APPENDIX

Appendix 1 -G codes list

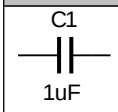
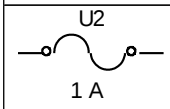
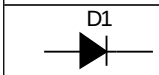
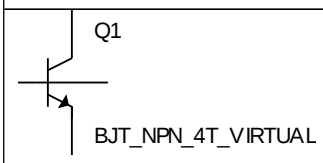
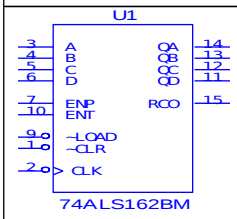
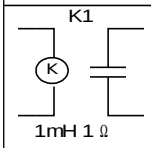
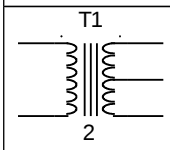
G code	DECODING
G0	Rapid positioning
G1	Linear interpolation
G2	Clockwise cirular/helical nterpolation
G3	counter Clockwise cirular/helical nterpolation
G4	Dwell
G10	Coordinate system origin setting
G12	Clockwise cirular pocket
G13	counter Clockwise cirular pocket
G15	Polar coordinate movec in G0 andG1
G16	Cancel Polar coordinate movec in G0 andG1
G17	XY plane select
G18	XZ plane select
G19	YZ plane select
G20	Inch unit
G21	Millimeter unit
G28	Return machine home(parameters5161 to 5166)
G30	Return machine home(parameters5161 to 5186)
G28.1	Reference axis
G31	Straight probe
G40	Cancel cutter radius compensation
G41	Start cutter radius compensation left
G42	Start cutter radius compensation right
G43	Apply tool length offset (plus)
G49	Cancel tool length offset
G50	Reset all scale factors to 1.0
G51	Set axis data input scale factors
G53	Move in absolute machine coordinate system
G61	Exact stop mode
G64	Constant velocity mode
G73	Canned cycle –drilling-fast pulllback
G80	Cancel canned cycle mode
G81	canned cycle-drilling

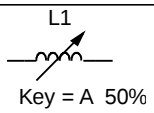
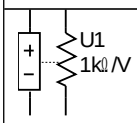
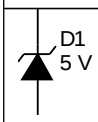
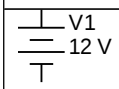
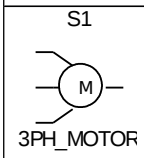
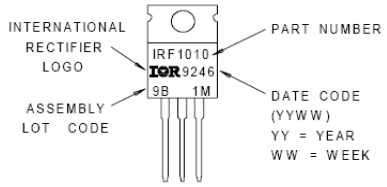
G82	***canned cycle-drilling with dwell***
G83	***canned cycle-peck drilling***
G84	***canned cycle-right hand rigid taping***
G85	***canned cycle-boring,no dwell,feed out***
G86	***canned cycle-boring ,spindle stop,rapid out***
G87	***canned cycle-back boring***
G88	***canned cycle-boring ,spindle stop,manual out***
G89	***canned cycle-boring,dwell,feed out***
G90	***Absolute distance mode***
G91	***Incremental distance mode***
G92	***Offset coordinates and set parameters***
G92.1	***Reset G92 offset and parameters***
G93	***Inverse time feed mode***
G94	***Feed per minute mode***
G95	***Feed per revolution mode***
G98	***Initial level return after canned cycles***
G99	***R –point level return after canned cycles***

Appendix 11- M codes list

M code	*DECODING*
M0	***Program stop***
M1	***Optional program stop***
M2	***Program end***
M3	***Rotate spindle clockwise***
M4	***Rotate spindle counterclockwise***
M5	***Stop spindle rotation***
M6	***Tool change***
M7	***Mist coolant on***
M8	***Flood coolant on***
M9	***All coolant off***
M30	***Program end and rewind***
M47	***Repeat program from first line***
M48	***Enable speed and feed override***
M49	***disable speed and feed override***
M98	***Call subroutine***
M99	***Return from subroutine/repeat***

