I have programmed ball mills but there are a lots of code for small flat surfaces.
- You are going to be working mainly in the 2D not 3D.
  - Think plates with holes or very simple geometry.

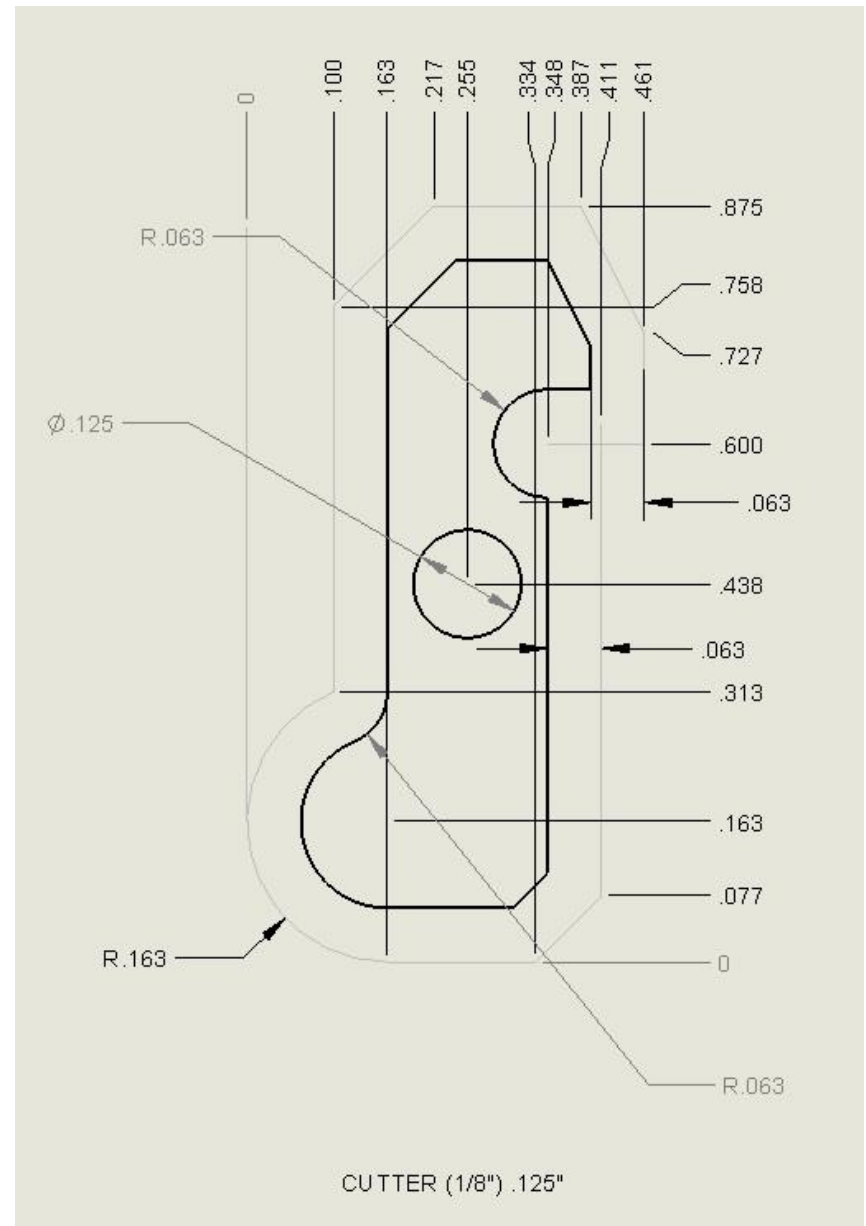
*Huge caveat, get it?*

# How I Generate G-Codes

- SolidWorks
- Draftsight
  - Formerly DWG Editor, formerly Auto-something
- StickFont
- CNC Mod

# SolidWorks

- Two methods:
  - Start a new drawing with the addition of “CNC” to the title.
  - Open existing drawing and add a “CNC” Detail.
- Offset milled edges by half cutter radius.
  - Add an explicit note on the drawing saying what cutter size and shape you plan on using.
- Use ordinate dimensions.
- Choose zero and orientation carefully.





