

# **THE MICROMILL DSLS 3000**

(Digital Sync Lock Servo)  
Desktop CNC Machining System

## **USER'S MANUAL**

Preliminary Release

MicroProto Systems

2006

# Table of Contents

Introduction .....	3
Machine Setup .....	3
Installing the Axis Motors.....	3
Setting the Coupling Gap.....	4
Assembling the Z-axis Column .....	7
Mach 3 Program Introduction .....	9
Program Installation.....	9
Program Calibration.....	9
Main Screen View.....	10
Main Program Control.....	11
G-Code Program Display.....	12
Example Program Setup and Execution.....	18
Example Program # 1.....	19
Example Program # 2.....	22
Example Program # 3.....	23
Example Program # 4.....	23
G- Codes .....	27
M-Codes .....	33
Axis Configuration .....	34
Rotary Table Mounting .....	35
Troubleshooting Problems .....	36
Machine Maintenance .....	39
Backlash Measurement .....	40
Adjusting the Lead Screw Nuts.....	41
Adjusting the Brass Gibs .....	42
Adjusting the Headstock .....	44
Importing BobCad Files .....	45
Digitizing Probe Operation .....	46
Leadscrew Compensation .....	50
Speeds and Feeds for Drilling .....	51
Speeds and Feeds for Milling .....	52
Calculating Milling Speeds and Feeds .....	53
Spindle Speeds .....	54
Wiring Configurations for Driver .....	55

## **Introduction:**

Congratulations on your purchase of the MicroMill DSLS 3000 Desktop CNC machining system implementing advanced Digital Sync Lock Servo (DSL S) control technology. We feel you have obtained the best value in the market today with a machine that will provide you with years of productivity and enjoyment. The MicroMill is a product of many years of Research and Development by experienced Engineers and Machinists. The MicroMill is being employed by Users all over the World and has found its way into many, many applications ranging from jewelry design to complex aerospace components manufacturing.

## **Machine Setup:**

Setting up your new MicroMill system requires only a short time (usually 30 minutes to 1 hour) to get the machine up and running. The MicroMill system is shipped mostly pre-assembled except for a few machine components. The Z-axis column, the drive axis motors and spindle motor are all that must be assembled by the user. The following tools shown in figure 2 are all that is necessary for initially setting up the machine.

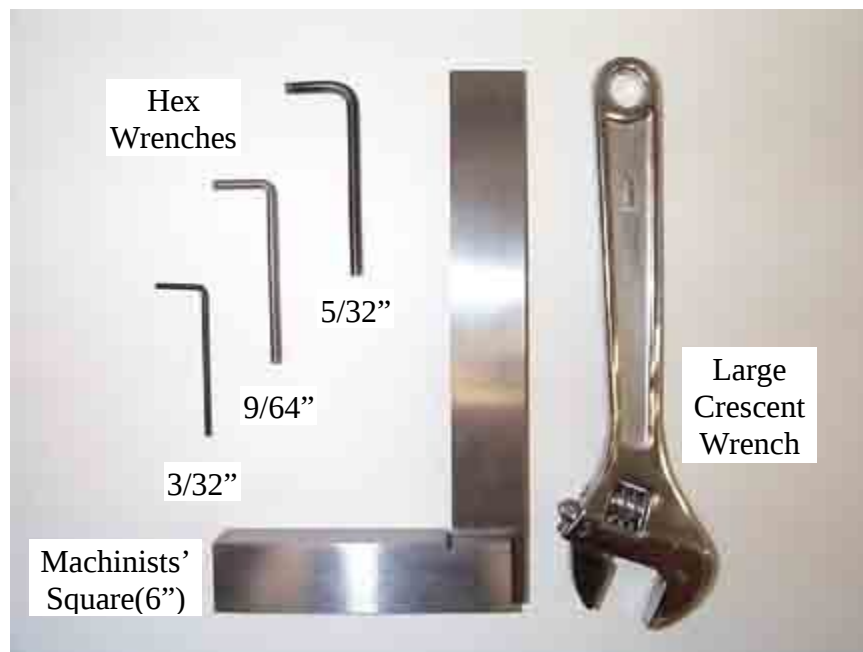


Fig. 2 - Machine setup tools.

### **Installing the Axis Motors:**

The axis stepper motors are coupled to each Lead Screw via 4 - 1/8" diameter nylon tubes. This coupling system was chosen for the following reasons:

1. To provide isolation of the motor bearings with any forces reflected from lead screw loads.
2. To provide a shock absorption coupling for smooth acceleration and deceleration of the axis.

When inserting the nylon tubes into the Lead Screw Collar it is essential that ALL 4 of them slide in freely without having to force or jam any of them in. This is tedious sometimes to get

them aligned properly, but with patience they will slide right together with a secure coupling. At this point it is important to set the gap properly between the Lead Screw Collar and Motor Coupler for the best system performance. If the gap is too large the coupling will not be rigid enough which will result in lower torque transfer and more 'play' in the system. If the gap is too small the coupling will be too rigid and not provide the necessary shock absorption for smooth acceleration and deceleration. By experimentation the optimum gap is found to be between 1/32" and 1/16". This is shown in figure 3a below.



Fig. 3a - Setting the coupling Gap.

Once the coupling gap is set use the Coupling Tube Clamp Ring to lock the Coupling Tube in place. Position it all the way to the front of the Coupling Tube and tighten down the Set Screw as shown in figure 3b below.



Fig. 3b - Locking the Coupling Tube in place with the Clamp Ring.  
Connecting the Step Motors to the Driver Box:

The X, Y, Z, and A axis step motor connection ports are labeled on the back of the driver box as shown in figure 3c. The step motor cable Din 6 connectors are inserted into their respective port with the metal tab facing upward. Once the step motors are connected to their proper ports the encoder cable connectors must be connected to their respective encoders. The step motor connected to the X-axis port must have the X encoder cable connected to its encoder. Likewise, for the Y, Z and A axes. The motors will NOT work if the wrong encoder cables are connected to them.



Fig. 3c – Driver box back panel showing axis ports.

Connecting the Relay Control Module:

The Spindle and Auxiliary Relay Control Module is connected to the I/O input port of the driver box and to a standard 120Vac outlet. The left 2 sockets are turned on with a standard M3 command and off with M5. The right 2 sockets are turned on with M8 and off with M9.



Fig. 3D – The Spindle and Auxiliary Relay Control Module.

Assembling the Z-axis Column:

The Z-axis column must next be aligned square with the table. In most cases this is a simple matter of using a standard machinist square set on the table and against the Headstock assembly as shown in figure 4 below. Rotate the column until the gap (indicated with arrows) between the Headstock and Square is reduced to zero. Use a large crescent wrench to securely tighten the column in place. It should be noted that the column contains a steel insert spacer that prevents the tubing from collapsing - so it can be tightened down with a fair amount of force. This simple process will square the column with the table for all but the most demanding cases.



