USER MANUAL -

INDEX

INDEX

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. The program also allows for probing a part and finding circle centerlines.

2.0 REQUIREMENTS

2.1 A licensed copy of Mach and a license for Mach3 Add-ons 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, XBOX360) 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.)

2.4 A probe tool.

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 DRO's and builds the code based on point selection.

3.2 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 plug-in which records all the moves.

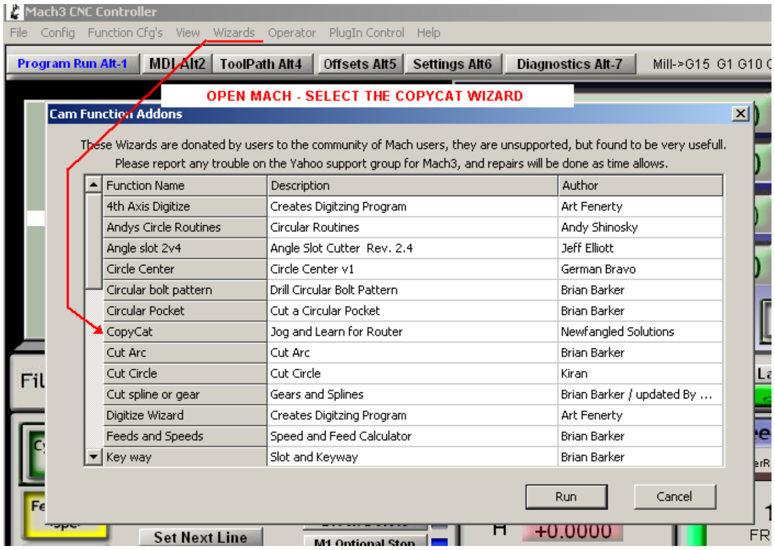
3.3 This wizard always writes to the same file, C:\Mach3\GCode\teach.tap This allows you to add to an existing file. If you want to start a new file be sure to click the 'NewTeach File' button.

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 A, D, and E.

4.0 The CopyCat Plug-in

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.

Copycat can also do probing via the Probe Screen. (see Section 6.0)

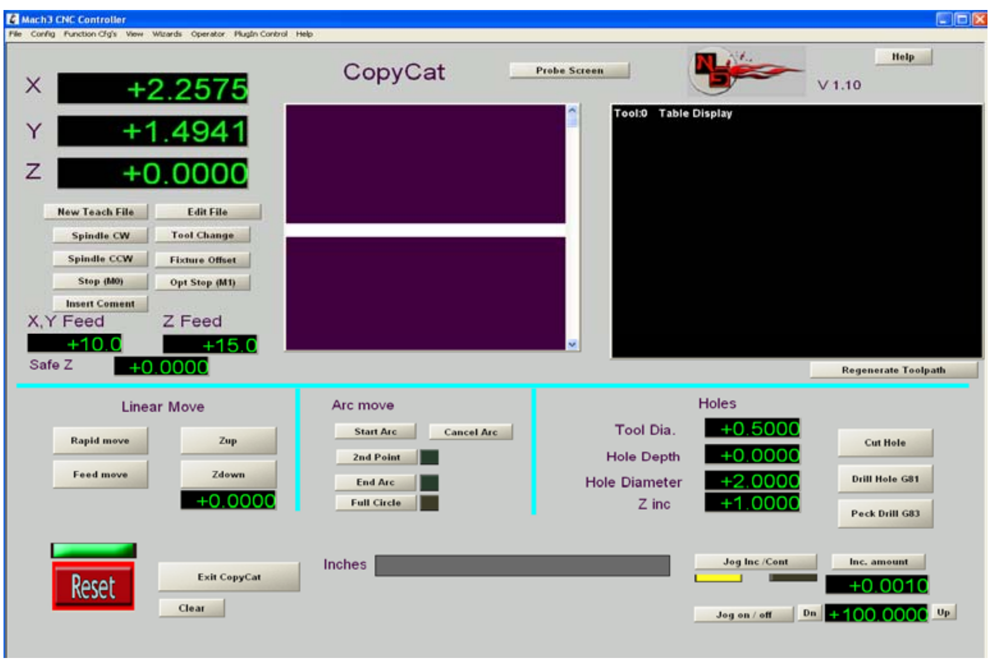


FIGURE 5.1

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.

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.

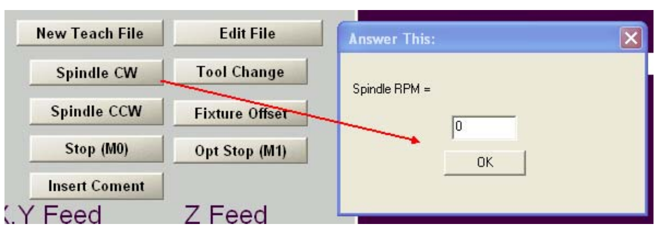


FIGURE 5.3

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.

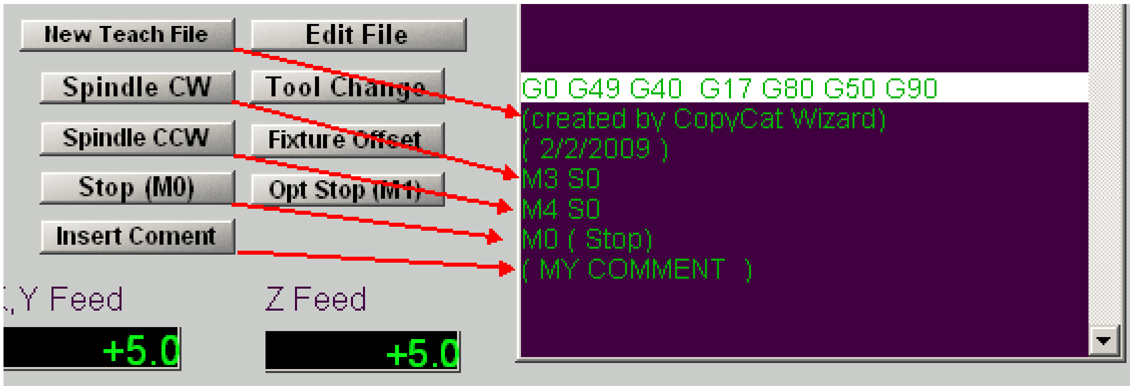


FIGURE 5.4

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 added 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 change 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.

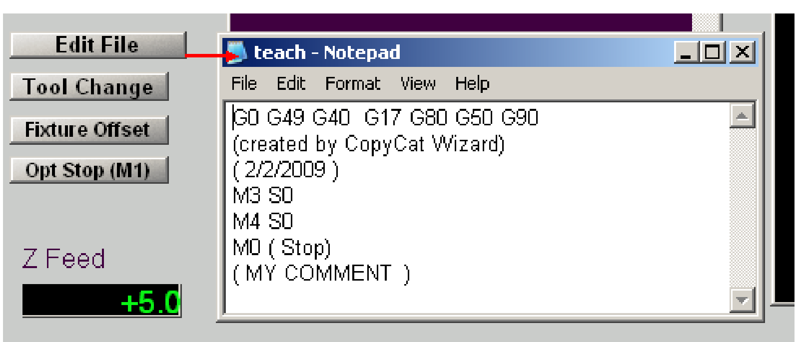


FIGURE 5.5

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.

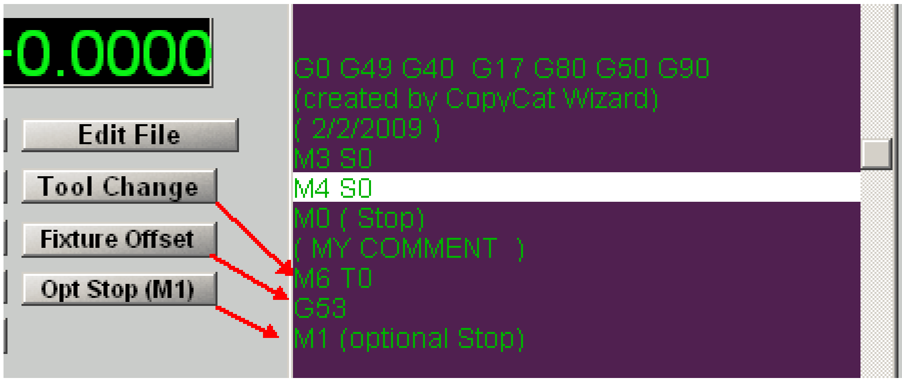


FIGURE 5.6

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 "A" for how to examples of using CopyCat.