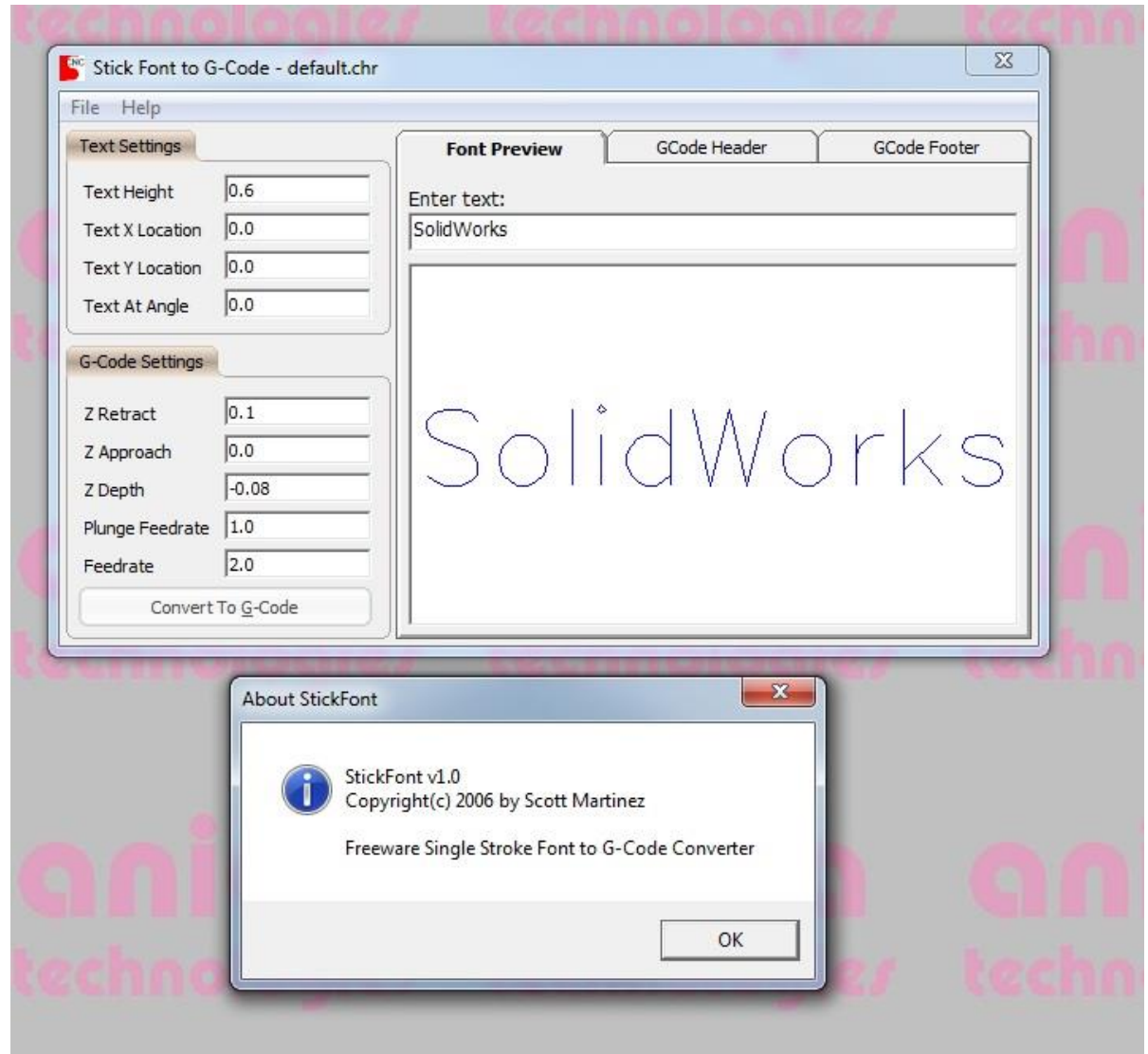
# Draftsight

- Occasionally, I have to go home to my wife and kids but I still need to program parts.
  - They seem to want to see me every day?
- Because I lose linkage from the 3D model as I program the G-codes, I can lose the linkage a little earlier and export a 2D drawing in DXF or DWG format.
- Again, I use offsets to create my tool path and ordinate dimensions.
- The biggest benefit of going this route is I can use virtually any computer including my Linux machines or really low end laptops.