Fig. 4 - Quick squaring of Spindle to Table.

If a higher degree of 'squareness' is required a **Dial Indicator** must be used with the following procedure.

1. Square the column with your machinist square as close as possible.
2. Insert Dial Indicator in the spindle and orient it at approximately 45 degrees as shown in figure 5 below.
3. Move it down and touch the table (with a small needle deflection) by slow jogging or manually with the use of the motor knob.
4. Move the indicator to the Left Side of the table and zero out the indicator.
5. Sweep the indicator to the Right Side and note the indicator deflection.
6. Carefully adjust the column until the indicator needle deflects to  $\frac{1}{2}$  of the previous reading.
7. Sweep the indicator back to the Left Side and note the indicator deflection.
8. Adjust the column again until the reading is  $\frac{1}{2}$  of what it was.
9. Repeat process 5 to 8 until your iterations show no dial indicator movement.
10. Carefully tighten down the column making sure it stays in place.



Fig. 5 - Precise Squaring of the Column using a Dial Indicator

This completes the initial mechanical setup of the MicroMill System. You are now ready to proceed onward with installing the software, jogging the axes around and then running the sample programs.



# MICROPROTO MACH 3 CONTROL INTERFACE

## Operation Manual

Kurt E. Daley, Ph.D.  
2006

### Introduction:

The MicroProto Mach 3 program is a custom designed interface that executes universal G-Code programs and a lot more. The interface was designed with many full size CNC machining center features in mind. It supports the complete set of G and M codes that are used in the world today. Hence, the user that masters all its capabilities should be well prepared to run similar equipment in a modern day CNC machine shop. The Mach 3 program will import standard post-processed ASCII G-Code files from any CAD/CAM system.

### Program Installation:

Download the Mach 3 program at [www.artofcnc.ca](http://www.artofcnc.ca) and install it. From the MPS2003 disk copy the files **Mach3mill.xml**, **Mach1lic.dat** and **MPS1024E.set** into the main Mach 3 folder. When prompted replace the existing **Mach3mill.xml**. From the MPS2003 disk copy the file **BackgroundToolpath3.jpg** into the Mach 3 subdirectory **Mach3\Bitmaps\Millbitmaps**. Upon starting the Mach 3 program this will now configure everything for the MicroMill DSLS 3000 system as shown in figure 5b below.

Execute the Mach 3 program by double clicking the Mach3mill icon on your desktop. Locate the tab buttons at the top of the screen view and click on the **Diagnostics Alt7** tab and then click on the **Toolpath Alt4** tab. The custom machine control screen should now be visible and you are ready to load and execute programs.

# Main Control Screen

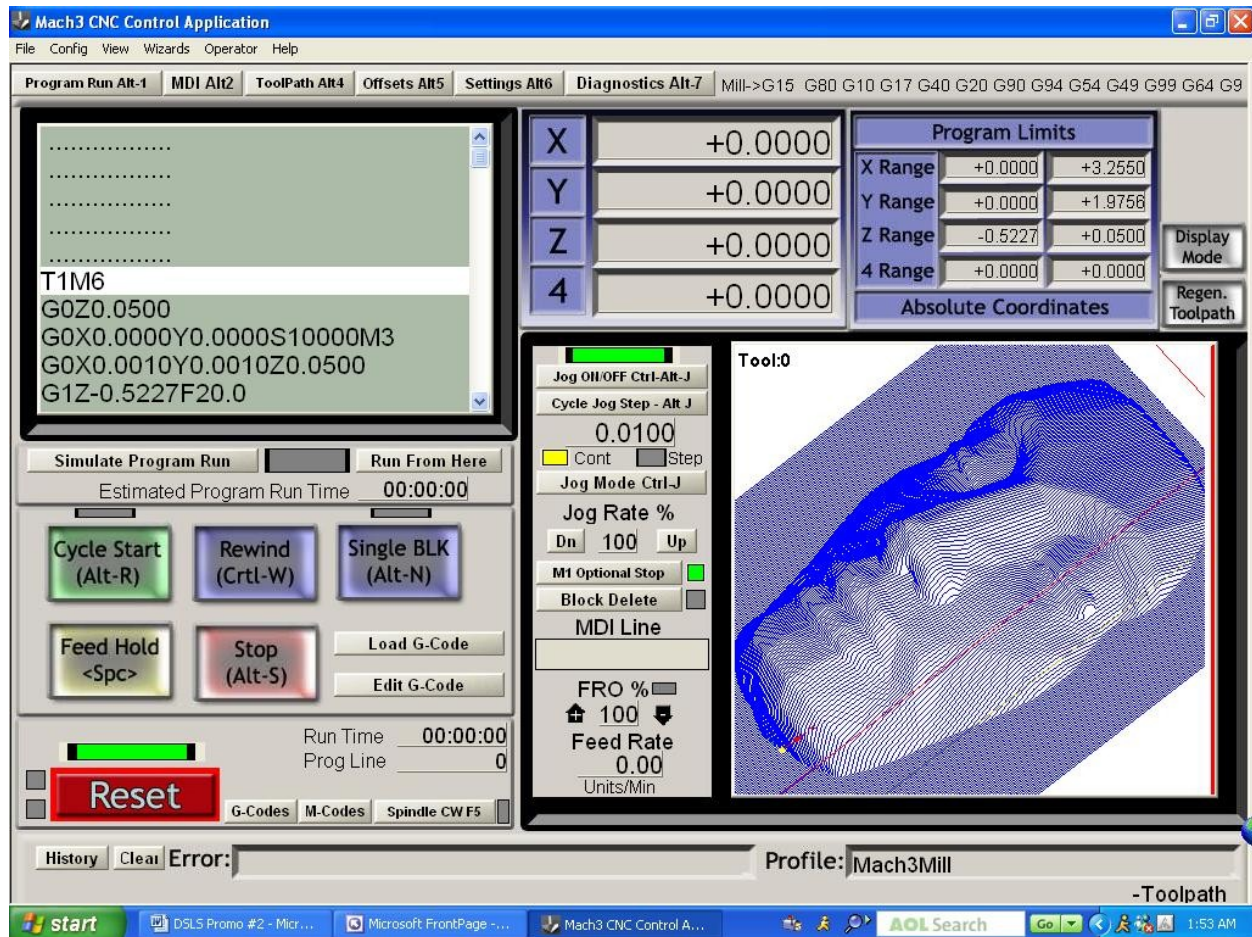
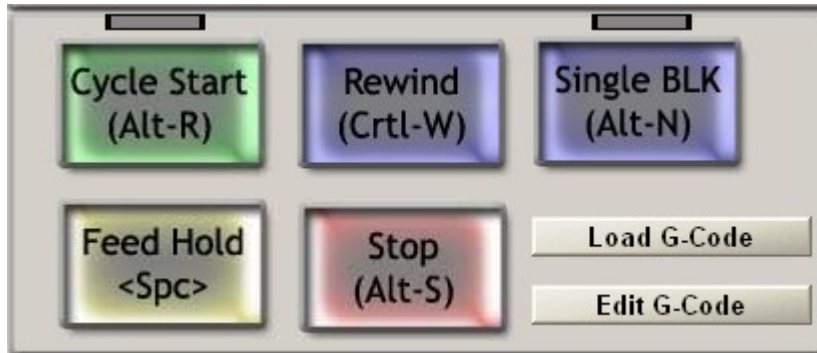


Fig. 5b – The main control screen showing all the various features.