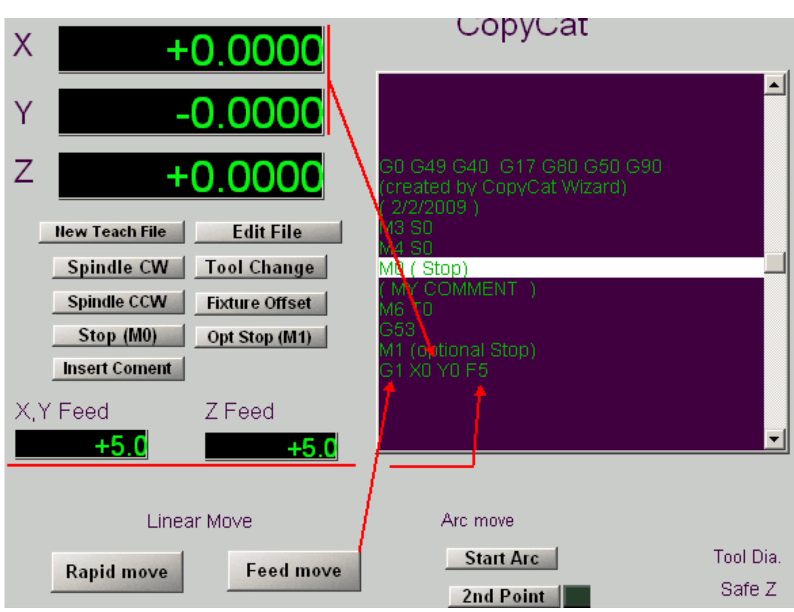


FIGURE 5.6

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.

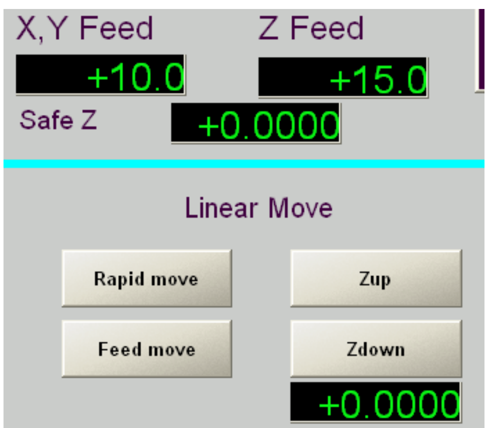
CopyCat will remember this point. You can jog anywhere and anyway until you arrive at another / next point, and then by clicking the rapid move button, another G0 will be implemented to that point.

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 Z - FEED MOVE



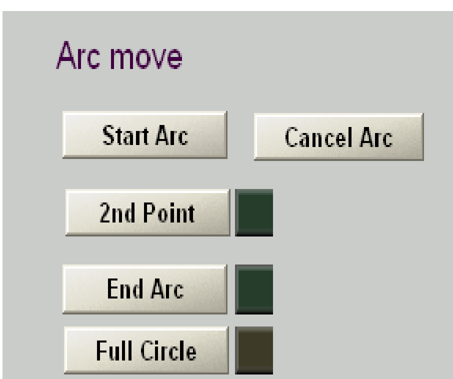
Zup – provides a rapid move (G00) to the Safe Z value in the dro and enters the code
Zdown – first movement is a rapid (G00) to 10% of the safe z value, then it provides a feed move (G01) to the z down value in the DRO at the set feed rate and enters the code.
Values for Z up or down can be positive or negative.

5.7.4 ARC 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. Note that you can cancel the arc at any time in the process.

The steps to create an Arc are as follows and shown in figure 5.7.4:

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 (LED 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 (LED 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



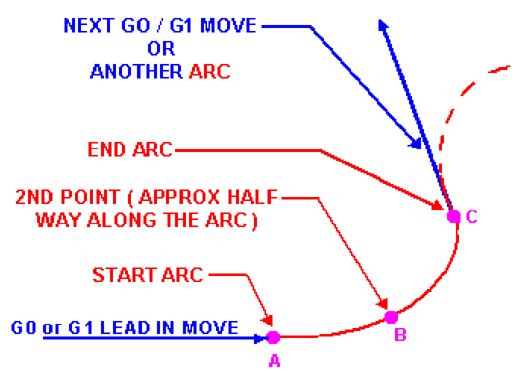


FIGURE 5.7.4

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

5.7.5 FULL CIRCLE

The steps to create a Full Circle are noted below along with an associated example shown in figure 5.7.5. Note how the moves are done and also the circle center relationship to the start and 2nd point.

1. A lead in move is made to the start point #1 on the circle unless you start it from the end of another arc or circle. The lead in should be longer than the tool diameter.
2. Now click the **Full Circle** box (LED turns yellow)
3. Click **Start Arc** [#2 & #3 point] box (LED turns green next to 2nd POINT box)
4. Move to 2nd point then click **2nd Point** box [#4] (end arc LED turns green and you are now prompted for an end point)
5. Move to end point [#5] and click the **End Arc** box.

The gcode for the circle will be generated and a comment added noting the center point coordinates.

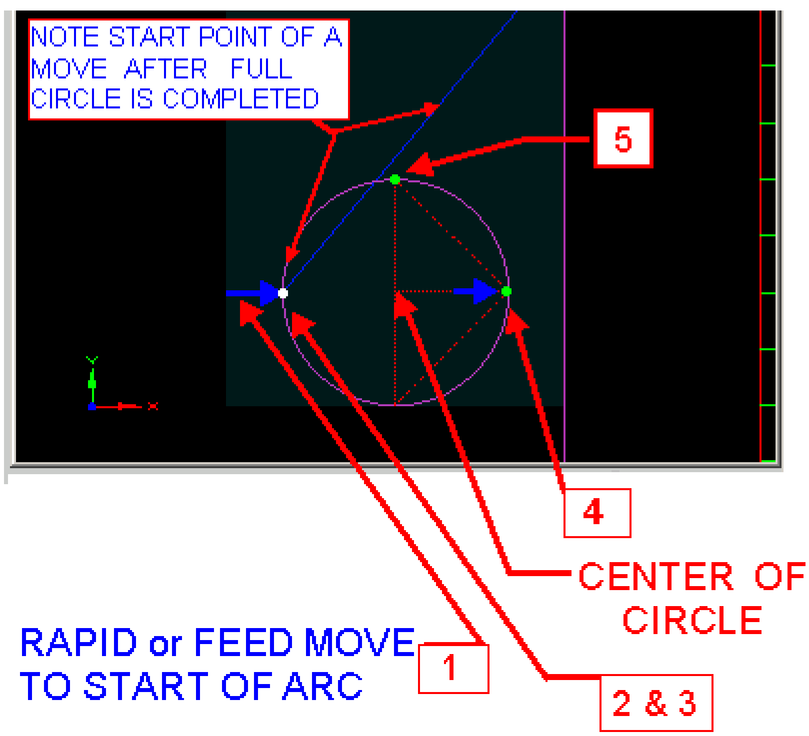
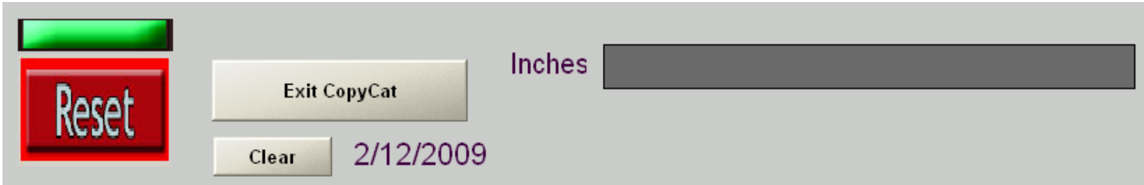


FIGURE 5.7.5

NOTE: The white and green dots, red dashed lines, arrows, etc were added for illustration purposes in figure 5.7.5.

5.8 RESET - EXIT – MDI



Reset – toggles the Estop on and off. Note that code will be written should you click one of the move boxes.
Exit CopyCat – when clicked you will exit Copycat and return to MACH3. The Teach file will be saved and generated code is output into MACH3.

Clear – will clear the MDI line

The MDI line functions the same as in MACH3.

5.9 JOGGING & MOVING TO A POINT

5.9.1 Jogging functions the same as in other Mach3 screens. Incremental or continuous is selectable and indicated by an LED. You can turn jogging on and off and a value can be input for an incremental distance. The Dn and Up keys allow for increasing or decreasing your default rapid speed from 0 to 100%.

FIGURE 5.9.1

5.9.2 In general, you move around the machine freely with any of the jog functions, then when you are at the desired point you press a movement button to define the type of move. Note that a lead in move may be required such as in doing an arc or circle.

5.9.3 HOLES

CopyCat provides a menu for cutting, drilling (G81 & G83) at a defined location based on inputs as shown in figure 5.9.3 below.

FIGURE 5.9.3

5.9.4 Here is how it works:
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 Z increments.

5.9.5 TEXT CODE & TABLE DISPLAY SCREEN

FIGURE 5.9.5

The text screen (above left fig 5.9) displays all CopyCat generated code and edits made during the process. You can use the slider to view tool paths in the Table Display on the right. The table display functions the same as other MACH3 table displays and can be manipulated by using the mouse.

6.0 PROBE SCREEN ACCESS

The probe screen can be accessed by clicking on the Probe Screen box with the left mouse key. Note also that some help files are provided when the help box is clicked.