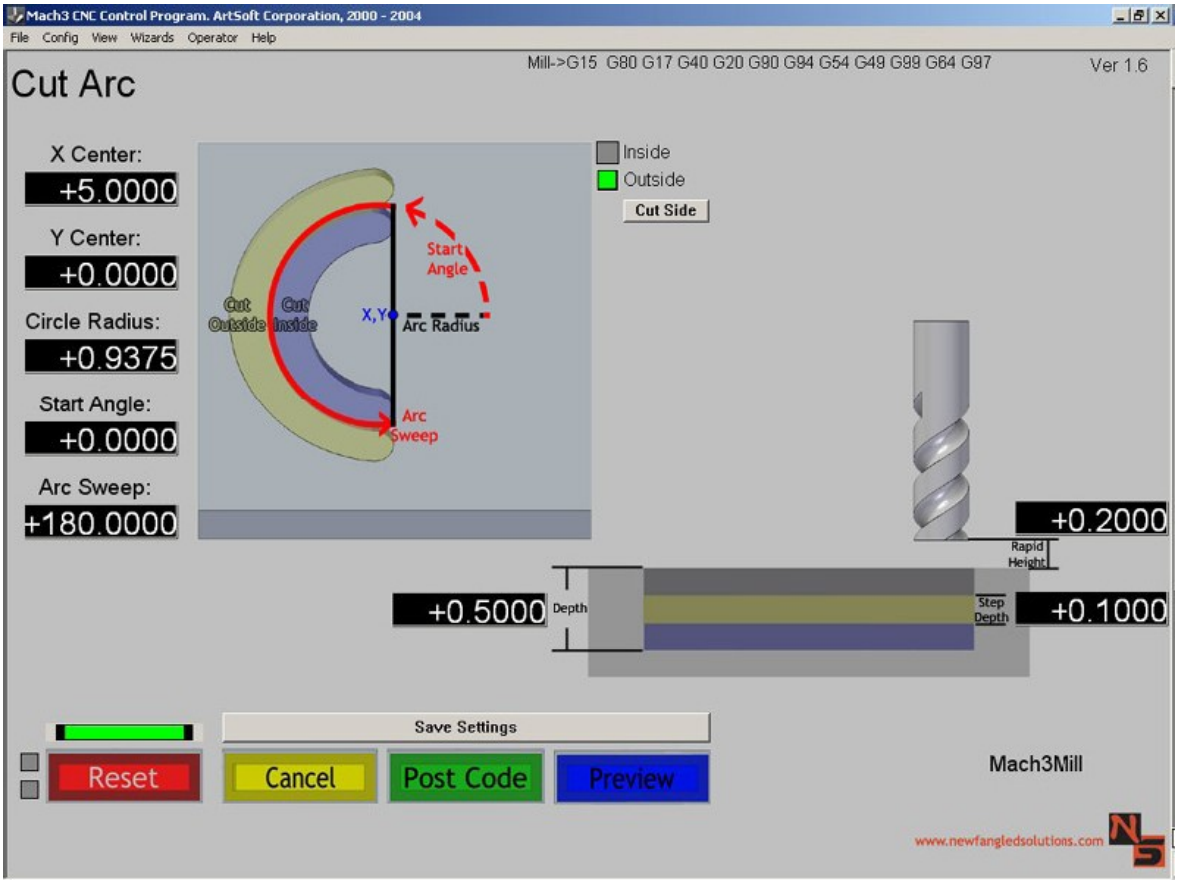
Appendix 111-ELECTRONICAL SYMBOLS

SYMBOL	TERMINALOGY
	***Capacitor***
	***Fuse***
	***Diode***
	***Transistor***
	***IC component 74LS….***
	***Relay***
	***Fixed resistor***
	***Transformer***

	***Variable resistor***
	***Voltage controlled resistor***
	***Zener diode***
	***DC power source***
	***Ground***
	***Three phase motor***
	IRF540N

Appendix 1V - Operations Screen Definitions

Cut Arc



Arc Radius

The radius of the arc to be cut.

Arc Sweep

A value in Degrees, which represents how much of an arc is desired.

Cut Side (Inside)

This selection causes the tool to cut on the inside of the user specified Arc. This makes the

Outside radius of the cutout the user specified value.

Cut Side (Outside)

This selection causes the tool to cut on the outside of the user specified Arc. This makes the

Inside radius of the cutout the user specified value.

Rapid Height

Distance above the work surface for any rapid moves..

Start Angle

The angle created between the defined X axis and the line intersecting the user specified circle

Where the first hole will be placed. Zero degrees is defined to be the 3:00 O'clock position.

Step Depth

Depth of material removed per tool pass.