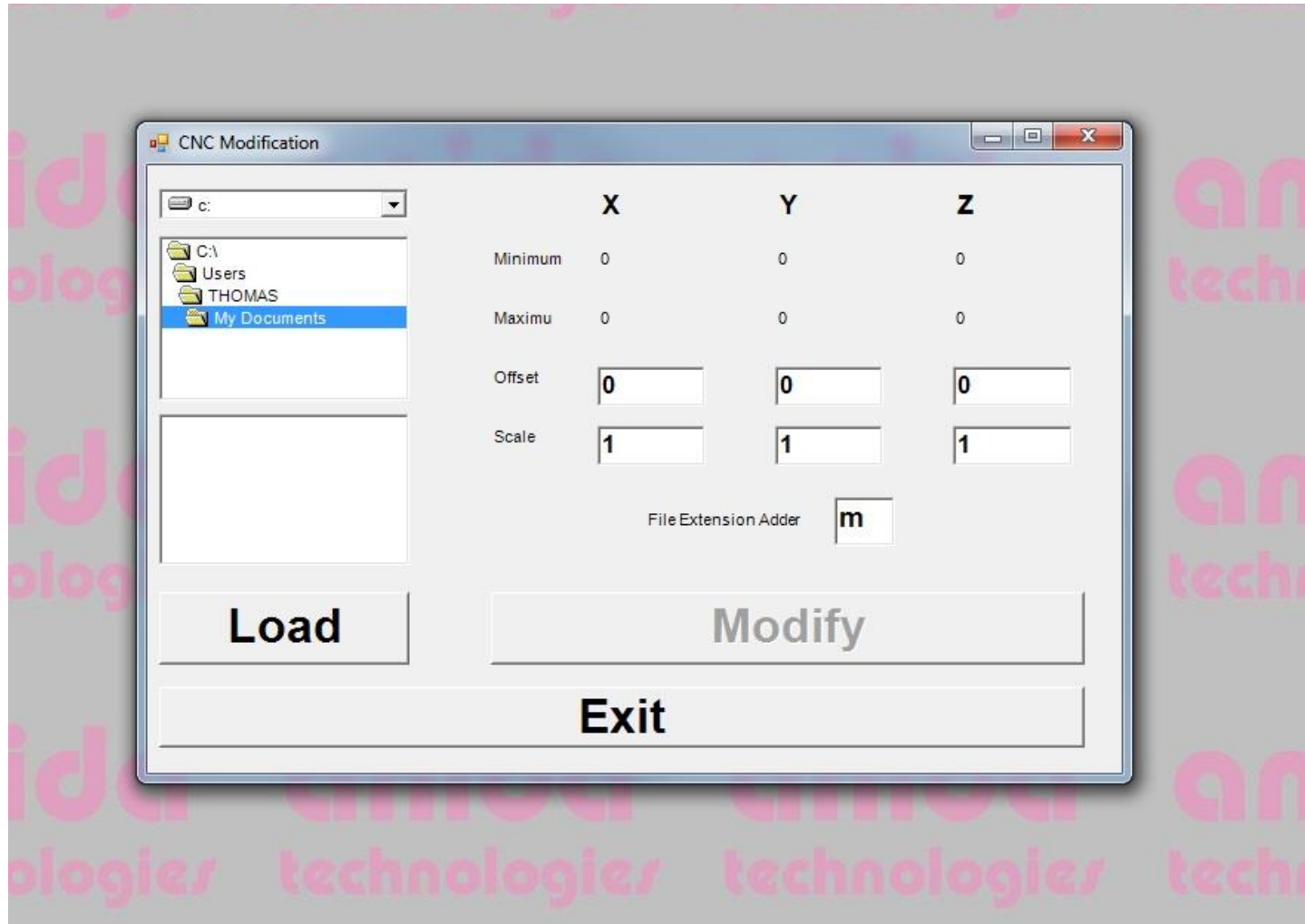
# StickFont

- Creates G Codes for text with single “stick” lines.
- Freeware program that is ultra simple to use.
- Now available in v2.0
- Lots of fonts now available.
- You can create your own font.



# CNC Mod

- Anida Technologies internally written program in Visual Basic 6.
- Recently updated in Visual Basic.NET
- Analyzes G code travel limits and allows you to offset and / or scale all the G code (including negative scales for mirroring).



*This is how I array parts.*

# (Comment your G Code)

- At the top of the program:
  - (PRIME 7)
  - (P7P-165 Revision -)
  - (Terminal Strip Mount)
  - (.125" ball mill)
  - (TAA 8/18/09)
  - (CENTER AT BOTTOM LEFT TOP OF MATERIAL - HANGING TO RIGHT OF CLAMP)
  - (SACRIFICIAL OR CLEAR ON BOTTOM)
  - (REVISED AFTER FIRST RUN)
  - G90 G80 G49 G20
  - NG00Z.1
  - S3000M3
  - GX-.0625Y-.0625
  - G00Z0
- At the start of each feature:
  - (CENTER AREA)
- At the start of each array position:
  - (\*\*\*\*\* Part 2\*\*\*\*\*)