FIGURE 6.0

7.0 PROBING – COMMON SCREENS

7.1 You can return to the main screen at any time by clicking the Main Screen box shown Figure 7.2.

7.2 Figure 7.2 , 7.2.1, 7.2.2 function the same as already explained.

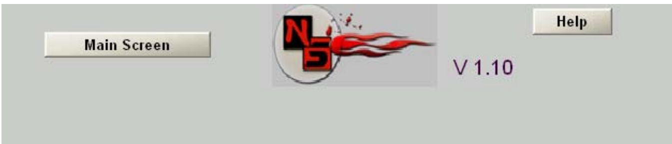
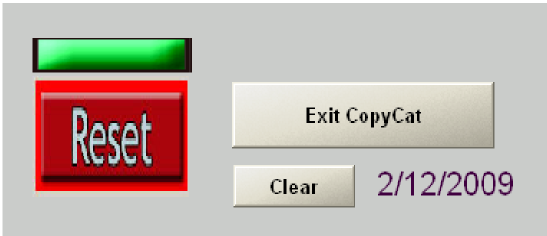


FIGURE 7.2



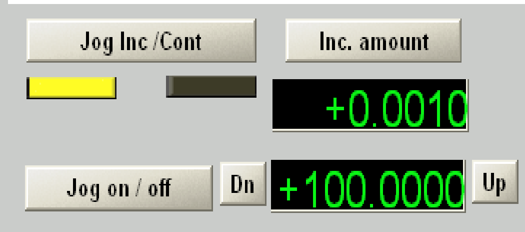


FIGURE 7.2.1 FIGURE 7.2.2

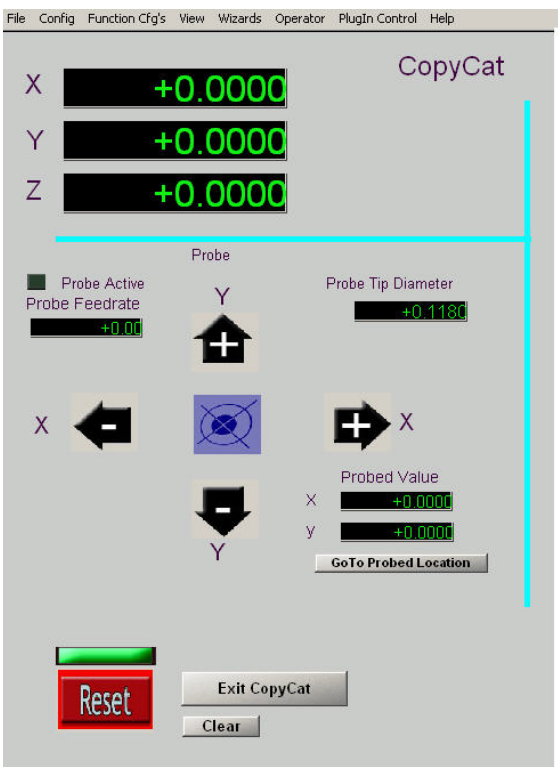
7.3 PROBING OPTIONS

The Probe Screen is shown below.

Two ways of probing are provided as shown in Figure 7.3.1 and 7.3.2 below. Figure 7.3.1 is used with a touch probe as described in Appendix "E" for finding a circle center or edges of plate.

Figure 7.3.2 should be used with video and **NOT WITH A PROBE** as described in appendix "A". The axis movement will not stop. The three point method is used for finding inside or outside circle centers using the video plug-in. (A different type of probe is required to use the three point method and is not covered in this manual.)

See also Appendix A, D, and E.



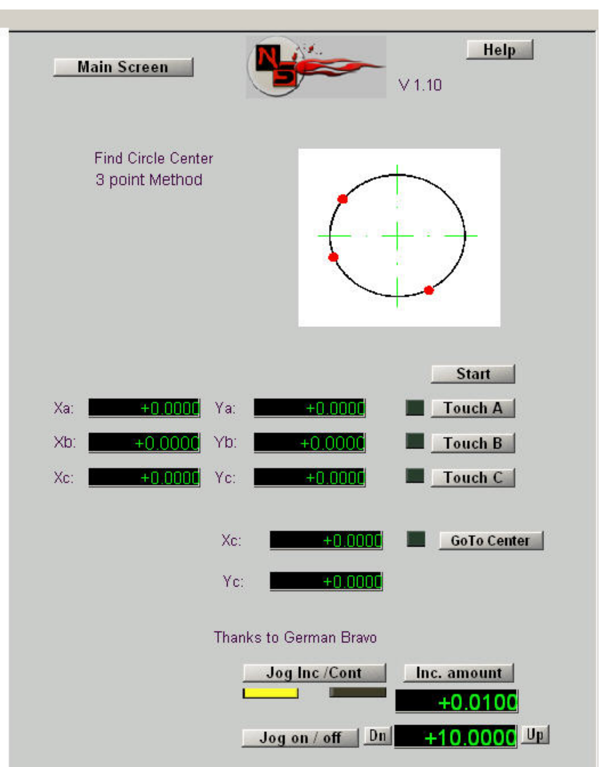


FIGURE 7.3.1 **FIGURE 7.3.2**

(PROBE) **(VIDEO)**

7.4 DRO

The DRO shown in figure 7.4 will record the hole center position and also provide a location for use in the CopyCat main screen should you want to move to or from the hole location. When you enter the Probe screen the current Copycat coordinates are displayed in the DRO. Likewise after the probing operation the DRO's now will provide the hole center location relative to your starting point should you so desire.

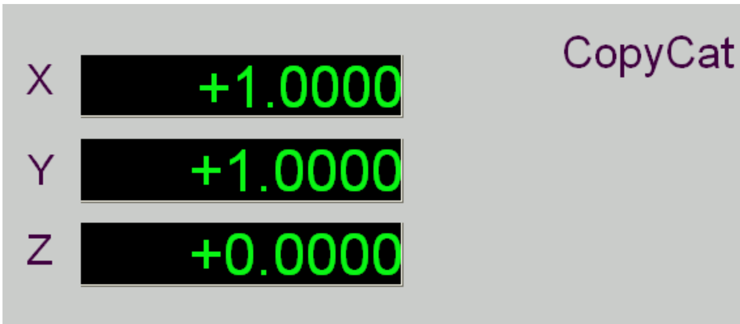


FIGURE 7.4

7.5 USING A PROBE TO FIND AN EDGE OR CIRCLE CENTER

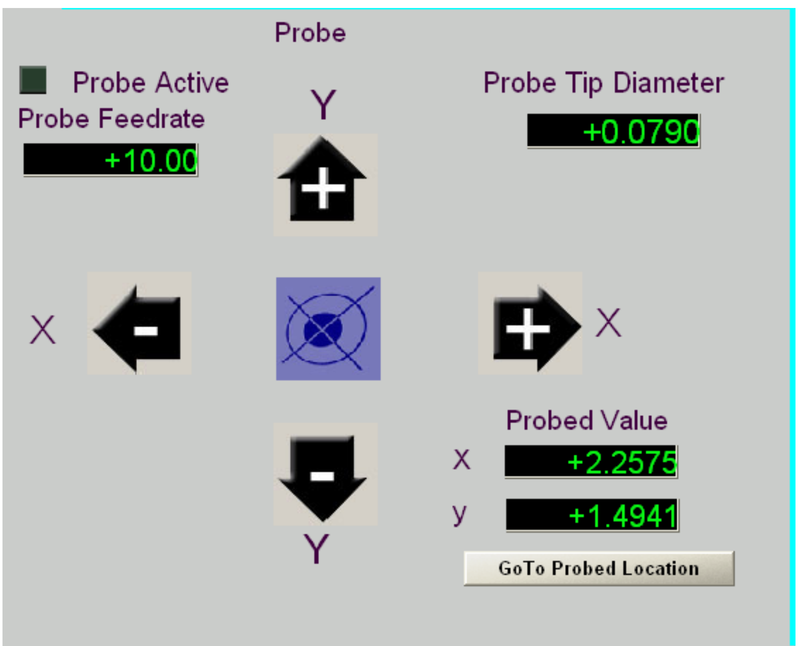


FIGURE 7.5

See Appendix "A" for how to make a probe, configure MACH3 for a probe, and find an edge or inside circle center.

7.6 USING THE THREE POINT METHOD FOR CIRCLE CENTER
(CAUTION: TO BE USED WITH VIDEO)

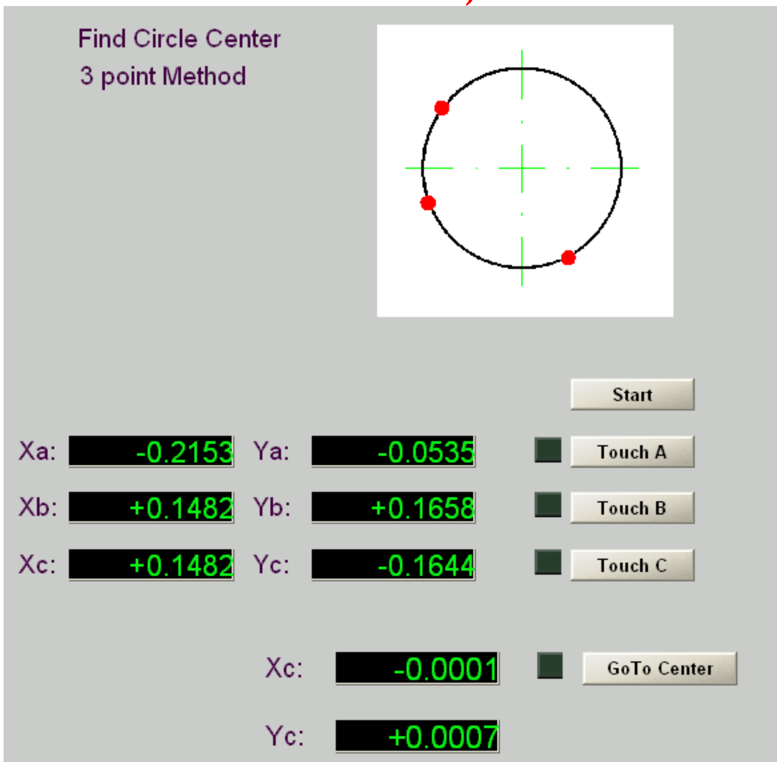


FIGURE 7.7

7.6.1 Here is how it works:
You simply move to three points on the circle (which can represent the outline of whatever) and click the Go to Center button. The center location is recorded in the DRO's and you machine will move to the center location.

