1.0 INTRODUCTION

1.1 CopyCat is a unique wizard used with MACH3. It is not a stand alone program. This wizard will allow you to jog a machine around and create a Gcode file from the movement.

2.0 REQUIREMENTS

2.1 A licensed copy of Mach and a license for Mach3 Addons for Mill by New Fangled Solutions is required to save the generated Gcode into Mach. The program is included in the default Mach3 installation.

2.2 ArtSoft USA recommends a PC with at least a 1 GHZ processor and 1024 x 768 pixel resolution screen to run Mach3. A keyboard or equivalent (MPG, Shuttle Pro) for use with Mach is also required. You can run CopyCat with a less able PC but the performance will suffer.

2.3 A video camera which will utilize the Video Window available in Mach3. (You can use some type of pointing device (laser, toothpick, etc) but some of the plug in intent will be lost.)

3.0 HOW IT WORKS

3.1 CopyCat monitors xyz axis movement via Mach. The movement is displayed in it's DRO's. You jog around with the keyboard until you are at a point, then click the button for the kind of move you want, namely a rapid, feed, or Arc. CopyCat reads the DROs and builds the code based on point selection.

3.2 That one records every move you make with a jog. CopyCat only records the points you select, so if you have to jog back and forth until you hit an exact point it will only record that point. This is very different from the Jcode plugin which records all the moves.

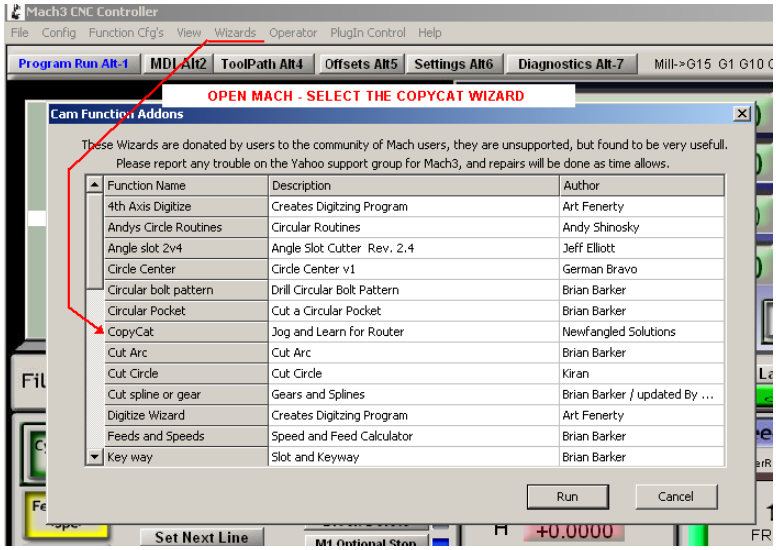
3.3 This wizard always writes to the same file, C:\Mach3\GCode\teach.tap This allows you to add to an existing file. If you want to start a new file be sure to click the 'NewTeach File' button.

3.4 Inside the wizard you are free to move the machine around with the normal jog commands, as well as by entering commands into an MDI box. Your movements are only saved and made part of the G code file when you press one of the action buttons.

3.3 Specific examples of using CopyCat are provided in Appendix -----.

4.0 The CopyCat Plugin

4.1 Open Mach3 and click the Wizard tab and from the pull down menu select CopyCat.



5.0 MAIN SCREEN

5.1 The main screen is divided into logical areas as shown in Figure 5.1 below. You have DRO's, basic commands, feeds, type of move selection, Gcode text screen, hole drilling and cut hole operations, MDI line for manual code input, and jogging controls. You can also exit and reset Copycat.