Figure 1 is a view of the main control screen showing all the various features the user has to work with. Each of the major screen features will be described in detail on the following pages.

## **Main Program Control Buttons**



### **Cycle Start**

This executes the current G-code program when pressed. The shortcut keys are Alt-R.

### **Rewind**

This rewinds the current G-code program back to the beginning. The shortcut keys are Ctrl-W.

### **Single BLK**

This activates single line program operation. When pressed the yellow indicator light above the button will flash to indicate it is active. When the Cycle Start button is pressed only a single line of G-code will be executed. The shortcut keys are Alt-N.

### **Stop**

This immediately stops the program or machine movement. The shortcut keys are Alt-S.

### **Feed Hold**

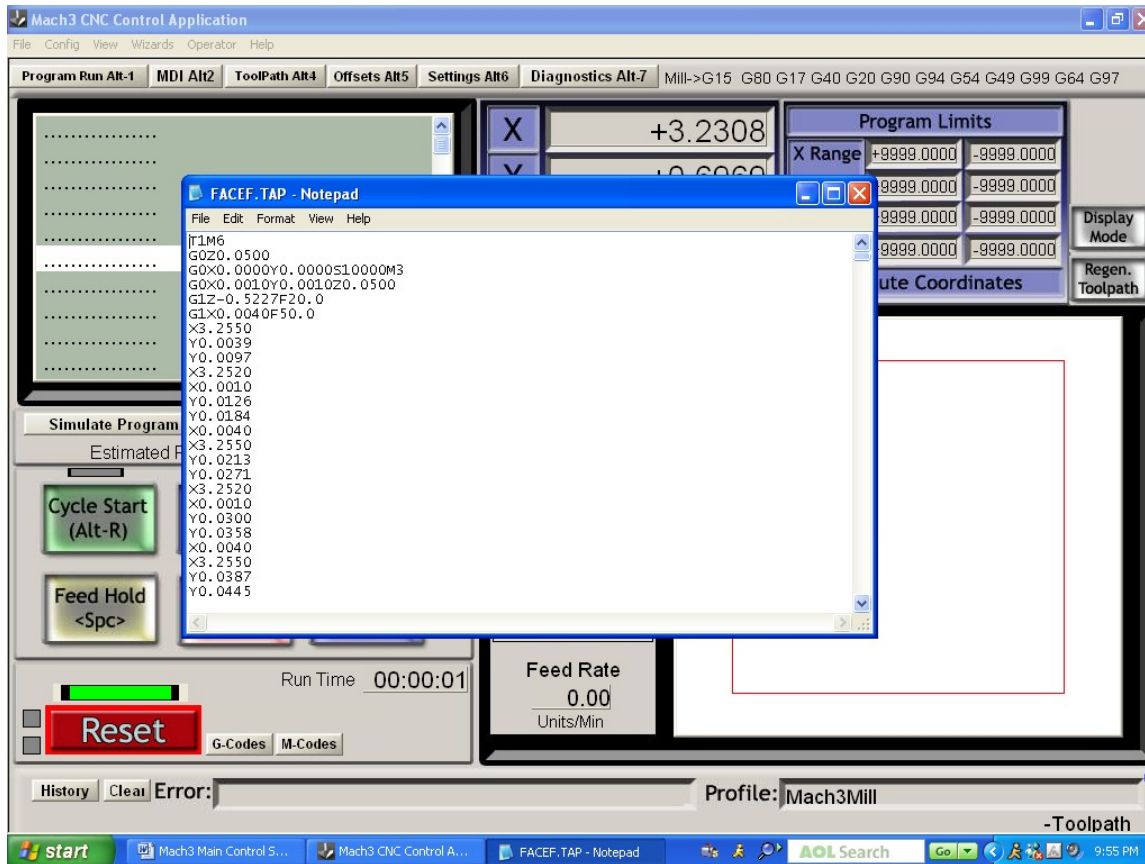
This stops the program after the current line has finished. The shortcut key is the Spacebar.

### **Load G-Code**

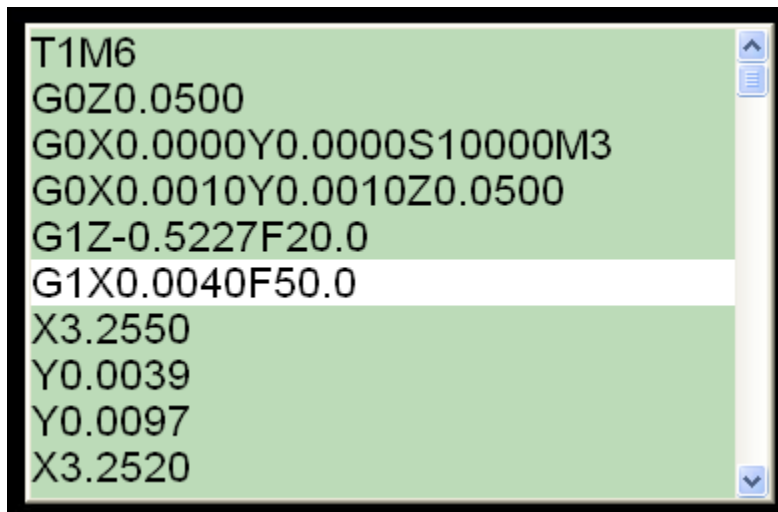
This opens up the standard Open File window that allows a G-code program to be loaded.

### **Edit G-code**

The Edit G-code selection button launches the standard Windows Notepad program displaying the current program loaded as shown below. The user can then edit the G-code program just like a standard Notepad text document. Saving the file immediately changes the currently loaded program upon exit from Notepad. The modified program can then be executed as normal. The program can also be saved as another file name with the standard Save As option, in which case the current program is not modified.