7.6.2 The points should be in three different quadrants of the circle.

Start – activates the probe activity and an LED next to *Teach A* will light indicating a move to point A of your choice is required.

Teach B- led will light and move is made to point B

Teach C -led will light and move is made to point C

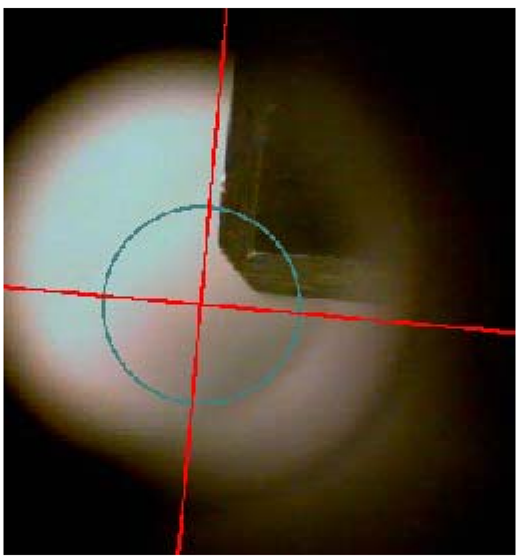
GoTo Center- The machine will move to the center of the circle based on the defined three points.

See Appendix A and D for examples on using a video camera.

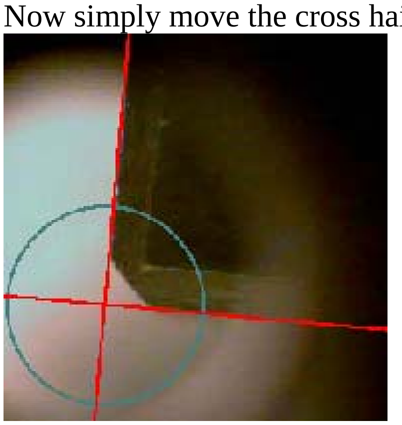
Prior to using CopyCat mount the video camera and have the probe installed. This tutorial will use both of the devices and movement of the mill axis's can be done with the keyboard keys or in this case the XBOX360 was used.

EXAMPLE #1 – DISTANCE BETWEEN SPINDLE AND VIDEO CROSSHAIRS

1. Open Mach, zero the DRO's and make sure no file is loaded.
2. Open the video window
3. Go to wizards>pick CopyCat
4. New Teach File, the DRO's should be zero
5. Open Probe Screen and use the probe to find the corner of the plate. You may want to set the Probe Feedrate per prior testing. Note that DRO's update to reflect the plates corner and the coordinates are saved when you go back to the Main Screen. Go back to the main screen.
6. Click Rapid move to document the location of the propped plate corner.
7. Adjust the quill or knee on the mill, if necessary, to bring the video camera into sharp focus on the item to be scanned. We are now going to scan the the corner of the plate. Since the plate dose not have a sharp corner, and the camera was not aligned with the X and Y axis, the slider was used to rotate the crosshairs in line with the corner as shown in figure below.

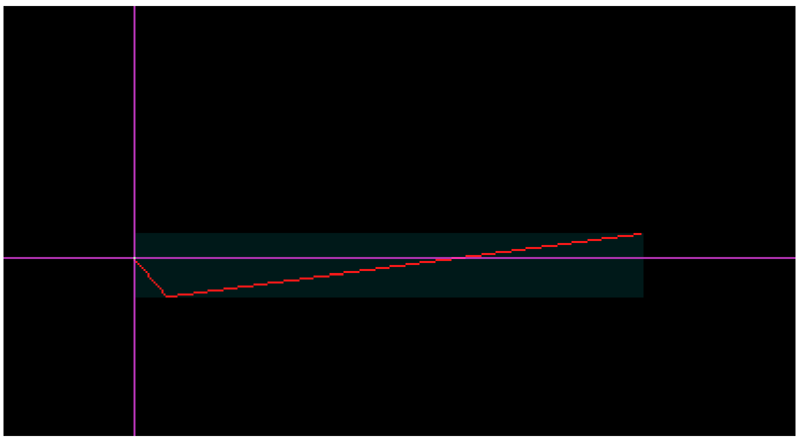


Now simply move the cross hairs to find the plate edge. As shown below.



8. Rapid move to document the video located corner of the plate.
9. Edit File and add a comment line to document the difference of probe to video center line values.
10. Save the file as is to a location of your choice. Here is the gcode thus far.

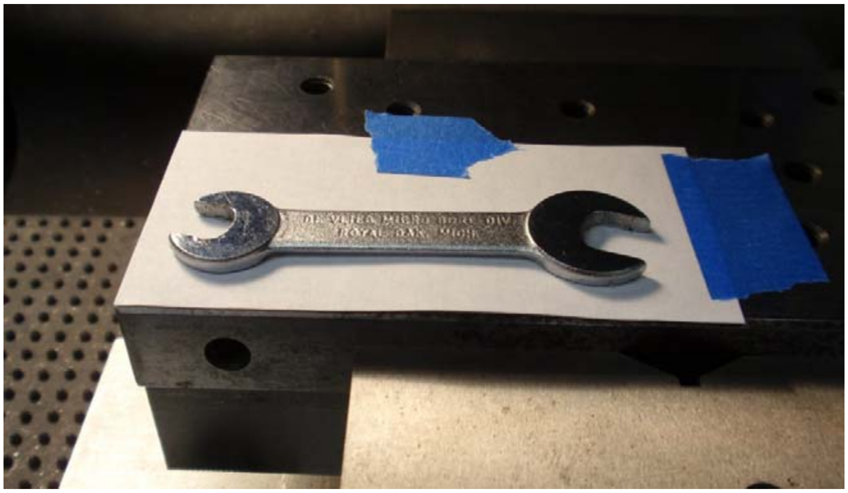
```
G0 G49 G40  G17 G80 G50 G90
(created by CopyCat Wizard)
( 3/8/2009 )
G0 X0 Y0 (RAPID MOVE TO 0,0,0)
G0 X0.2446 Y-0.2783 (RAPID MOVE TO DOCUMENT PROBED CORNER PLATE LOCATION)
G0 X3.7579 Y 0.1846 (RAPID MOVE TO DOCUMENT VIDEO LOACTION OF PLATE CORNER
LOCATION)
(    X3.5133 Y 0.4629 DIFFERENCE OF PROBE TO VIDEO CENTER LINES)
```



There are other ways to find the difference between the video and probe center line.
One such method would be to probe to find the plate corner, probe to the center of a circle, now probe the center of the circle using the video camera (3 point method).
You now have the coordinates of each probing and can calc the difference between them.

EXAMPLE #2 - USING VIDEO

Here we are going to generate the Gcode to cut the profile of a small wrench. The wrench is placed on a plate. The corner of the plate can be probed and considered 0,0,0. This is a nice quick exercise (10 minutes to generate the code and back in Mach3 ready to cut!). Remember that you must make a rapid or feed move before you can do an arc and that also you can continue from one arc to another arc.