# Another G Code Sample

(Air Wrench - small)

(.180" end mill)

(TAA 9/9/09)

G90

G80

G49

G20

NG00Z.1

S3000M3

G01Z-.025

G01X0Y.530

G01X.24Y.53

G01X.24Y0

G01X0Y0

G01Z-.05

G01X0Y.530

G01X.24Y.53

G01X.24Y0

G01X0Y0

G01Z-.075

G01X0Y.530

G01X.24Y.53

G01X.24Y0

G01X0Y0

G01Z-.1

G01X0Y.530

G01X.24Y.53

G01X.24Y0

G01X0Y0

G01Z-.125

G01X0Y.530

G01X.24Y.53

G01X.24Y0

G01X0Y0

G01Z-.15

G01X0Y.530

G01X.24Y.53

G01X.24Y0

G01X0Y0

G01Z-.175

G01X0Y.530

G01X.24Y.53

G01X.24Y0

G01X0Y0

G01Z-.2

G01X0Y.530

G01X.24Y.53

G01X.24Y0

G01X0Y0

G00Z1

NM09

NM01

NM06

NM30

(END)

# This Isn't a Machining Talk But..

- Like playing pool, remember where you leave the cue ball after your shot.
  - Where the tool is left at the end can help or hinder the part removal or getting ready for the next part.
- Clamp the material.
  - Duh...
- Beware running the tool into the part or clamp.
  - It's going to happen multiple times but it never feels good to destroy a new end mill or your favorite drill bit.

# All G Codes

*Just because  
you see a code  
here, your CNC  
may not use it!*

- G00 rapid move
- G01 straight line cut
- G02 clockwise arc cut
- G03 counterclockwise arc cut
- G04 program dwell
- G09 exact stop (non-modal)
- G10 set tool offset value
- G12 clockwise circle cut
- G17 XY plane for arcs
- G18 XZ plane for arcs
- G19 YZ plane for arcs
- G20 set inch input
- G21 set millimeter input
- G22 stored stroke limit on
- G23 stored stroke limit off
- G25 program copy
- G27 reference point check
- G28 return to machine zero
- G29 return from machine zero
- G40 cancel tool radius compensation
- G41 compensate tool radius to left
- G42 compensate tool radius to right
- G43 add tool length compensation
- G44 subtract tool length compensation
- G49 cancel tool length compensation
- G60 unidirectional approach (rapid)
- G61 Exact stop on (modal)
- G64 exact stop off (modal)
- G70 bolt hole circle
- G71 bolt hole arc
- G72 bolt hole line
- G73 high speed peck drill
- G74 reverse tapping
- G76 boring w/ spindle orient
- G77 back boring
- G80 cancel spindle cycle
- G81 drill (feed in, rapid out)
- G82 spot face (G81 with dwell)
- G83 peck drilling (fixed peck)
- G84 forward tapping
- G85 boring (feed in, feed out)
- G86 boring (feed in, stop, rapid out)
- G87 boring (feed in, stop, manual out)
- G88 boring (G87 with dwell)
- G89 boring (G85 with dwell)
- G90 absolute positioning
- G91 incremental positioning
- G92 program universal coordinates
- G94 set feed rate to in/min
- G95 set feed rate to in/rev
- G98 return to initial point (spin cycles)
- G99 return to R plane (spin. cycles)
- G100 cancel high speed cutting
- G101 high speed cut (sequence mode on)
- G102 high speed cut (memory mode on)

# All M Codes

- M00 program stop with reset
- M01 optional stop
- M02 end of program
- M03 spindle on clockwise
- M04 spindle on counterclockwise
- M05 spindle stop
- M06 tool change
- M07 spray mist on
- M08 flood coolant on
- M09 coolant off
- M18 spindle neutral
- M19 spindle lock on
- M20 spindle lock off
- M21 fourth axis clamp
- M22 fourth axis unclamp
- M30 end of program
- M46 clear spindle at program end
- M48 feedrate override, sw active
- M49 feedrate override, sw inactive
- M94 mirror image off
- M95 mirror image on
- M98 go to subprogram
- M99 return to main program

***Just because  
you see a code  
here, your CNC  
may not use it!***

# Farewell

- If you have any questions about G-Codes, please let me know at [tallsup@anidatech.com](mailto:tallsup@anidatech.com).
- Connect with me at LinkedIn
  - Thomas Allsup

Thanks

**anida**  
technologies