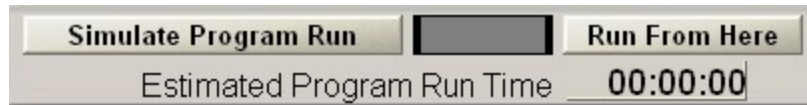
## G-Code Program Display



The G-code program display shows 10 program lines of code. The current line of code that is being executed is highlighted in white. The previous 5 lines of code and the next 4 lines of code are shown so the user can easily follow the program progress. When the G-code program is

stopped the user can click in the display box causing the entire box to become bright. From this point the user can scroll through the program code with the slide bar or mouse wheel.

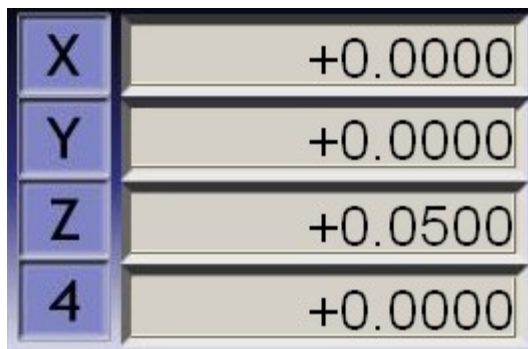
## **Simulate Program Run and Run From Here**



The Simulate Program Run button allows the user to quickly execute the program without any machine movement to get an estimated program run time. This time is then displayed. It is worth noting that this doesn't give a very precise time estimate in many cases since it doesn't account for axis acceleration and deceleration which may add up to a significant amount of time if you have a program with many short movement segments.

The Run From Here button allows the user to highlight a line of code in the program from which to start the program when the Cycle Start button is pushed. The user simply scrolls down through the program until the desired line is highlighted in white. When the desired starting line is highlighted press the Run From Here button and the program lines will be quickly scrolled through until that line of code is reached. The user then simply presses Cycle Start to execute the program from that line.

## **Machine Tool Coordinates**



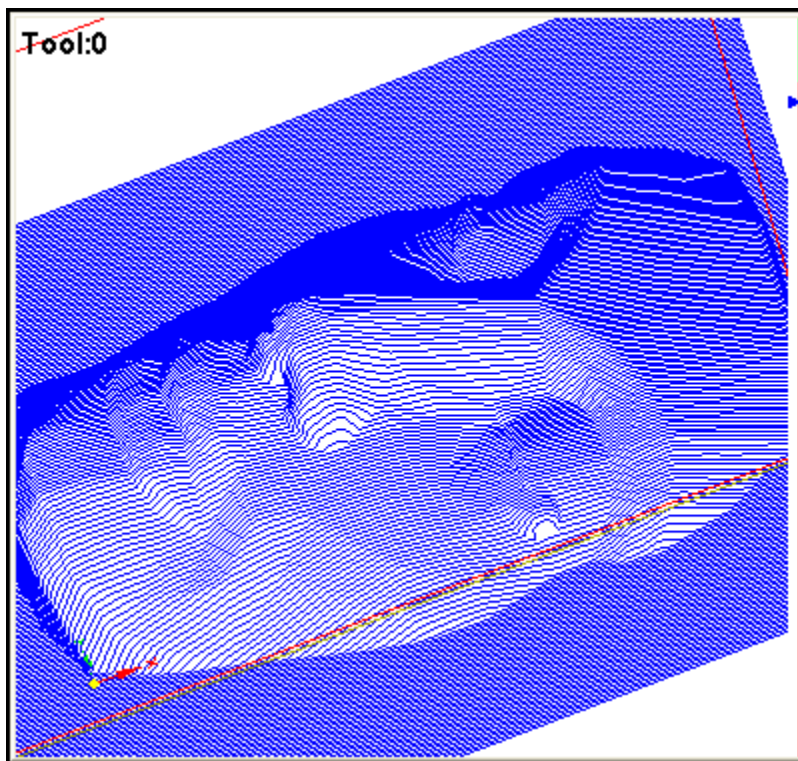
This displays the current X, Y, Z, and 4<sup>th</sup> axis coordinates of the machine tool. The display is updated dynamically as the machine movement occurs. The instantaneous machine position is, therefore, displayed at all times. The coordinate of any axis can be changed by clicking in its box and highlighting it in white. Any number value can then be typed in to change it. The Enter key must be pressed at the end to change it to the new number.

## **Program Limits**

Program Limits		
X Range	+0.0000	+3.2550
Y Range	+0.0000	+1.9756
Z Range	-0.5227	+0.0500
4 Range	+0.0000	+0.0000
Absolute Coordinates		

The program limits display shows the minimum and maximum values for each axis that a part program will move to when executed. It is immediately displayed after the G-code program is loaded.

### 3D Graphic Part Display



The graphic display shows the complete part geometry. The perspective view can be manipulated by clicking on the part with the left mouse button and rotating it with the mouse. The right mouse button allows the user to translate the part view in the window while the mouse wheel allows zooming in and out. A double mouse click resets the view back to a plane view directly down on the part. During program execution the toolpath is drawn over the part geometry as a green line.



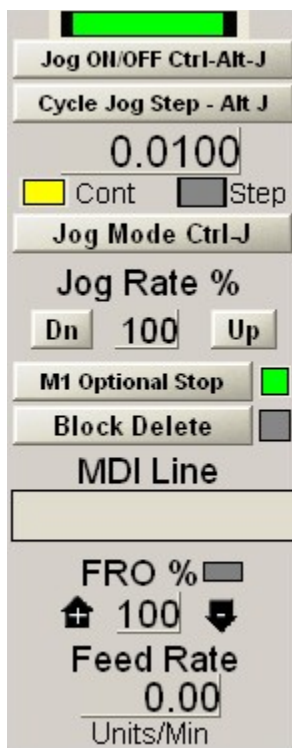
## Graphic Display Buttons



The Display Mode button changes between different graphic display views. Pressing it toggles between the part limit view and machine limit view.

Regen. Toolpath is pressed to redraw the part program to refresh it or when the coordinates are changed to reposition the toolpath display.

## Custom Machine Control Section



### **Jog ON/OFF**

This activates manual jogging of the machine tool via the keyboard. If manual jogging is activated the indicator above the button will be green. The shortcut keys are Ctrl-Alt-J.

### **Cycle Jog Step**

Pressing this button cycles through the 10 preset jog increments that are defined in the Config – State section. The user can also click in the number display box and enter any desired jog

increment number. Each axis will move this jog increment value when its jog key is pressed and the Jog Mode (see next) is set to Step. The shortcut keys are Alt-J.

### **Jog Mode**

This toggles between Continuous and Step Jog Mode as indicated by which indicator light is yellow. In Continuous Jog mode the machine axis is moved at Rapid Speed as long as the axis jog key is pressed. In Step mode the machine axis is moved the jog step distance set with the Cycle Jog Step button. The shortcut keys are Ctrl-J.