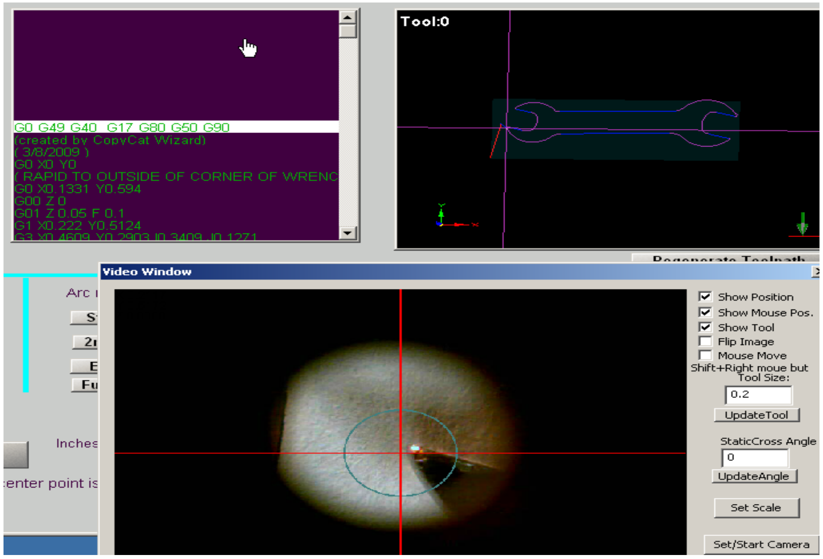


1.OPEN A NEW COPY CAT FILE
G0 G49 G40 G17 G80 G50 G90
(created by CopyCat Wizard)
(3/8/2009)
2.THE CORNER IS PROBED AND THEN SET TO X0 Y0
G0 X0 Y0
3.A RAPID AT CLEAR Z MOVE IS MADE AWAY FROM THE CORNER
G0 X0.1331 Y0.594
4. Z FEED MOVE INTO THE MATERIAL
G00 Z 0
G01 Z 0.05 F 0.1
5.FEED MOVE TO THE START OF THE FIRST ARC
G1 X0.222 Y0.5124
6.THE ARC IS STARTED
G3 X0.4609 Y0.2903 I0.3409 J0.1271
(The X center point is 0.3409 Y center point is 0.1271)
7.A NUMBER OF ARC MOVES ARE MADE UNTIL YOU GET TO THE STRAIGHT HANDLE SECTION OF THE WRENCH, THEN A FEED MOVE IS MADE TO THE END OF THE HANDLE.
G1 X2.9045 Y0.4567
Just continue arc and feed moves until you get to the end. Since you are generating code to cut the profile no additional Z moves are required until the end.

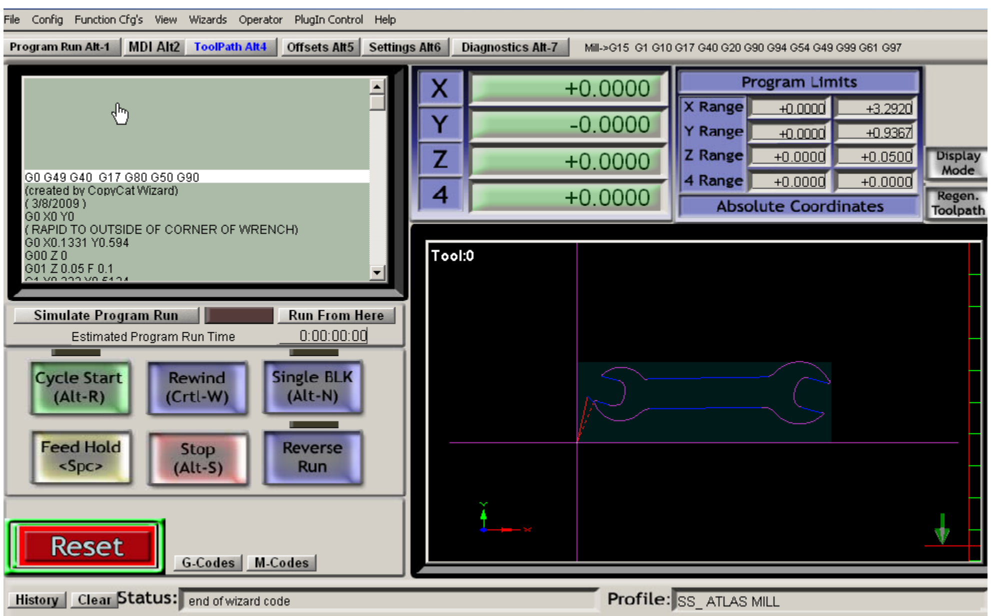
G0 Z 0
G0 X0 Y0
(10 MINUTES TO DO FROM SCAN INTO MACH READY TO CUT)
(end of wizard code)

Note that if at any time you loose track or make a bad move you can always “edit file” and do MDI or jog moves to go back to some point and then just delete code and start again from that point.

Here is a picture of completed scan in CopyCat.



When you exit CopyCat the code file is opened in MACH allowing dry run and editions if so desired. You can also save the file from MACH.



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/Millimeter 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/Millimeter 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

Summary of G-codes

Code	Description	Code	Description
G0	Rapid positioning	G54	Use fixture offset 1
G1	Linear interpolation	G55	Use fixture offset 2
G2	Clockwise circular/helical interpolation	G56	Use fixture offset 3
G3	Counterclockwise circular/Helical interpolation	G57	Use fixture offset 4
G4	Dwell	G58	Use fixture offset 5
G10	Coordinate system origin setting	G59	Use fixture offset 6 / use general fixture number
G12	Clockwise circular pocket	G61/G64	Exact stop/Constant Velocity mode
G13	Counterclockwise circular pocket	G68/G69	Rotate program coordinate system
G15/G16	Polar Coordinate moves in G0 and G1	G70/G71	Inch/Millimetre unit
G17	XY Plane select	G73	Canned cycle - peck drilling
G18	XZ plane select	G80	Cancel motion mode (including canned cycles)
G19	YZ plane select	G81	Canned cycle - drilling
G20/G21	Inch/Millimetre unit	G82	Canned cycle - drilling with dwell
G28	Return home	G83	Canned cycle - peck drilling
G28.1	Reference axes	G84	Canned cycle - right hand rigid tapping
G30	Return home	G85/G86/G88/G89	Canned cycle - boring
G31	Straight probe	G90	Absolute distance mode
G40	Cancel cutter radius compensation	G91	Incremental distance mode
G41/G42	Start cutter radius compensation left/right	G92	Offset coordinates and set parameters
G43	Apply tool length offset (plus)	G92.x	Cancel G92 etc.
G49	Cancel tool length offset	G93	Inverse time feed mode
G50	Reset all scale factors to 1.0	G94	Feed per minute mode
G51	Set axis data input scale factors	G95	Feed per rev mode
G52	Temporary coordinate system offsets	G98	Initial level return after canned cycles
G53	Move in absolute machine coordinate system	G99	R-point level return after canned cycles

M-code	Meaning
M0	Program stop
M1	Optional program stop
M2	Program end
M3/4	Rotate spindle clockwise/counterclckwise
M5	Stop spindle rotation
M6	Tool change (by two macros)
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 "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 - coordinated 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 millimeter (if millimeters 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 the-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 helix

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.

1.0 INTRODUCTION

1.0 This appendix highlights some information on a video camera and provides for modifcations external to the camera.. The modications allow for better use of the camera when selecting points on a viewed item via the video screen. Most any web cam can be used along with the video window as long as it is recognized by the the plug in program. (some cameras may not work)

This appendix does not cover the different types of cameras or provide information on optics.

2.0 THE VIDEO PLUGIN

To use the camera make sure that the Video plugin is turned on in Mach as shown in figure 2.0.
No config is currently available.

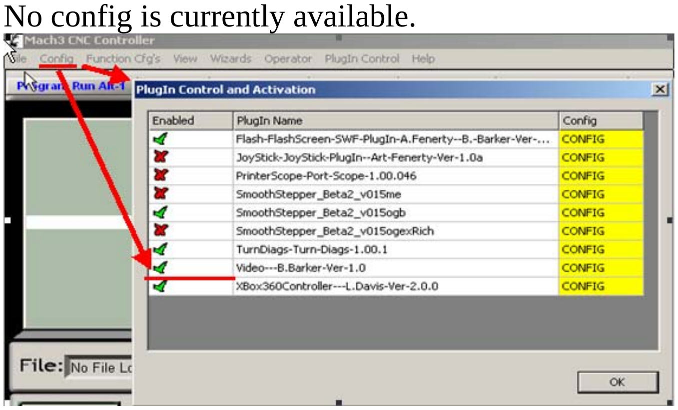


FIGURE 2.0

While in Mach and before using CopyCat go to the PlugIn Control tab and select the Video Window as shown figure 2.1.

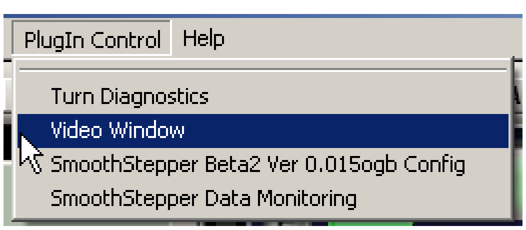


FIGURE 2.1

The video window should open as shown in figure 2.2
The slider on the bottom will allow rotation of the crosshairs.