Total Depth

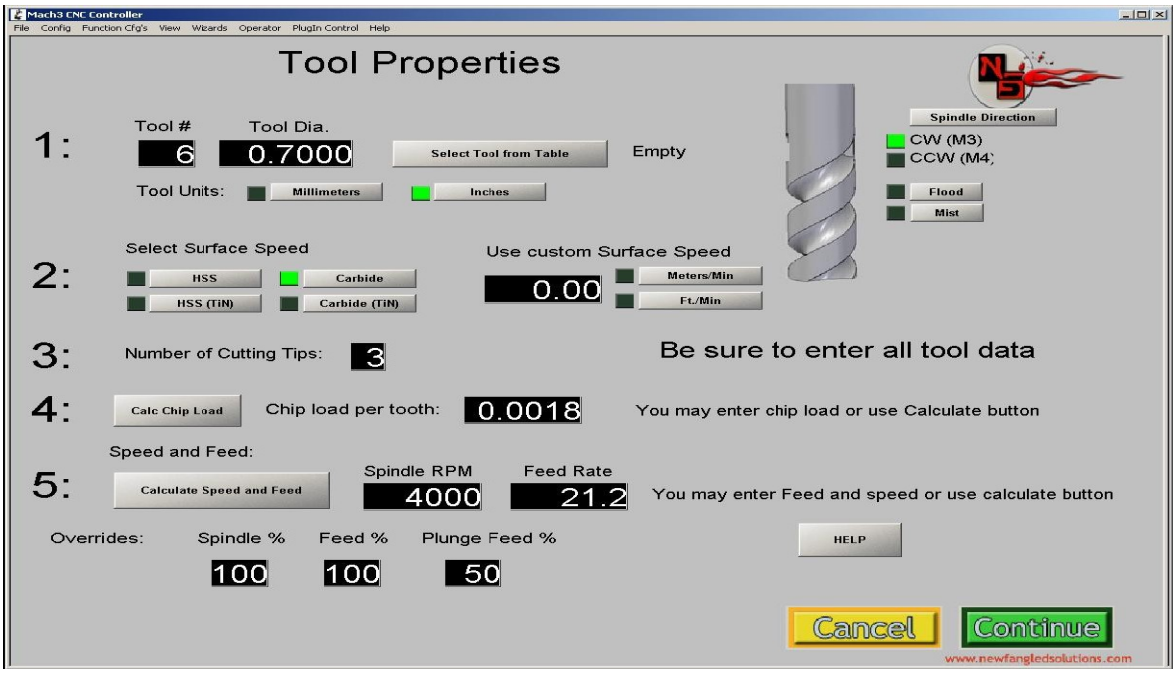
Final depth of cut. (Total amount of material removed after completing all steps or pecks.)

Interface Basics:

Essential values used throughout the Mach3 package are displayed to the user as a Digital Read Out (DRO - See Figure 5-29 below). At different points, the user will need to provide information necessary for an operation to be completed. When entering such values in this

screenset, you must select the DRO you wish to use by positioning the mouse over it and pressing the left mouse button (Left Clicking). You will notice the DRO color change upon selection. After entering the desired value, the user must then press the Enter key.