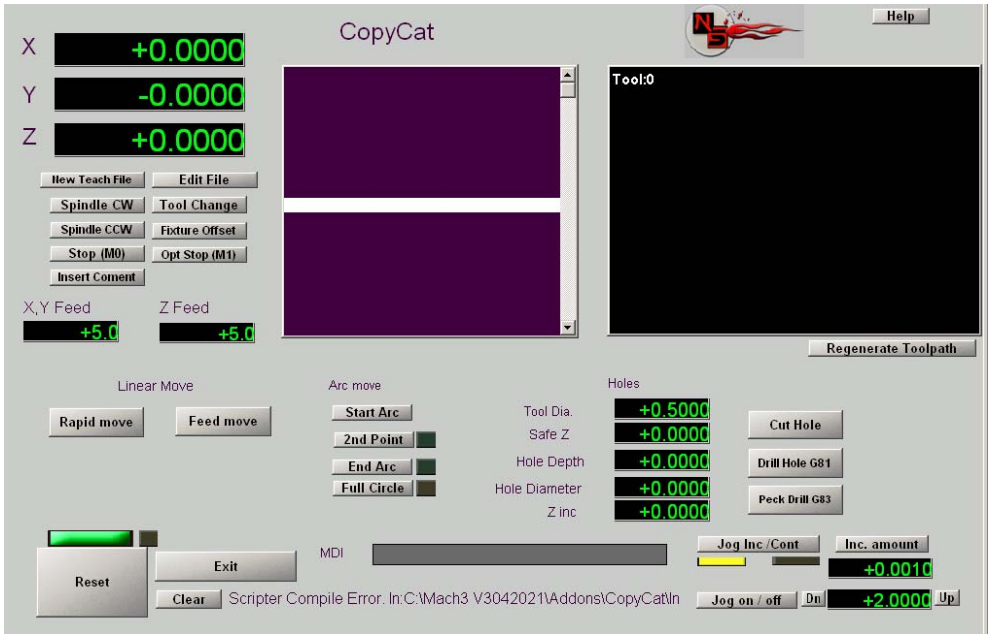


FIGURE 5.1

5.2 The XYZ axis DRO's indicate table movement in real time. You can jog to a point using the pc keyboard keys (or any similar device). You can also manually input locations into the DRO's by clicking on the box, adding a value, and hitting the enter key. Point indication is discussed in "Video Camera" Appendix ----------

FIGURE 5.2

5.3 The following are brief descriptions for each of the buttons below the DRO's. Note that some of the buttons will open a pop up window for data entry as shown in figure 5.3 below.

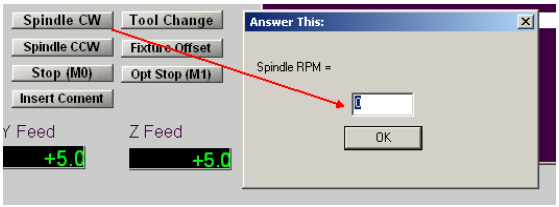


FIGURE 5.3

5.4 Codes are automatically added to the Gcode text window along with associated data upon clicking with the left mouse button as shown in figure 5.4 below.

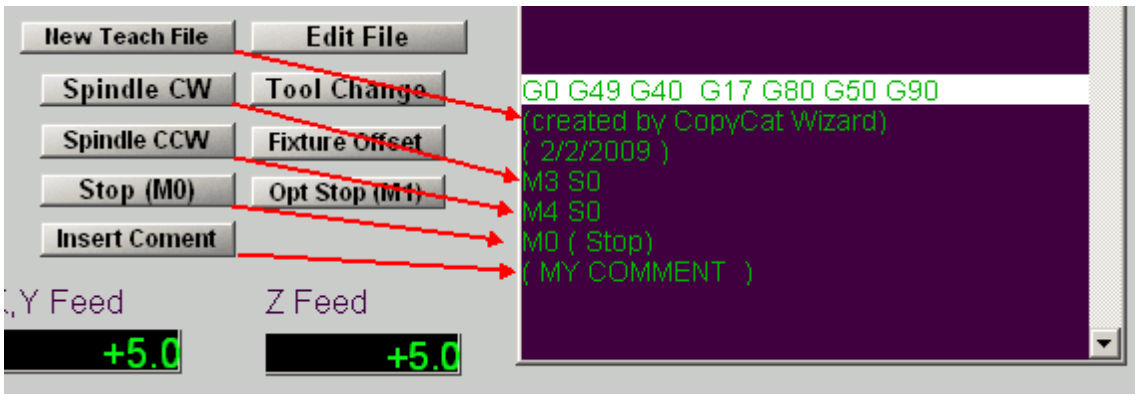


FIGURE 5.4

New Teach File – You need to click this to create a file. You may additionally load an existing Gcode using the File > Load G-Code Tab.