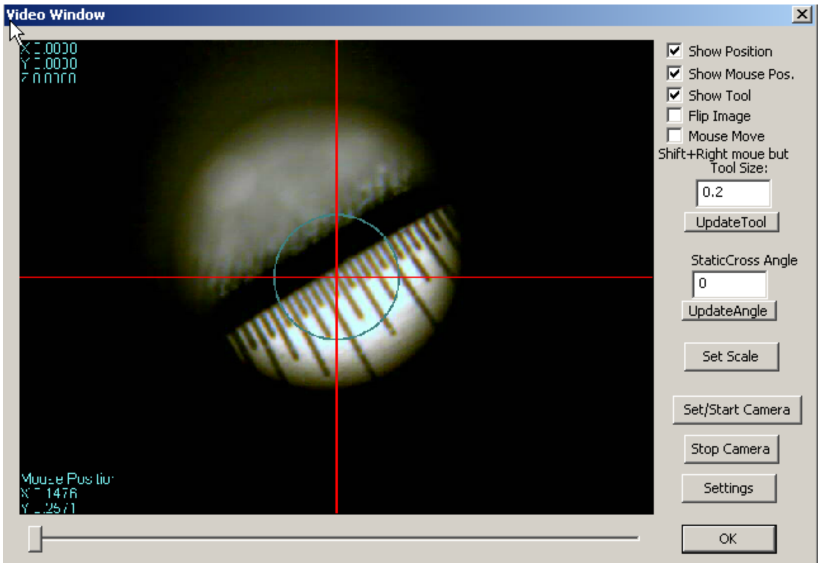


FIGURE 2.2

You can change the color of the DRO, Tool Color, and Cross Color by clicking on the Settings box as shown in figure 2.3. The only required item is the crosshair so you can align it on a point when using CopyCat. Show Position, Mouse Pos., and Toll are not required and can be turned off by removing the check mark in the upper right boxes.

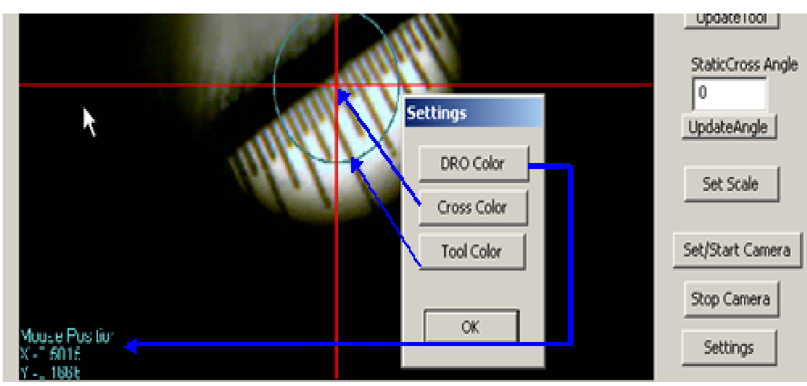


FIGURE 2.3

3.0 WEB CAM

The PC Camera shown was purchased from Radio Shack manufactured by Gigaware (25-0157) for approx $20.00. It uses the USB port.This particular camera is manually focused and has a ring which allows you to do the focusing.

Shown below in figure 3.1 is the video camera attached to the mills spindle. The plate attached to the quill along with a pipe clamp is used to hold the camera in an adjusted / fixed position. The plate shown was custom made for other uses and remains in place when milling.

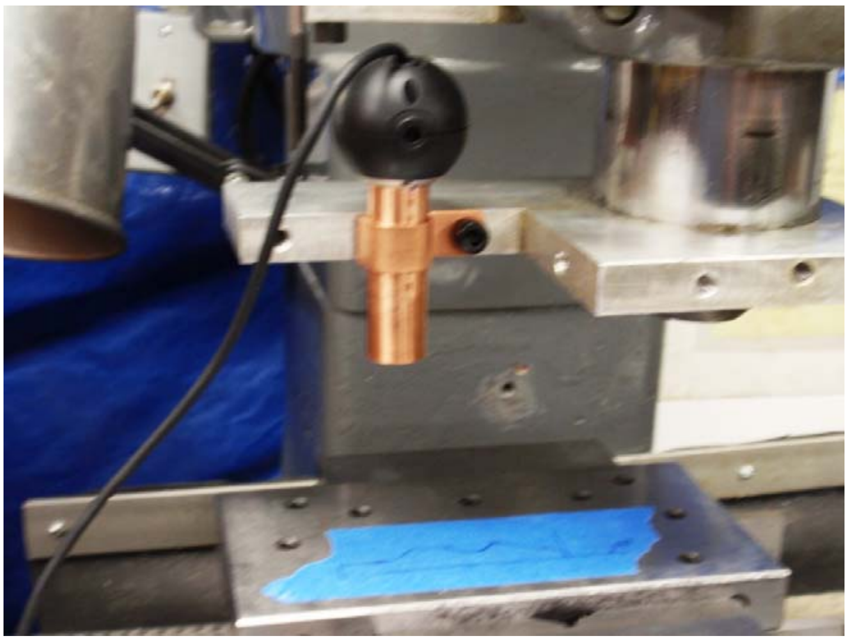


FIGURE 3.0

Figure 3.1 below shows the resulting working distance from a work surface . Approx 0.150” adjustment in depth of field is available via up or down movement of the quill. The “system” composed of the camera and magnifying lens will determine the working distance, tube length, and lens to camera focusing ring to some magnification desired. Adjustment of any part of the system will change others so there are trade offs. The trade offs are viewable area (less magnification- more viewing), magnification (accuracy of crosshairs on a location), image sharpness (too much magnification and you will pixilate in the video window) , working distance (want to be able to view up and down using Quill movement). So it will become a users preference .
Desirable is a maginification of 5 to 10X, a reasonable working distance, and sharpness of the image when viewed in the Video Screen are preferred.
You can also just use the camera as is if it suites your needs.

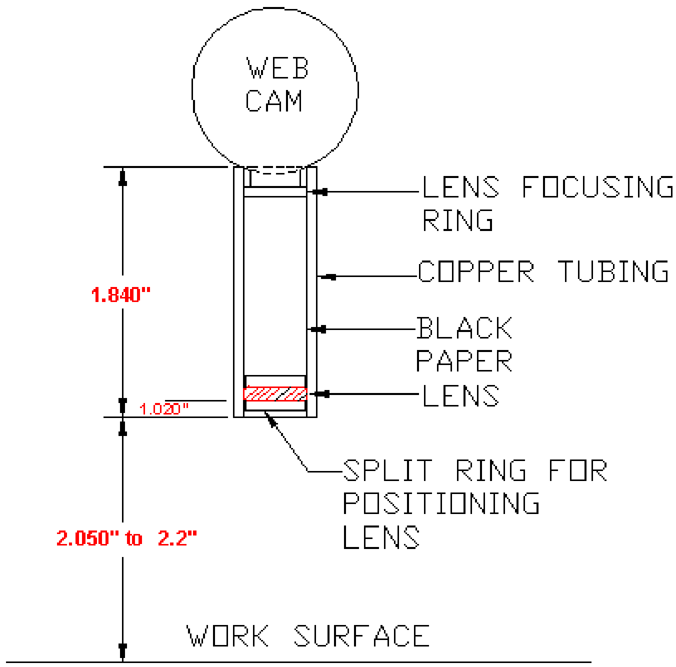


FIGURE 3.1

Here is how to find some basic dimensions. Hold the lens in your hand and raise it along a scale while viewing something (a small divided scale) make note on the scale where the viewing goes to max magnification and is not distorted. This is the max working distance. Now hold the lens over the scale at the working distance and bring the camera to above say 3" away from the lens. Adjust the cameras focusing ring for a sharp video picture (The camera must connected to the computer and viewing the video screen in MACH). This will set the tube length. Make a tube (in this case it was just a piece of copper tubing) and allow approx an additional inch. Make two rings to hold the lens (provide a slit in the rings so they can compress some and slide inside the tube and hold the lens in place. Now you can refine the positioning of the magnifying lens and camera lens. Take dimensions of where the lens is relative to the camera lens ring and make a neat assembly. You will need to play a little and remember the tradeoff's talked about earlier.

You will want to minimize reflections by placing a piece of flat finish black paper inside the tube. You can also sharpen the edge of the viewing area and other modifications if so desired. KISS sometimes is good!

Here is a picture of the using the camera as is. Make note of the small scale divisions in figure 3.2 as each are 0.010" wide.

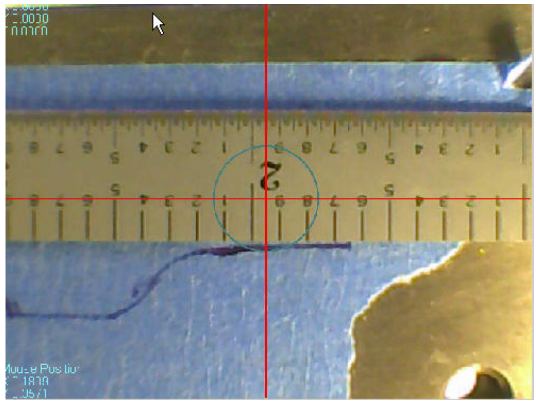


FIGURE 3.2

Below in figure 3.3 and 3.4 is a before and after comparison of the same camera at the same distance above a scale. Magnification is 5x. Now to confuse the world with a "new relative definition" of magnification. It is simply MEASURED IMAGE SIZE divided by HOW BIG THE VIDEO IMAGE is. So if a rule was placed on the video screen you have a physical size of 0.5" on the rule to a 0.1" size length on the screen, thus .5/.1= 5x

Considering other parameters desired, the combination below allows for repeatable accuracy of 1 or 2 thousands of an inch when using CopyCat. The camera assembly is quite compact.