### **Jog Rate %**

This sets the jog rate as a percentage of the maximum speed. The maximum setting of 100 makes the jog rate equal to the rapid G0 speed. Use the Up and Down buttons to change the jog rate or click in the box and enter a percentage value.

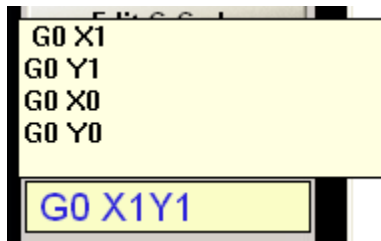
### **M1 Optional Stop**

This activates the M1 optional stop function in a G-code program. When activated the indicator light will be green and all M1 codes in a program will stop the program at that line.

### **Block Delete**

Activates the standard block delete in a G-code program. The indicator light will be green when activated. When activated any line starting with a slash \ will not be executed when the program is run.

### **MDI Line**



This is the Manual Data Input (MDI) line that allows the user to type in a short line of G-code. When the user clicks in the box it is highlighted yellow showing the cursor and also displays a small window showing the last 4 lines of code entered as shown above. A single line of G-code can then be entered. This G-code line is immediately executed upon hitting the **Enter** key. Note: If you need to type in a line of code that exceeds the length of the given box then you must go to the **MDI Screen** for this.

### **FRO %**

This is the Feed Rate Override that displays the percent value used to scale the current Feed rate in the G-code program. The value is increased using the plus arrow to the left of the display while the negative arrow to the right of the display decreases it. The value can also be changed by clicking directly in the display box and Entering a new value. A value of 100 allows the Feed



rates to be used as programmed. Any value other than 100 turns the indicator light yellow to show that the Feed rate is being scaled by the percentage value shown.

### **Feed Rate**

This shows the instantaneous Feed Rate that the axes are moving at. It is a true vector value of the dynamic machine movement velocity. The rate is indicated in Units per Minute. The Units can be in mm or INCH.

## **Other Display Features**



### **Run Time**

This shows the current run time of the program.

### **Prog Line**

This shows the program line number currently being executed or highlighted..

### **G-Codes**

Clicking on this shows a descriptive list of all the G-codes.

### **M-Codes**

Clicking on this shows a descriptive list of all the M-codes.

### **Spindle CW F5**

Clicking this or hitting F5 toggles the spindle on or off just like a standard M3 or M5 command. A flashing yellow indicator shows that the spindle is on.

## **Program Reset**



Clicking the Reset button at any time disables the motor drive circuitry. The indicator above the Reset button will flash between red and green in this disabled state. If the Reset button is hit during a machine movement the motors are stopped instantly and the servos will most likely lose positioning. The machine reference zero, therefore, must be reset upon clicking the Reset button. A solid green indicator shows that the drive is enabled.

## **Example Program Setup and Execution:**

Several example programs are included with the MicroProto Mach 3 program to guide the user through the proper setup and execution of various machining operations. The user is encouraged to spend a sufficient amount of time going through the following examples step by step to become familiar with the various attributes of the Mach 3 program and the MicroMill itself. The part programs will start with the simplest and proceed to more complex machining operations and setups.

### **CAUTION**

The MicroMill DSLS 3000 machining system is a serious industrial machine tool piece of equipment. **DO NOT** proceed with the operation of this machine until you completely understand its use and practice standard machine shop safety such as the following:

**ALWAYS** wear standard Hard Plastic Safety Glasses to protect your eyes when operating machine tools. Material chips or broken cutters can and will follow unpredictable projectile paths toward exposed body parts or unprotected eyes.

**NEVER** wear loose clothing, have exposed long hair or anything that could potentially get caught in rotating or moving machine parts.

**NEVER** start a machining procedure until you have completely simulated or tested the expected motion and know exactly what the end result of the machining operation will be in general.

#### **Example Program # 1 (Engraving Text):**

This first example will demonstrate the simple use of the MicroMill for engraving text. The program file name is **TEXTDEMO.TAP** and will engrave the MicroMill text as shown in figure 8a below. The text letter height is 0.25 inches with a total text length of 2 inches. In this

example you want to use an engraving tool bit or a small diameter flat end mill. Select a 1/16" diameter flat end milling cutter and insert it into the spindle. The cut depth is programmed to be 0.025 inches with an X,Y feed of 20 inch/min. and Z plunge feed of 10 inch/min.

# The MicroMill

Fig. 8a - The MicroMill text engraving demo.

## **Setup Procedure:**

1. Execute the Mach 3 program by double clicking on the Mach3Mill Icon.
2. Select the continuous jog mode by pressing the Jog Mode button.
3. Using the Up/Down and Left/Right keyboard keys jog the spindle to the approximate center of the material.
4. Select each axis and Jog the machine to a safe material loading position.
5. Secure Sample Material # 1 plate (3" x 2" x 0.25") in the vise and tighten it down.
6. Fast Jog the X and Y axis to the approximate center of the material.
7. Fast Jog the Z to approximately 0.5 inches above the material surface.
8. Click in the Jog Rate box and enter 25 to slow it down. Jog the tool bit to 0.1 inches above the material surface.



9. Click in the Jog Rate box and enter 5 and then slow jog the tool bit down until it just touches the surface.
10. Click in the Z-axis Machine Tool Coordinate box and enter 0. This sets the surface of the material right at 0.

X	+0.0000
Y	+0.0000
Z	+0.0000
4	+0.0000

11. Click in the Jog Rate box and enter 100 to set it back to fast jogging.
12. Click in the MDI Line box and enter G0Z0.1.

MDI Line

13. The machine is now ready to execute the part program.

### **Executing the Program:**

You are now ready to load and execute the program with the following steps.

1. Click the **Load G-Code** button.

Load G-Code
Edit G-Code

2. Type **\*.TAP** in the File name box. This will select only the **.TAP** files. Select the G-code folder in the Mach 3 directory.
3. Select the program **TEXTDEMO.TAP** and double click it or hit Enter. The text should now show up in the graphic display window.
4. At this point it is a good idea to Dry Run the part before any cutting is done. This allows the user to verify that the motion looks right to prevent damage to the tool bit or work piece if something was not done right.
5. In the MDI line box enter G0Z1 to move the tool bit 1.0 inch above the material surface. Click in the Z-axis Machine Tool Coordinate box and enter 0. This will set the Z-axis 0 reference point 1.0 inch above the material surface. The machine is now setup to DRY RUN the part which executes the X and Y axis movements with the Z-axis movements offset up a specified safe height of 1.0 inch from the material surface.
6. Click on the large green **Cycle Start** button to execute the program.



The machine will execute the X,Y and Z movements at this time with the Z-axis position offset 1.0 inch upward. If the movement looks correct with no collisions with fixtures or anything else you are now finally ready to start machining the part. The following machines the part program.