Spindle CW – inserts the M3 along with entered spindle RPM

Spindle CCW – inserts the M4 along with entered spindle speed RPM

Stop (MO) – inserts an M0 m code

Insert Comment – put any comment you like by typing the text into the pop up Window box and click. Parentheses are automatically add before and after the text. Click the OK box when done.

5.5 *Edit File* – allows editing of the file using your default MACH3 editor at any time. Should you make an error in the process you can delete, add, change any line of the code. Remember that you may chang move locations and affect the program run if you insert errors. You may desire to alter the initialization string, add more file descriptions, or move the comment to the associated code line. Use File>Save to implement the changes.

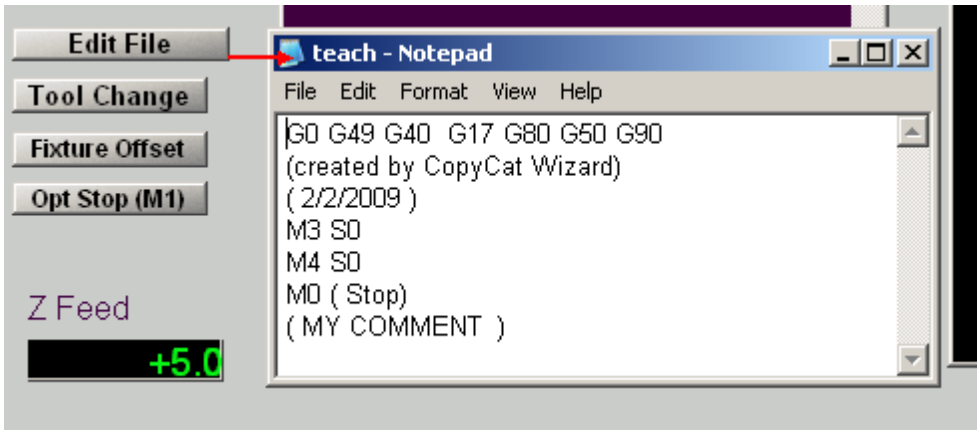


FIGURE 5.5

5.6 The remaining three buttons allow for tool, fixture offset, and a stop. The tool and fixture offset are based on current defined Mach3 values.

FIGURE 5.6

Tool Change – inserts M6 code including the entered tool number

Fixture Offset – Inserts the input value from the pop up window

Opt Stop (M1) – inserts the optional program stop code

5.7 The following is a description of how a rapid and feed move is coded. Note the XYZ point and feeds currently shown in figure 5.6. Only definitions are noted here. See "CopyCat Quick and Dirty" Appendix ---- for a how to example of using CopyCat.

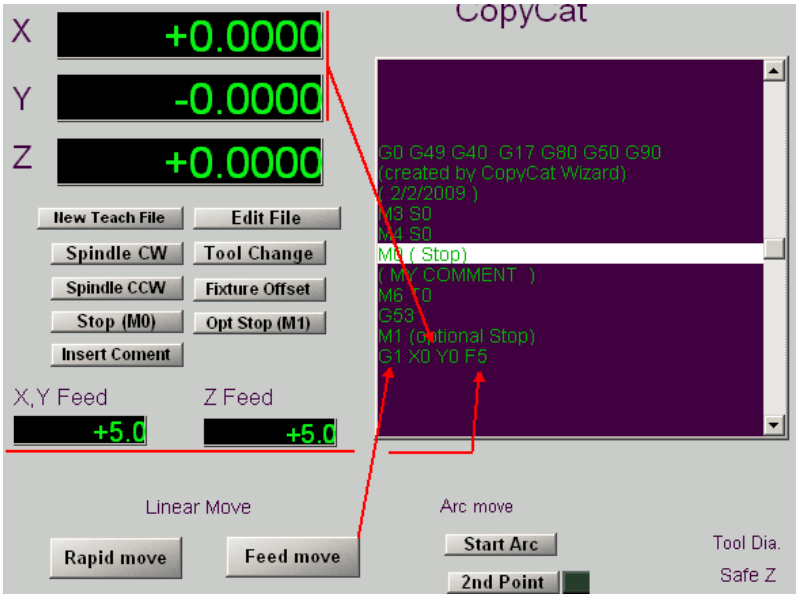


FIGURE 5.6

5.7.1 RAPID MOVE

You will probably want to define a starting point at XYZ=0. When you click the Rapid move button, a rapid positioning code G0 along with the current DRO values will be inserted into the text window. The rapid feed rate will be as currently defined in Mach 3. No F is added to the line of code.