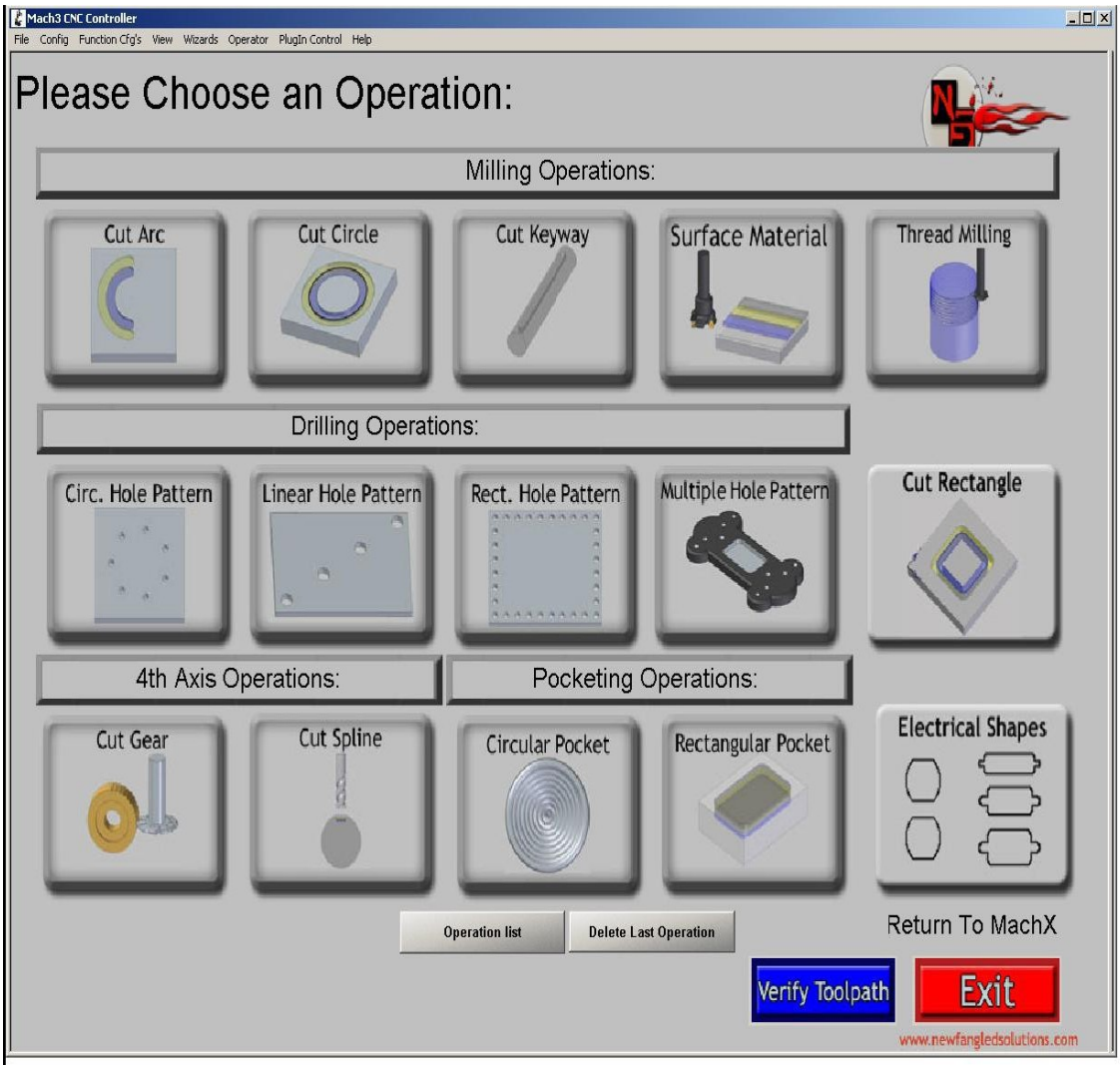
NOTE: Not pressing the ENTER key is the most common beginner mistake.



Digital Read Out Example

If process requires a finish pass, recommend leaving yourself some room. Create pockets, holes or inside cuts **slightly undersize**, make your surfacing or outside cuts **slightly oversize** and then simply run the wizard a second time, changing the required settings to your desired finish values. Since your settings should be saved, it's quite simple!

Included Operations:



Milling operations

Milling Operations

Cut Arc – Commonly used for fillets, or milling curves

Cut Circle – Commonly used for cutting circles, or circular groves for oil rings etc.

Cut Keyway -- Commonly used to mill Keyways in round sock

Surface Material – Commonly used to initially face material before milling

Thread Milling – Commonly used to create internal or external threads on various parts

Cut Rectangle- Commonly used to square off a piece of stock

Hole Patterning Operations

Circular Hole Pattern – Commonly used to position holes along a circular pattern

Linear Hole Pattern - Commonly used to position holes in a linear pattern

Rectangular Hole Pattern - Commonly used to position holes along a rectangular pattern

Multiple Hole Pattern – Commonly used to “drill” in up to 20 predetermined locations