1. Set the Spindle Speed to the Top pulley position as shown in figure 9.
2. Click in the Z-axis Machine Tool Coordinate box and Enter 1.1000 to add 1.0 inch to the position. The machine will now know that it is this far above the material surface.
3. Click on the **Cycle Start** button to execute the program.

The machine tool will move to the start position, turn on the Spindle and begin machining the part.

**CAUTION:** Always wear your **SAFETY GLASSES** when inspecting a machining process up close. Chips can and will fly off randomly in all directions and will surely injure your eyes if they are unfortunate enough to be in the path of a chip projectile.



Fig. 9 - Spindle set to top speed position.

**NOTE:**

Once you become familiar with the machine and setting up the reference positions and so forth you may bypass the Preview and Dry Run steps and machine the part directly. The preceding process, though, is ALWAYS A GOOD IDEA when setting up a new part, experimenting with different Feed values or changing to a new fixture location.

### **Example Program # 2 (An Ellipse):**

In this second example the user will machine a 2D profile part. The sample program is an elliptical profile (**ELLIPSE.TAP**). This program was generated as a combination of several G02 circular interpolation moves. Hence, the circular interpolation capabilities of the machine will be demonstrated in this example. The cut depth is set at 0.050 inches with a Z plunge feed of 5 inch/min and an X,Y feed rate of 10 inch/min. Select a 1/8 inch diameter flat end mill as the cutting tool and follow the same setup and run procedure as in example # 1.

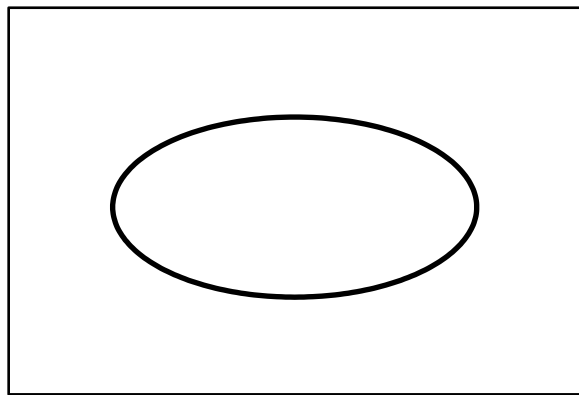


Fig. 10 - Part profile representation of ELLIPSE.TAP.

### **Example Program # 3 (Drilling Holes):**

The next example shows the use of the MPS2003 program in drilling hole patterns using standard G83 peck drilling cycles. The following bolt hole pattern as displayed in figure 11 below will be generated. The program name is **BOLTHOLE.TAP** and contains 10 evenly spaced holes around a 1.0 inch diameter circle with a final drill depth of 0.200 inches. The program was generated in the Cad/Cam machining package BobCad™ using the canned bolt hole pattern function.

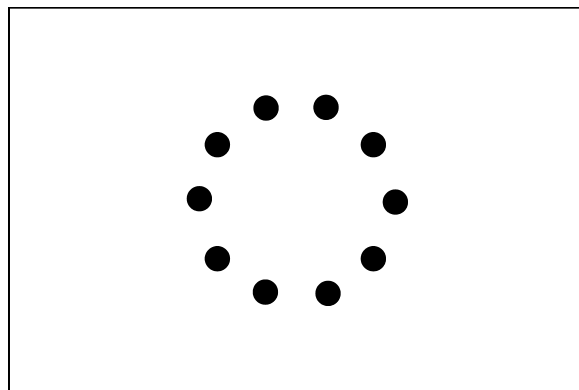


Fig. 11 - Drilling a bolt hole pattern using G83 peck drilling cycles.

Load sample material # 2 in the vise and clamp it down securely. Set the reference (0,0,0) position at the center of the material. Use a 1/8 inch diameter drill or flat end mill and follow the same setup as before.

**Example Program # 4 (3D Surfacing):**

The next sample illustrates the program capabilities in machining a complex 3D surface profile. A solid model of the finished part profile is shown in figure 12 below. If you are a MillWizard™ user you will recognize this as the sample Nefertiti.3ds file. Hopefully at the end of this exercise you will have machined a similar looking part.



Fig. 12 - Solid model representation of the finished 3D surface profile part FACEF.TAP.

**Machining Description:**

This example will consist of two machining operations being that of Roughing and Finishing. This requires a tool change and will, therefore, also demonstrate how to set a new tool reference. The part profile is machined by starting in the lower left hand corner of the part and machining the XZ profile until the right hand side is reached. The Y axis is then incremented positively a small fraction of the milling cutter diameter and the XZ profile is machined again back to the left side. This process is repeated until the upper right hand corner is finally reached. This part was designed to be machined in this raster scan method using the features available in MillWizard™. The process is schematically indicated in figure 13 below.

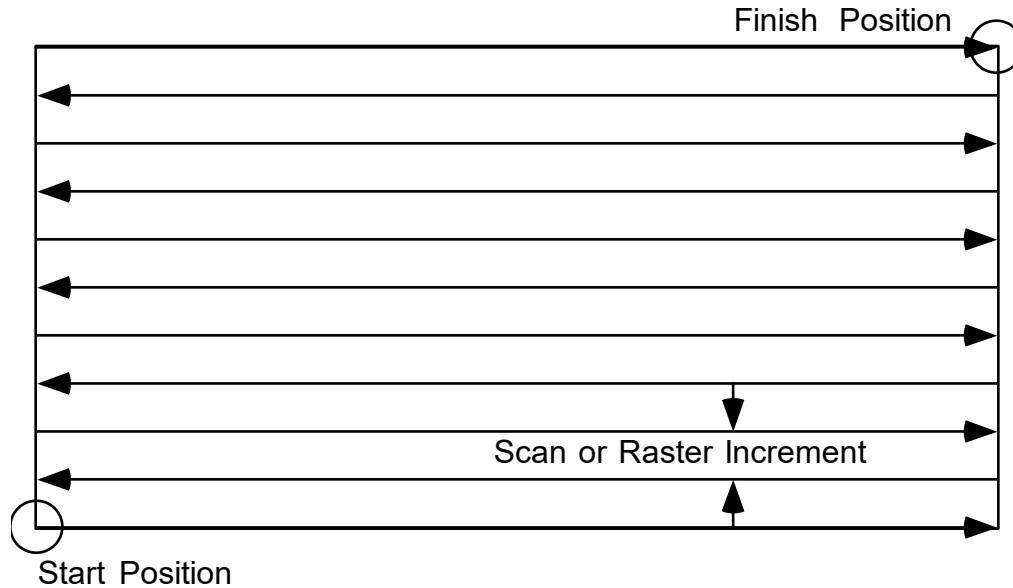


Fig. 13 - Raster method used to machine the 3D surface profile.