CopyCat will remember this point. You can jog anywhere and anyway until you arrive at another / next point, and then by clicking the rapid move button, another G0 will be implemented to that point.

5.7.2 FEED MOVE

The feed move works the same way as the rapid move only a linear interpolation code G1 is inserted into the file along with the limiting axis feed rate.

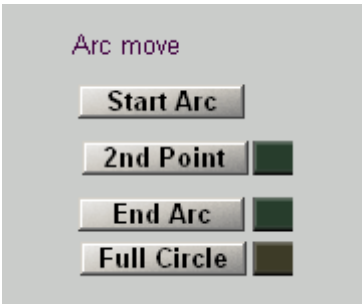
You can change the feed rates as desired for any move at any time.

5.7.3 ARC & FULL CIRCLE MOVE

CopyCat will create the proper coding for clockwise / counter clockwise circular interpolation G2 & G3 coding and insert it into the file. You will need to have a G0 or G1 move prior to starting an arc. You can continue with another arc without a lead in move.

The steps to create an Arc are as follows and shown the following figure:

1. A lead in move is made to the start point “A” of the arc. The lead in should be longer than the tool diameter.
2. Now click the Start Arc box (a box turns green and you are now prompted for point “B”)
3. Move to point “B” which is approx half the way along the arc
4. Now click the 2nd Point box (box turns green and you are now prompted for point “C”)
5. Move to point “C” which is the end of the arc
6. Now click the End Arc button



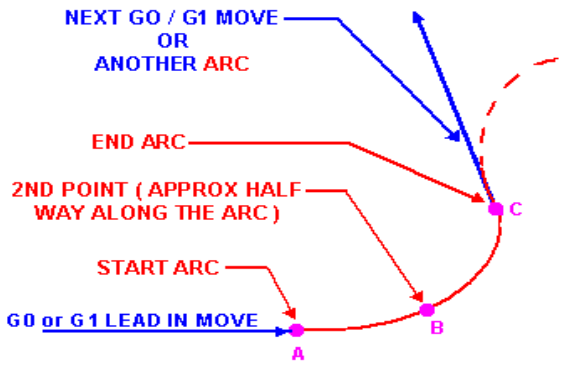
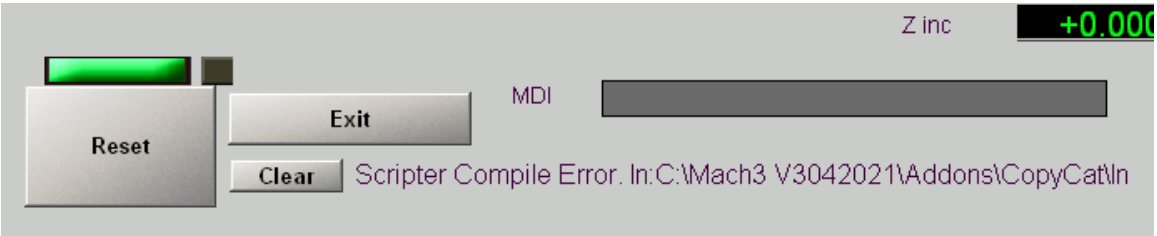


FIGURE 5.7.3

Notice that the Gode describing the arc is inserted into the file. You can now move to define another adjoining arc or point away from the arc.

A full circle move may be generated.. Press "Start Arc' to remember this point. Next. If the 'Full Circle' button has been pressed, and its LED is on, then a full circle will be generated. Otherwise an arc from the start to end points is generated.