Pocketing Operations

Circular Pocket – Commonly used to create a circular recessed pocket

Rectangular Pocket - Commonly used to create a rectangular recessed pocket with corner

radius of the tool diameter

4th Axis Operations

Cut Gear – Commonly used to cut gears using a gear cutting tool

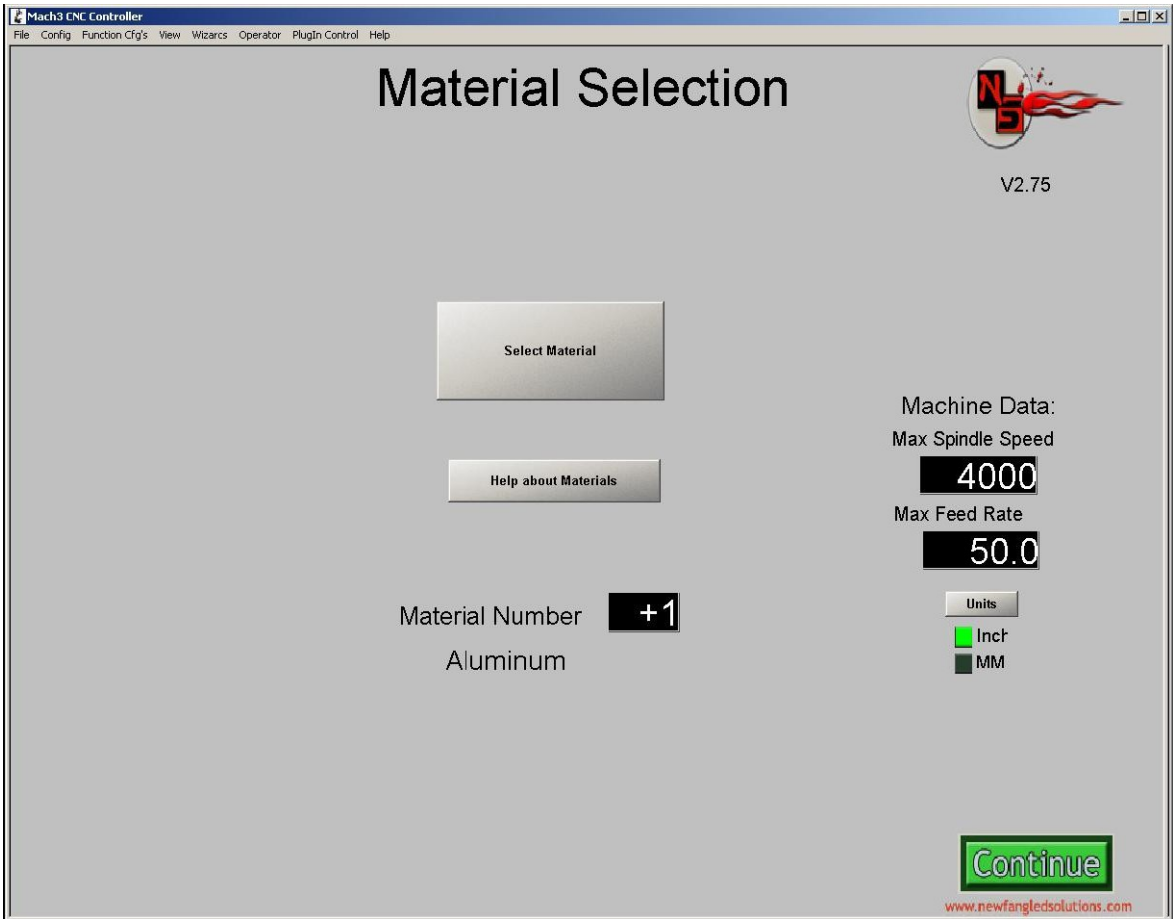
Cut Spline - Commonly used to create a spline using a spline cutting tool

Special Operations

Electrical Shapes- Cuts panel openings for common electrical devices.

Material Selection

This is the first screen to appear when the Newfangled wizard is opened. There are two sections of the screen, Material data and fixed machine data. Once machine data is entered the wizards will save it for all future runs. It may be altered as needed.



Material selection

Machine Data: Max Spindle Speed

Maximum Spindle RPM for the Mill (One Time Entry for Most Users)

Machine Data: Max Feed Rate

Maximum Feed Rate for the Mill (One Time Entry for Most Users)

Current Material Options

The material to be machined is selected from a list box that will appear when the “Select Material” button is pressed. Simply click on the desired material and then the “OK” button. Material and tool data is taken from a table that can be modified by the user. The table is a

simple text file, stored at Addons\Newfangled\Material.txt.. The table may be edited to add materials, or to alter the settings for cutting speed.