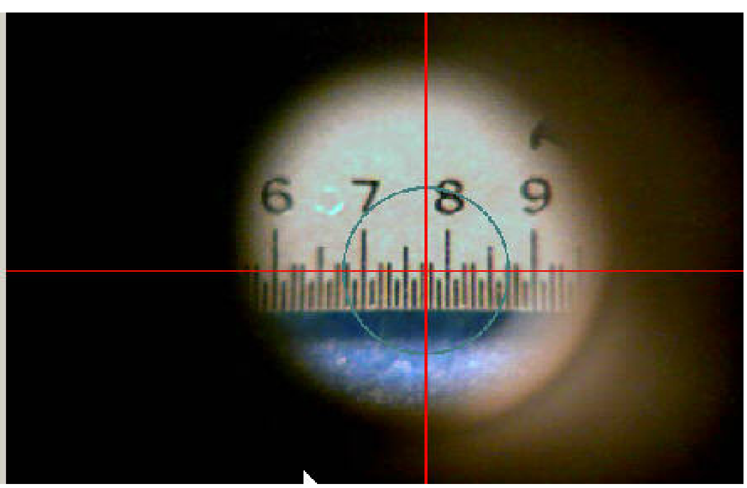


FIGURE 3.3
(NO MAGNIFYING LENS)

FIGURE 3.4
(MAGNIFYING LENS)

1.0 INTRODUCTION

1.1 One of CopyCat's features is probing. Probing was discussed in section 6 & 7 of the CopyCat Manual. You can use a touch probe to find an edge in either of the four directions or a circle center. You could then return to the main screen and use that location for one of the movement commands.

1.2 The "Using Mach3 Mill Manual" section 4.11 (Digitize Probe) & 10.7.12 (G31 – Straight Probe) is suggested reading.

1.3 This appendix provides information on how to make a probe, configure Mach, and actually use the probe in context of CopyCat.

2.0 PROBE

2.1 PROBE BASICS

Commercial probes are available or can be home made. A probe can be as simple as a wooden dowel with a protruding nail in the bottom. The probe must be isolated / insulated from the machine chuck else the chuck would act as a ground. Figure 2.1 shows a probe which is short, very accurate and provides for a removable / replaceable tip. The picture is self explanatory and the only comment is that the tip not make contact with the body of the probe.

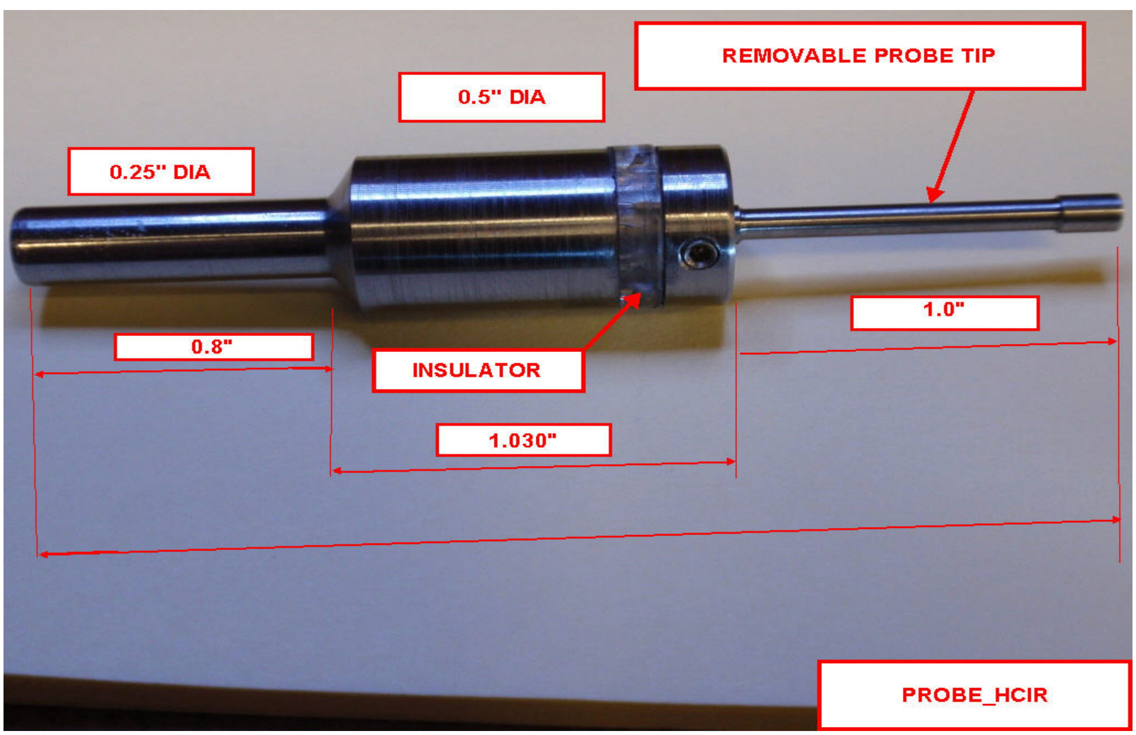


FIGURE 2.1

2.2 HOW THE PROBE WORKS

When the probe makes contact with a metallic surface it triggers an input to MACH. That input is used by Mach to control axis motion in some manner.

The computer parallel port has 25 pins. Pins 10,11,12,13 and 15 are used as inputs to Mach. In the computer, whatever pin is allocated to your probe is normally kept high (5v approx) by the internal resistors in the computer so the computer thinks the pin is on until the pin is grounded. So if one of these pins were to touch ground, in this case the machine frame, the state of that pin would be change from high to low. Thus it becomes “active low”. The input to the computer is designed so that in-going signals pull down the pin to zero.[Mote: Should you be using an external break out board consult the manufactures instructions.]

You can check the state of your pin using Mach3 diagnostics.

1-With a wire attached to the pin, open Mach3 Mill, go to the “Diagnostics ALT 7” screen and touch / ground the wire to the machine. You will see the Digitize LED turn green. The LED will go out upon removal of the wire from ground.

The diagnostic lights only show when the switch is "active". You can change when the switch is active (+5v or a 0v) in Ports & Pins.

2-The pin state can also be checked using the SmoothStepper plugin. Click the PlugIn Control tab in MACH and select “SmoothStepper Data Monitoring”. When the wire which is attached to the pin is grounded you will see the check mark for that particular bin become unchecked.

2.3 PROBE MOUNTING

The probe in Figure 1.1 was made to be used in the collet or chuck of a cnc mill. Since the tip is insulated from the body the switching wire is attached to the probe tip. When the tip of the probe touches a metallic surface Mach will stop any axis movement. Figure 2.3 shows the probe along with the wire in the mills chuck. An electrical plug or jack attached to the your case can allow for removal of the wire when not in use.

FIGURE 2.3

2.4 PROBE WIRING

You will need to configure Mach and physically provide a connection for the probe from the input pin. Figure 2.4 shows the probe wire directly attached to a pin on one of the SmoothStepper board ports. Pin #15 / Port 2 is used for example.

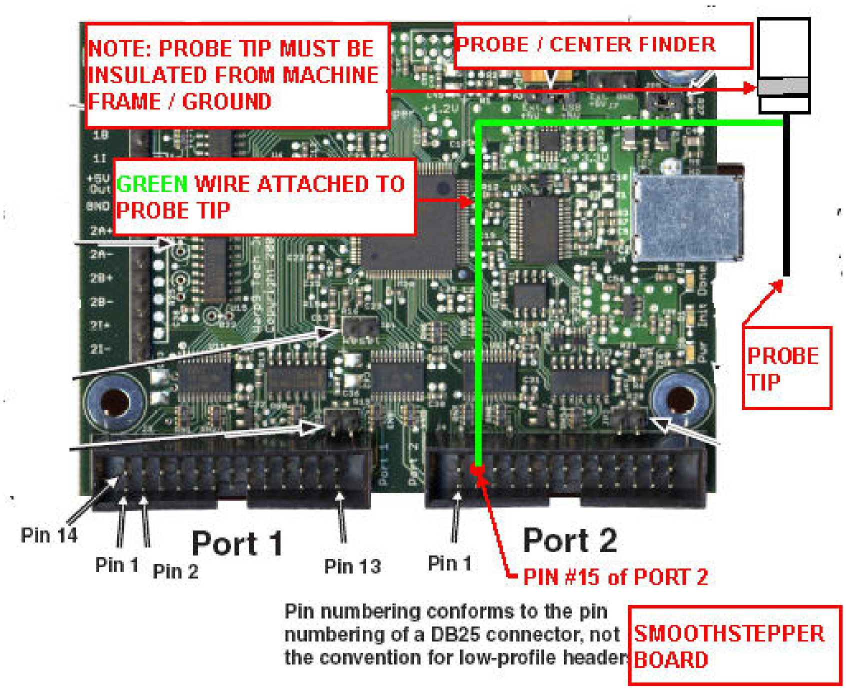
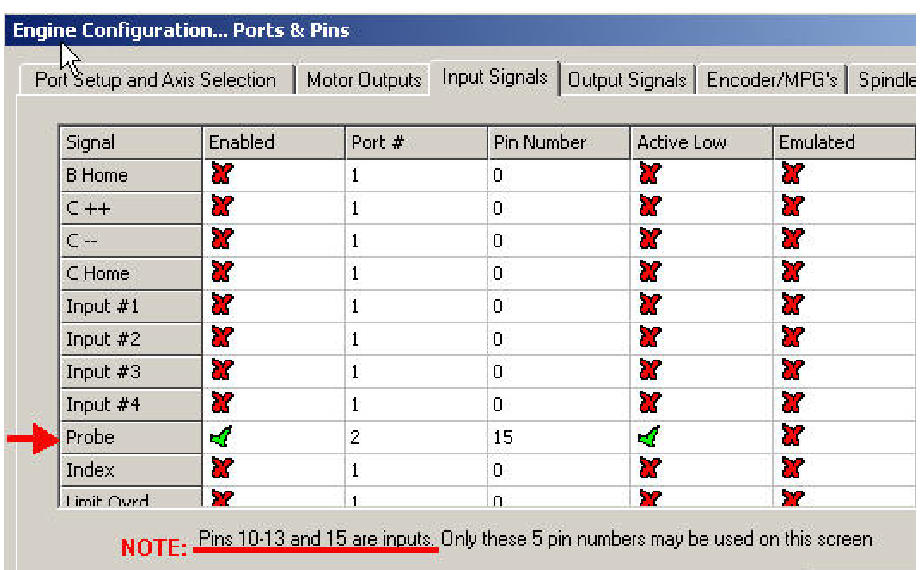