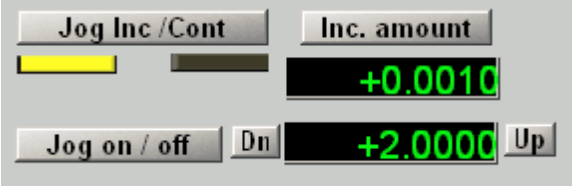
5.8



5.9 MOVING TO A POINT

Movement Buttons

In general, you may move around the machine freely with any of the normal jog functions, then when you are at the desired point you press a movement button when you are at a point you want in your program.



5.8 HOLES

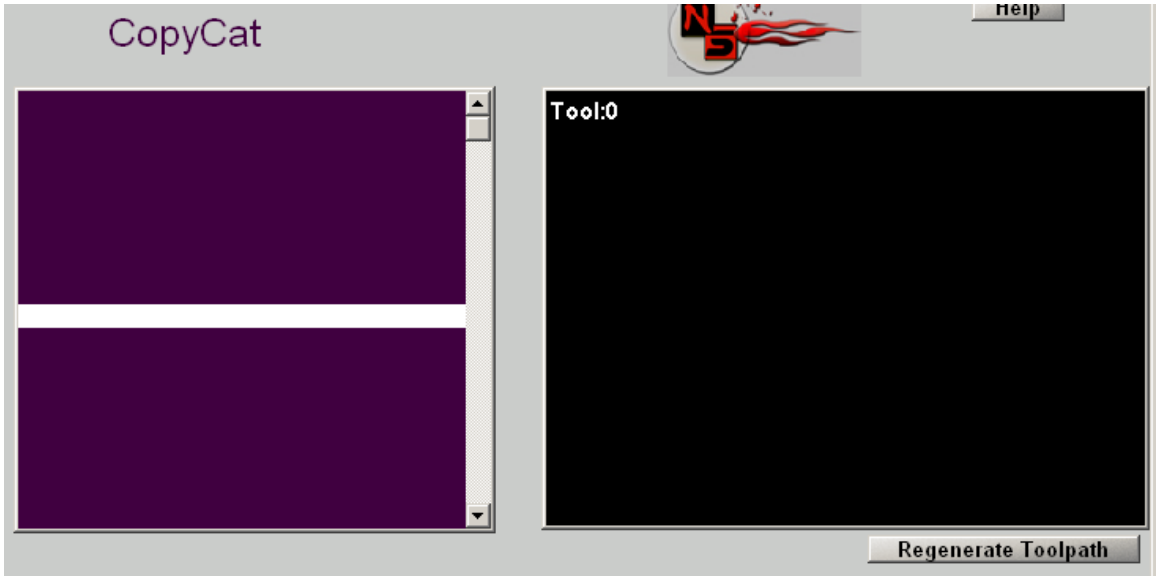
Holes

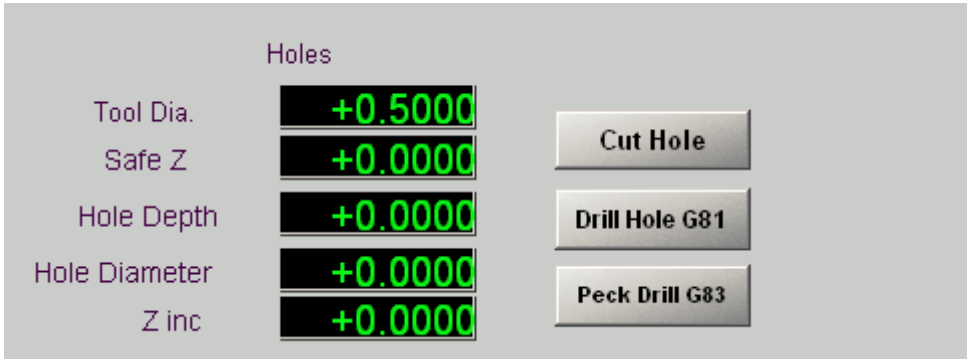
Move to the center of any hole and select a Rapid or Feed move. Note that this move should be above the work at a safe height. Then select the kind of hole.