#### Roughing Operation:

The first program file that will be executed is the Roughing Operation named **FACER.TAP**.

The purpose of this roughing operation is to use a large diameter milling cutter to remove a large amount of material. This process leaves the part profile in a 'Rough looking state' that must next be machined to the final form with the Finishing Operation. The following procedure applies.

1. Insert a 3/16 Ball End milling cutter into the spindle. Note: A 2 flute flat end mill will work just as well for this roughing operation.
2. Load sample material #3 as explained before and set the (0,0,0) position reference at the Lower Left Corner of the material as shown in figure 14 below. The Z = 0 position should be set directly at the surface. The X = 0 position is set at the edge and the Y = 0 position is set 0.25 inch inward from the side so the milling cutter does not cut into the vise jaws.



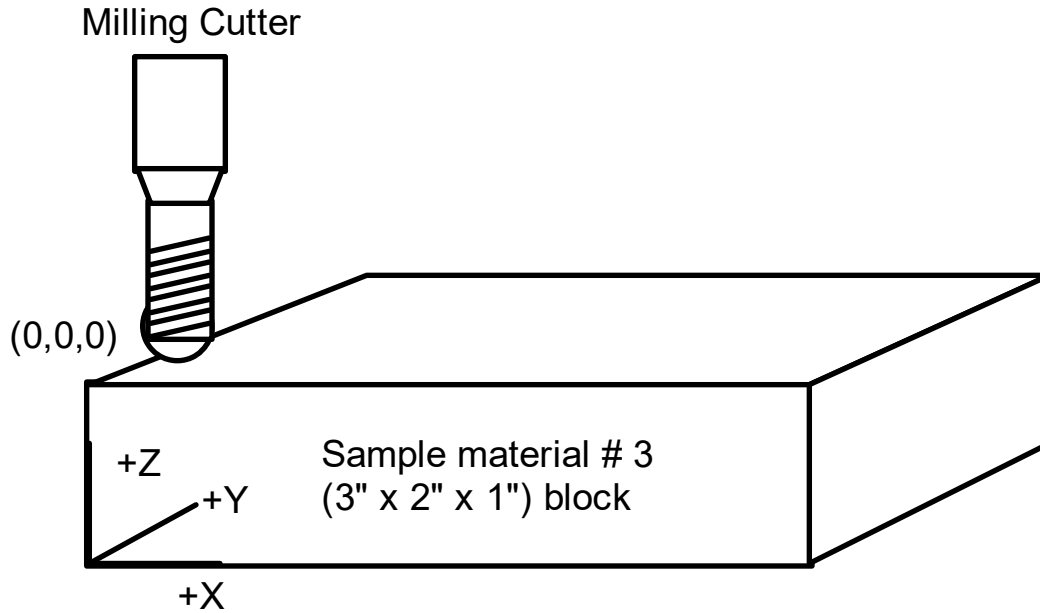


Fig. 14 - Setting the position reference for machining the part program FACER.TAP

3. Load the program **FACER.TAP** as explained before
4. Execute a Dry Run as before if necessary to check for collision avoidance.
5. Set the belt to the 3<sup>rd</sup> pulley position groove from the top and turn on the spindle motor.
6. **RUN** the program.

The total machining time for this should be approximately 45 minutes. During this time it is a good idea to 'keep an eye' on the machining operation to make sure it is performing correctly. A crash or jam could occur if the Feed was set improperly, the milling cutter is dull or not tightened into the collet properly, a power line spike occurs or some other unexpected problem happens.

**CAUTION:** Always wear your **SAFETY GLASSES** when inspecting a machining process up close. Chips can and will fly off randomly in all directions and will surely injure your eyes if they are unfortunate enough to be in the path of a chip projectile.

#### Finishing Operation:

Next the finishing operation must be performed to create the final smooth form of the contour surface. This is the file **FACEF.TAP** and requires the use of a 1/16" Ball End Mill. Put a 1/16" Ball End Mill into the spindle and, using SLOW JOG, touch it down on the part of the material surface that was not machined by the roughing operation.. Zero out the Z-axis ONLY leaving the X and Y axes the same. This is an easy method of setting a new tool length. Jog the Z-axis above the part surface 0.100 inches. Set the pulley to the high speed or top position and execute the part program as before.

## G - CODES

The following are all of the supported G-Codes used by the Mach 3 program.

### **G00**

Set all axis movement to rapid.

#### Example:

G00 X0.0Y0.0Z1.0

Rapid to the coordinate (0,0,1).

### **G01**

Linear interpolation movement at the specified feed rate F.

#### Example:

G01 X2.0Y2.0F10.0

Starting at (1,1) in absolute mode this moves the machine to (2,2) at a feed of 10 in/min.

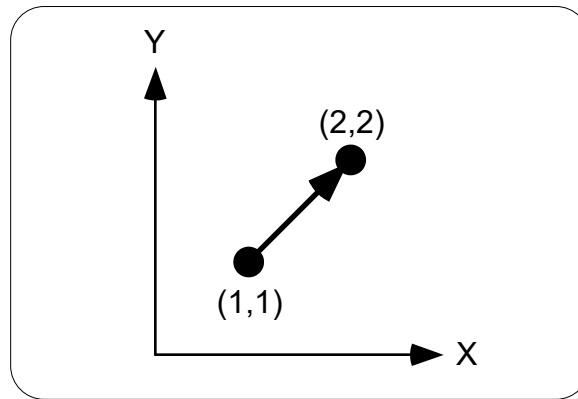


Fig. 15 - G01 X2.0Y2.0F10.0 movement from coordinate (1,1) to (2,2).

### G02

Clockwise circular interpolation at the specified feed rate F.

Example (Absolute Positioning Only):

G02 X2.0Y0.0I1.0J0.0F5.0

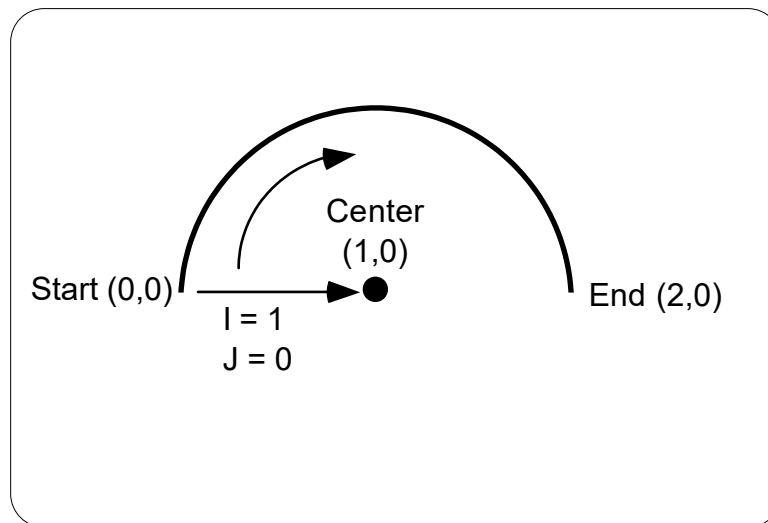


Fig. 16 - G02 Clockwise Circular Interpolation Move

### G03

Counter clockwise circular interpolation at the specified feed rate F.