FIGURE 2.4

2.5 MACH CONFIGURATION

Figure 2.5 shows how to configure the probe in Mach3 Mill.



Signal	Enabled	Port #	Pin Number	Active Low	Emulated
B Home	✗	1	0	✗	✗
C ++	✗	1	0	✗	✗
C --	✗	1	0	✗	✗
C Home	✗	1	0	✗	✗
Input #1	✗	1	0	✗	✗
Input #2	✗	1	0	✗	✗
Input #3	✗	1	0	✗	✗
Input #4	✗	1	0	✗	✗
Probe	✓	2	15	✓	✗
Index	✗	1	0	✗	✗
Limit Ovrd	✗	1	0	✗	✗

FIGURE 2.5

Figure 2.6 shows additional configuration MACH>PLUGIN CONTROL> SMOOTHSTEPPER……….which may be required if you are using the SmoothStepper. I found this value along with a probe feedrate of 5.0 or 10 to be accurate and avoid probe tip pressure upon contact.

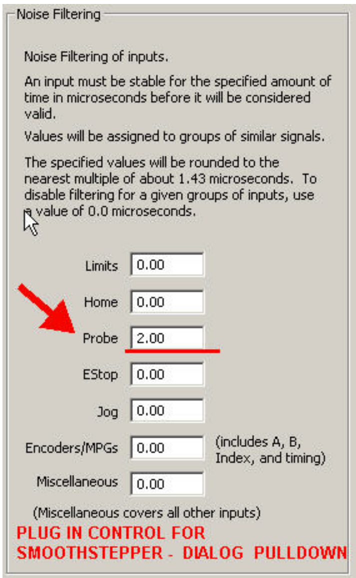


FIGURE 2.6

3.0 FINDING A CIRCLE CENTER

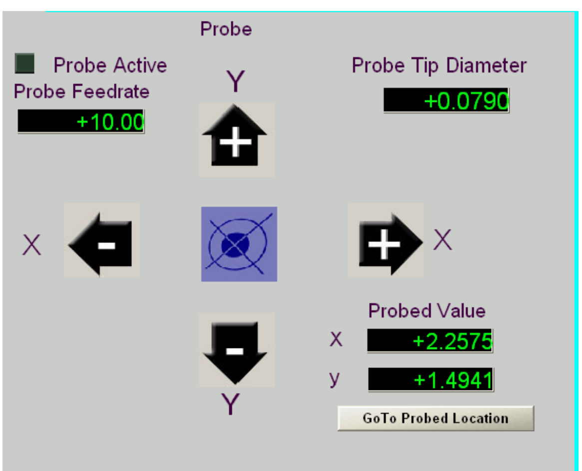


FIGURE 3.1

CAUTION: Before using the probe always check that the probe is functional. This is easily done by touching the probe tip via a wire to ground. The LED next to Probe Active in figure 3.1 will light / turn green. Also it is *highly recommended* that as long as the probe is below a clear surface the input wire never be removed from the probe tip. Damage to yourself or the probe may occur should you not heed this caution.

3.1 Here is how the probing routine works:
When the [x bullseye] is clicked the Y axis will move Y+ until the probe touches metal, the movement will reverse, touch the other Y surface, return to the center of the Y travel and then proceed in the +X until it touches, reverse and proceed to the other X surface, and finally moves to the center of the probed hole / circle.

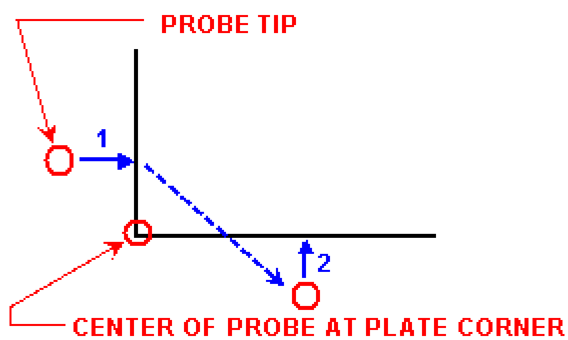
Before probing, set the Probe Feedrate and also input your Probe Tip Diameter. You have the option to clear the DRO's if you wish. It is suggested that you make a few trial runs to find a feedrate which yields the best accuracy. Note that upon probe contact the axis must stop, and frankly nothing stops instantly. Thus a slower feedrate is in order.

The DRO's keep track of all the movements. The final value in the DRO's are kept, and should you go back to the main screen, those values can be used for one of the movement commands. Note that the value displayed in the DRO's are the X & Y offsets of probe centerline relative to the circle center before probing the circle.

The X & Y arrow buttons are used for edge finding and not to find the center of a circle.

4.0 FINDING A CORNER / EDGE

Figure 4.0 shows a probe in a starting position. The specific steps to find the edge are noted below.



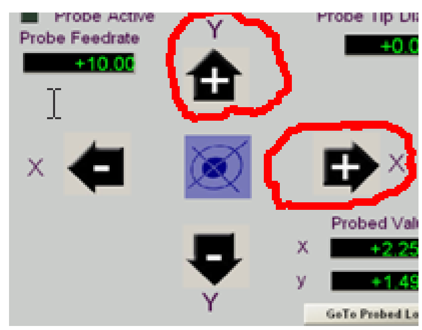


FIGURE 4.0

1. Lower the probe tip such that it can make contact with the side of the plate when moved. Click the +X arrow, the table moves until the probe touches the plate. The Probe active LED is flashing, raise the probe to a clear the top of the plate. Jog / move the axis to the bottom of the plate.
2. Lower the probe again such that it can make contact with the bottom of the plate. Click the +Y arrow, the plate moves until the probe touches the plate.
3. RAISE THE PROBE (the probe active light should not be flashing) to clear the top of the plate. Click the " GoTo Probed Location" box and the tables will move such that the center of the probe will be at the corner of the plate.

The DRO's keep track of all the movements. The final value in the DRO's are kept, and should you go back to the main screen, those values can be used for one of the movement commands. You also have the option to change the DRO's to any value you wish. ie;You start at a home position and find the corner of your plate and now call that 0,0,0. Note that the value displayed in the DRO's are the X & Y offsets of probe centerline relative to the corner of the plate before probing to find the corner.

4.1 You can also probe along the edge of the plate to check alignment of the plate to table travel. You simple probe one side point, raise the probe up and away from the edge, travel some distance and re-probe another edge location. Any difference from the first probed reading and the second reading will be the difference in plate edge location verses table movement.

NOTE: You can make moves "parallel" to the plates edge, with the probe still touching the plate, using the arrow keys. The DRO will not change and should the plates edge not run true to table movement you can put undue pressure on or damage your probe.

4.2 You can probe the inside or outside corners of fixture. Just adjust the probe movements accordingly.

5.0 FINDING CENTER OF A SQUARE / RECTANGLE

The specific steps to find the mid-point between opposite sides are noted below.

Probing the X axis (the Y values remain at zero)

1. Zero (input 0.0000) the DRO's and Probed Values
2. Lower the probe tip such that it can make contact with the side of the plate when moved. Click the appropriate arrow, the table moves until the probe touches the plate. The Probe Active LED is flashing, *raise the probe* to a clear the top of the plate.
3. Click [GoTo Probed Location]
4. Zero both the DRO's and Probed Values
5. Jog / move the probe to the opposite side, lower the probe so it can make contact with the plate, and probe to the plate. *Raise the probe.*
6. Click [GoTo Probed Location]
7. Input ½ the value of the DRO into the Probed Value. The input value should be of opposite sign of current DRO value (if + then make the value -).
8. Zero the DRO for the axis.
9. Click [GoTo Probed Location]

The tables will move such that the probe is midway between the opposite sides.

If you repeat the above steps for the Y axis your will be in the center of the square or rectangle.