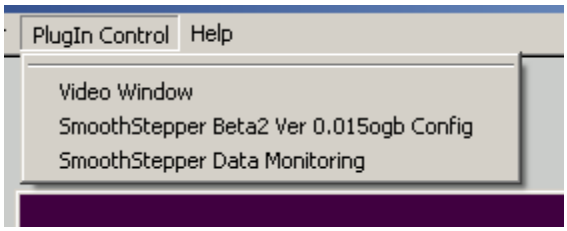
Hole will mill a hole by making a circular cut. The tool will spiral down to the requested depth, then make a full circle at that depth to fully clear the hole. Be sure you are using a cutter that can be plunged!

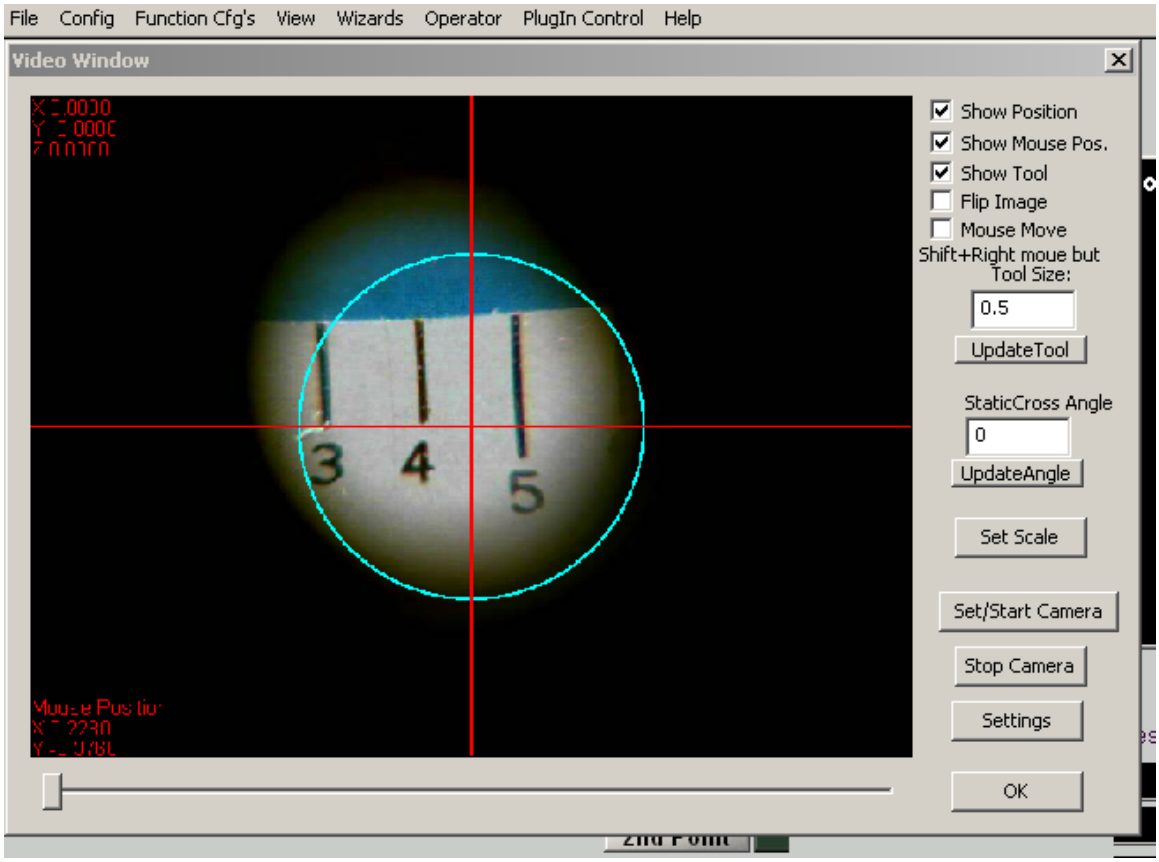
Drill Hole G81 will make a plunge drill operation to the selected depth.

Peck Drill G83 will make a peck drill cycle to the selected depth in steps of Zinc.