Each entry in the table has 5 parameters, separated by commas. The first is the material name, followed by the Cutting speed (surface feet per minute) for each tool type. The default file is as follows:

Material Name,Hss,HSStin,Carbide,CarbideTin

Aluminum,500,650,850,1000

Soft Steel,95,130,250,250

Medium Steel,75,115,215,275

Hard Steel,25,65,125,215

Brass,230,325,550,700

Bronze,200,275,450,550

Soft Cast Iron,90,110,225,270

Hard cast Iron,25,65,130,200

Other,100,100,100,100

Aluminum

Appropriate for general milling, also appropriate for soft metals, wood, and some

plastics such as PTFE, PVC, and UHMW

Steel (Soft)

Appropriate for Common Steels such as Hot and Cold Rolled 1018

Steel (Medium)

Appropriate for Harder Steels such as 4140, O1, D2, or other Tool Steels

Steel (Hard)

Appropriate for Hardened Tool Steels, and Stainless Steels

Brass

Appropriate for Brass, medium metals, and many plastics susceptible to melting during

cutting such as Polypropylene, Polyethylene, and Acrylic

Bronze (Hard)

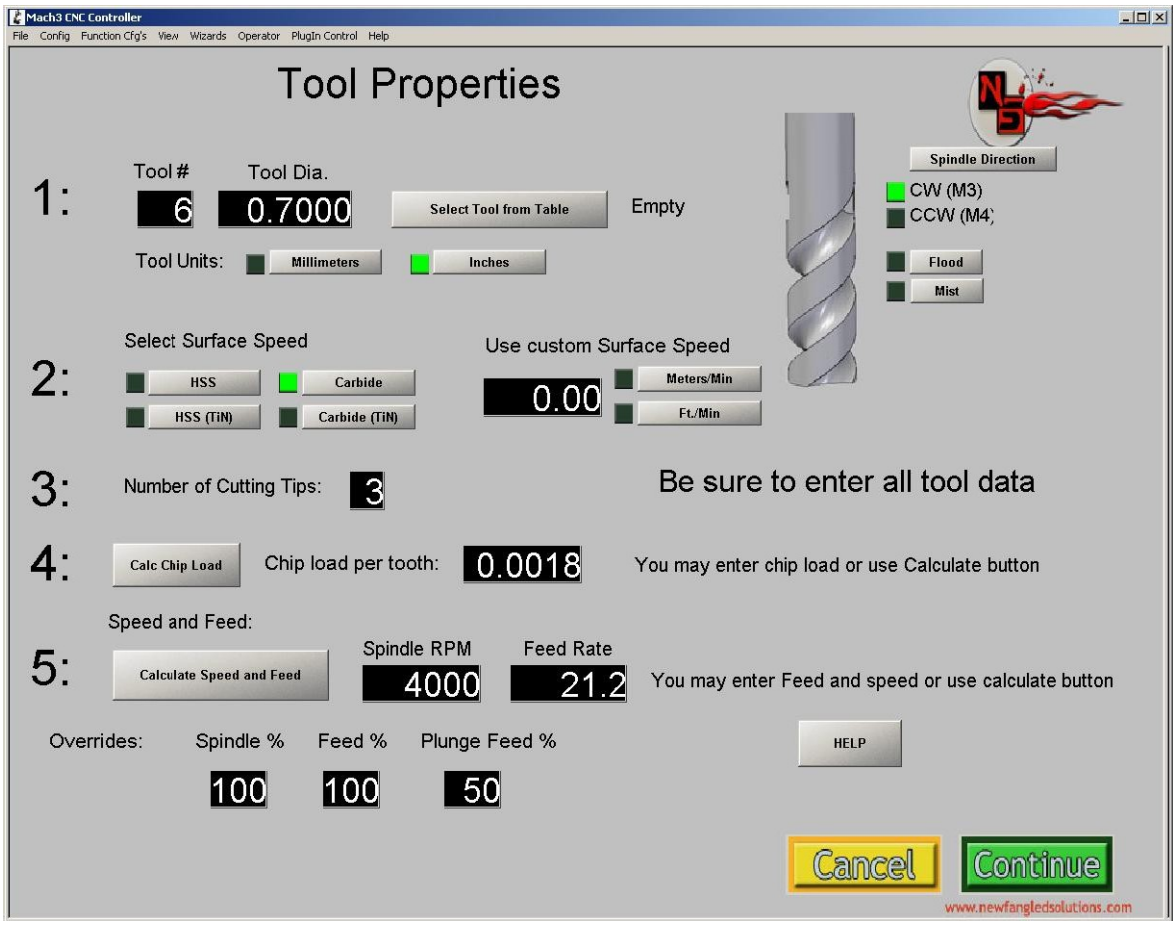
Appropriate for Bronze and Coppers

Cast Iron (Soft)

Cast Iron (Hard)

Note that this screen also indicates the Version number of the current Newfangled wizard.