CopyCat
Help
Tool:0
Regenerate Toolpath


Holes
Tool Dia.
+0.5000
Safe Z
+0.0000
Hole Depth
+0.0000
Hole Diameter
+0.0000
Z inc
+0.0000
Cut Hole
Drill Hole G81
Peck Drill G83


PlugIn Control
Help
Video Window
SmoothStepper Beta2 Ver 0.015ogb Config
SmoothStepper Data Monitoring


File
Config
Function Cfg's
View
Wizards
Operator
PlugIn Control
Help
Video Window
Show Position
Show Mouse Pos.
Show Tool
Flip Image
Mouse Move
Shift+Right moue but
Tool Size:
0.5
UpdateTool
StaticCross Angle
0
UpdateAngle
Set Scale
Set/Start Camera
Stop Camera
Settings
OK
3
4
5

APPENDIX "A"
COPYCAT QUICK & DIRTY

APPENDIX "B"
G CODE LIST

G1 Linear interpolation
G2 Clockwise circular/helical interpolation
G3 Counterclockwise circular/Helical interpolation
G4 Dwell
G10 Coordinate system origin setting
G12 Clockwise circular pocket
G13 Counterclockwise circular pocket
G15/G16 Polar Coordinate moves in G0 and G1
G17 XY Plane select
G18 XZ plane select
G19 YZ plane select
G20/G21 Inch/Millimetre unit
G28 Return home
G28.1 Reference axes
G30 Return home
G31 Straight probe
G40 Cancel cutter radius compensation
G41/G42 Start cutter radius compensation left/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
G52 Temporary coordinate system offsets
G53 Move in absolute machine coordinate system
G54 Use fixture offset 1
G55 Use fixture offset 2
G56 Use fixture offset 3
G57 Use fixture offset 4
G58 Use fixture offset 5
G59 Use fixture offset 6 / use general fixture number
G61/G64 Exact stop/Constant Velocity mode
G68/G69 Rotate program coordinate system
G70/G71 Inch/Millimetre unit
G73 Canned cycle - peck drilling
G80 Cancel motion mode
G81 Canned cycle - drilling
G82 Canned cycle - drilling with dwell
G83 Canned cycle - peck drilling
G85/G86/G88/G89 Canned cycle - boring
G90 Absolute distance mode
G90.1 Absolute IJK mode

G91 Incremental distance mode

G91.1 Incremental IJK mode
G92 Offset coordinates and set parameters
G92.x Cancel G92 etc.
G93 Inverse time feed mode
G94 Units Per Min.
G98 Rapid Height By Z Height
G99 Rapid Height By R Height

APPENDIX "C"

G0 – G1- G2 &G3 and ARC EXPLAINATIONS

GO - RAPID MOVE EXPLAINATION

(a) For rapid linear motion, program G0 X~ Y~ Z~ A~ B~ C~ , where all the axis words are optional, except that at least one must be used. The G0 is optional if the current motion mode is G0. This will produce coordinated linear motion to the destination point at the current traverse rate (or slower if the machine will not go that fast). It is expected that cutting will not take place when a G0 command is executing.

(b) If G16 has been executed to set a Polar Origin then for rapid linear motion to a point described by a radius and angle G0 X~ Y~ can be used. X~is the radius of the line from the G16 polar origin and Y~ is the angle in degrees measured with increasing values counterclockwise from the 3 o'clock direction (i.e. the conventional four quadrant conventions).

Coordinates of the current point at the time of executing the G16 are the polar origin. ¨ l axis words are omitted.

If cutter radius compensation is active, the motion will differ from the above; see Cutter Compensation. If G53 is programmed on the same line, the motion will also differ; see Absolute Coordinates.

G01 Linear Move

(a) For linear motion at feed rate (for cutting or not), program G1 X~ Y~ Z~ A~ B~ C~ , where all the axis words are optional, except that at least one must be used. The G1 is optional if the current motion mode is G1. This will produce co - ordinated linear motion to the destination point at the current feed rate (or slower if the machine will not go that fast).

(b) If G16 has been executed to set a polar origin then linear motion at feed rate to a point described by a radius and angle G0 X~ Y~ can be used. X~ is the radius of the line from the G16 polar origin and is the angle in degrees Y~ measured with increasing values counterclockwise from the 3 o'clock direction (i.e. the conventional four quadrant conventions).

Coordinates of the current point at the time of executing the G16 are the polar origin.

It is an error if:

¨ l axis words are omitted.

If cutter radius compensation is active, the motion will differ from the above; see Cutter Compensation. If G53 is programmed on the same line, the motion will also differ; see Absolute Coordinates.

G02 & G03 Arc Move

A circular or helical arc is specified using either G2 (clockwise arc) or G3 (counterclockwise arc). The axis of the circle or helix must be parallel to the X, Y, or Z -axis of the machine coordinate system. The axis (or, equivalently, the plane perpendicular to the axis) is selected with G17 (Z -axis, XY - plane), G18 (Y -axis, XZ -plane), or G19 (X -axis, YZ -plane). If the arc is circular, it lies in a plane parallel to the selected plane.

If a line of code makes an arc and includes rotational axis motion, the rotational axes turn at a constant rate so that the rotational motion starts and finishes when the XYZ motion starts and finishes. Lines of this sort are hardly ever programmed.

If cutter radius compensation is active, the motion will differ from the above; see Cutter Compensation.

Two formats are allowed for specifying an arc. We will call these the center format and the radius format. In both formats the G2 or G3 is optional if it is the current motion mode.

Arc Center Format

In the center format, the coordinates of the end point of the arc in the selected plane are specified along with the offsets of the center of the arc from the current location. In this format, it is OK if the end point of the arc is the same as the current point. It is an error if:

¨ en the arc is projected on the selected plane, the distance from the current point to the center differs from the distance from the end point to the center by more than 0.0002 inch (if inches are being used) or 0.002 millimetre (if millimetres are being used).

The center is specified using the I and J words. There are two ways of interpreting them. The usual way is that I and J are the center relative to the current point at the start of the arc. This is sometimes called Incremental IJ mode The second way is that I and J specify the center as actual coordinates in the current system. This is rather misleadingly called Absolute IJ mod.The IJ mode is set using the Configure>State … menu when Mach3 is set up. The choice of modes are to provide compatibility with commercial controllers. You will probably find Incremental to be best. In Absolute it will, of course usually be necessary to use both I and J words unless by chance the arc's centre is at the origin.
WI~ J~ (or use G3 instead of G2). The axis words are all optional except that at least one of X and Y must be used. I and J are the offsets from the current l respectively) of the center of the circle. I and J are optional except that at least one of the two must be used. It is an error if:

¨ X and Y are both omitted,

¨ I and J are both omitted.

When the XZ -plane is selected, program G2 X~ Y~ Z~ A~ B~ C~ I~ K~ (or use G3 instead of G2). The axis words are all optional except that at least one of X and Z must be used. I and K are the offsets from the current location or coordinates -depending on IJ mode (X and Z directions, respectively) of the center of the circle. I and K are optional except that at least o
¨ X and Z are both omitted,

¨ I and K are both omitted.

When the YZ -plane is selected, program G2 X~ Y~ Z~ A~ B~ C~ J~ K~ (or use G3 instead of G2). The axis words are all optional except that at least one of Y and Z must be used. J and K are the offsets from the current location or coordinates -depending on IJ mode (Y and Z directions, respectively) of the center of the circle. J and K are optional except that at least one of the two must be used. It is an error if:

¨

¨ and K are both omitted.

Here is an example of a center format command to mill an arc in Incremental IJ

mode:

G17 G2 x10 y16 i3 j4 z9

That means to make a clockwise (as viewed from the positive z -axis) circular or helical arc whose axis is parallel to th-axis, ending where X=10, Y=16, and Z=9, with its center offset in the X direction by 3 units from the current X location and offset in the Y direction by 4 units from the current Y location. If the current location has X=7, Y=7 at the outset, the center will be at X=10, Y=11. If the starting value of Z is 9, this is a circular arc; otherwise it is a helic arc. The radius of this arc would be 5.
The above arc in Absolute IJ mode would be:

G
In the center format, the radius of the arc is not specified, but it may be found easily as the distance from the center of the circle to either the current point or the end point of the arc.

APPENDIX "D"
VIDEO WINDOW AND CAMERA