Tool Properties



Tool properties

It is likely that a complex part program will consist of several steps, often performed using different tools. Therefore the tool Property screen will be display before each function screen. Be sure to enter all the tool data following the 5 steps indicated. Most of the properties have two ways to select, they may be directly entered into the DROs, or they may be calculated by the wizards.

1. **Select a tool number and diameter.** You may simply enter the tool diameter into the DRO. Do not use the tool table built-in to Mach leave the tool number as 0. Do use the tool table you may select the tool number and diameter from a list box that will be displayed if press the "Select Tool from Table" button.

2. Select Surface Speed. Pressing one of the 4 tool type buttons will select a surface speed based on the material chosen in the Material Screen and the data stored in the Material table. If want to enter a special surface speed enter it in the DRO and select the appropriate units.

3. Enter the number of cutting tips, or flutes.

4. Calculate the Chip Load. This is the amount of material to be removed by each tip, or tooth, as the tool revolves. The wizard will calculate a value based on the tool diameter and flutes, or may enter a special number.

5. Calculate Speed and Feed. The calculate button will make its calculations based on the previous entries for diameter, surface speed and chip load. If you want to force a specific value you may enter it into the DRO instead of pressing the Calculate button.

Overrides: Feed %

Feed Rate override allowing user to customize output values to their specific equipment. Entered value of 100% or less will decrease Feed Rate to the entered percentage of the otherwise calculated value. (Calculated Feed * override% = new Feed Therefore entering 80%

will change a calculated Feed Rate of 14.4 in/min to a value of 11.5 in/Min)

Overrides: Spindle %

Spindle speed override allowing user to customize output values to their specific equipment. Entered value of 100% or less will decrease spindle RPM to the entered percentage of the otherwise calculated value. (Calculated RPM * override% = new RPM Therefore entering 80%

will change a calculated spindle RPM of 4000 to a value of 3200)

Percent Plunge Feed Rate

Plunge Feed Rate override allowing user to customize output values to their specific needs. Entered value of 100% or less will decrease Plunge Feed Rate to the entered percentage of Feed Rate value. (Calculated Feed

Rate * override% = new Plunge Feed Rate) Therefore entering

50% will change the Plunge Feed Rate from the Calculated Feed Rate of 14.4 in./min. to a value of 7.2 in./min.) This screen also lets you select to use coolant and to set spindle direction

Flood

User selected Flood coolant option

Mist

User selected Mist coolant option

Spindle Direction: CCW (M4)

Spindle turns in a clockwise direction

Spindle Direction: CW (M3)

Spindle turns in a clockwise direction

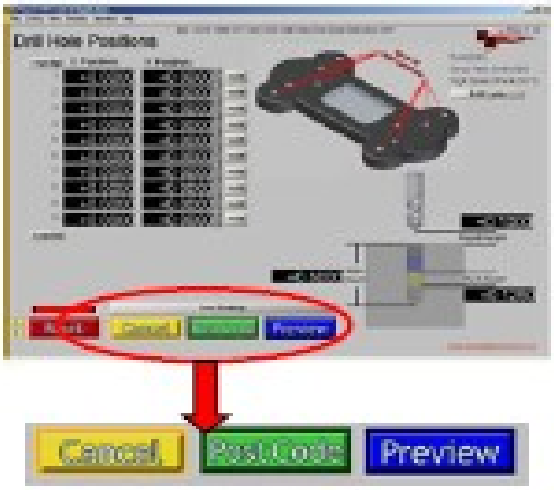
Tool Units: Inches

User selected Units for general tooling

Tool Units: Millimeters

User selected Units for general tooling

Navigating through Operations:



Navigating through Operations

Cancel

Returns user to Select Operation screen

Post Code

Appends G code from selected operation to G code file

Preview

Gives user a view of the tool path and G code generated by the selected operation

Verify Tool path

Verify Tool path

Gives User a view of the toolpath(s) created by the G code contained in the G code file.

Exit

Returns user to the Run Program screen of Mach3 and loads the generted G code file.