Example (Absolute Positioning Only):  
G02 X0.0Y0.0I-1.0J0.0F5.0

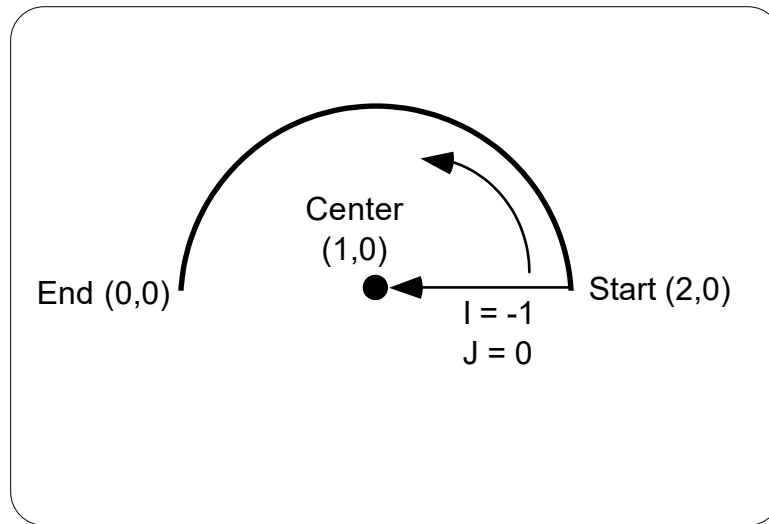


Fig. 16 - G03 Counter Clockwise Circular Interpolation Move

### **G17**

Select the X,Y Plane for circular interpolation moves.

### **G20**

Select INCH mode programming. All values encountered will be interpreted as distances in inches and feed rates as inches/minute.

### **G21**

Select metric mode programming. All values encountered will be interpreted as distances in mm and feed rates as mm/minute.

### **G43**

Tool Length Compensation

#### Example:

G43 Z0.1 H1

#### Description:

The G43 command adds the specified tool offset length to all the following Z-axis moves.

### **G81**

Drilling Cycle

Description:

G81X1.0Y1.0R0.1Z-0.25F5.0

The drill cycle moves to the (X,Y) location of the hole start point and then rapids to the R plane. It then proceeds at the feed rate F to the final drill depth. The process is completed by a rapid move back to the R plane or I plane.

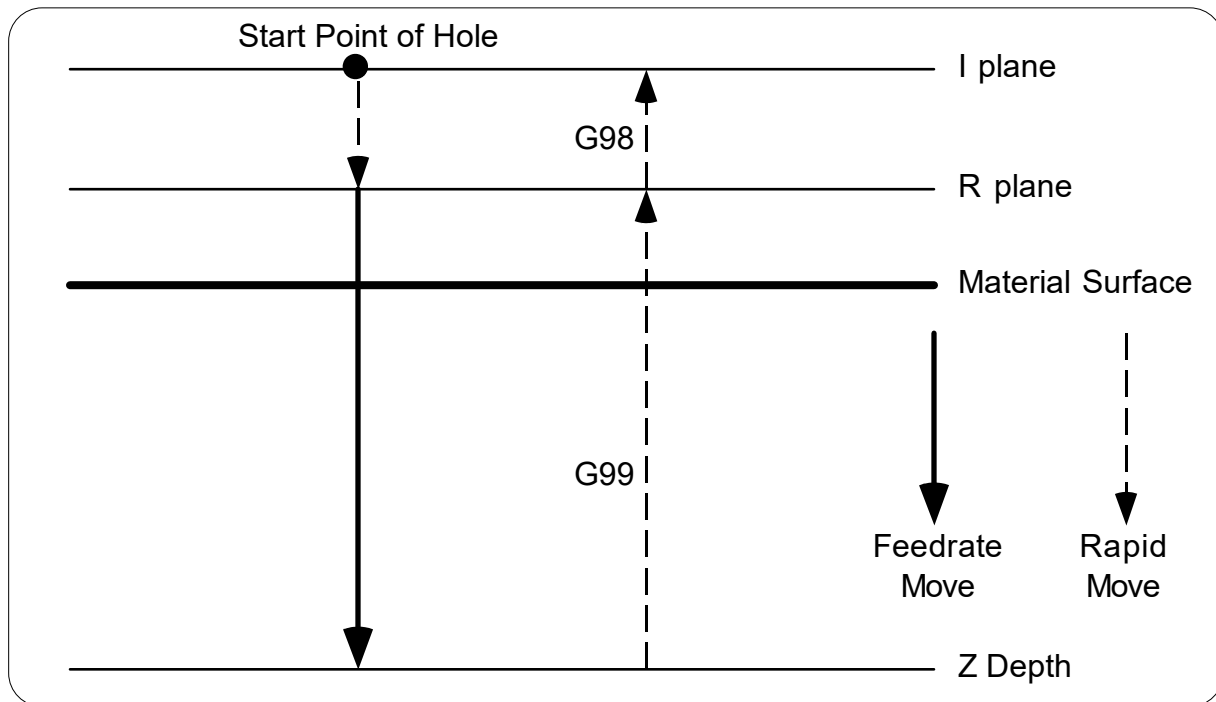


Fig. 17 - G81 Drill Cycle Operation

**G83**

Peck Drilling Cycle

Description:

The peck drilling cycle is a necessity anytime deep holes must be drill. Drilling a deep hole continuously to the hole depth would surely fail since chip load buildup occurs. The drill must be retracted to let the chips out and then re-enter the hole and proceed toward the drill depth. The peck drill cycle allows just this process whereby the drill retracts and re-enters the hole repetitively each time drilling the hole a specified amount deeper until the final depth is reached. Figure 18 shows the process for the following command line which is described step by step below.

G83X1.0Y1.0R0.1Z-1.0Q0.25F5.0

1. Move to the X,Y coordinate (1.0,1.0).

2. Start the drill process at the I plane ( Initial plane).
3. Rapid down to the R plane.
4. Feed down the incremental depth Q.
5. Rapid back to the R plane ( if G99 is activated ) or the I plane ( if G98 is activated ).
6. Rapid down to the point D (default value = 0.025 inch) above the last Q increment.
7. Feed down the next incremental depth of D+Q
8. Rapid back to the R or I plane.
9. Repeat steps 4 to 8 until the final drill depth is reached finishing with a Rapid back to R or I.

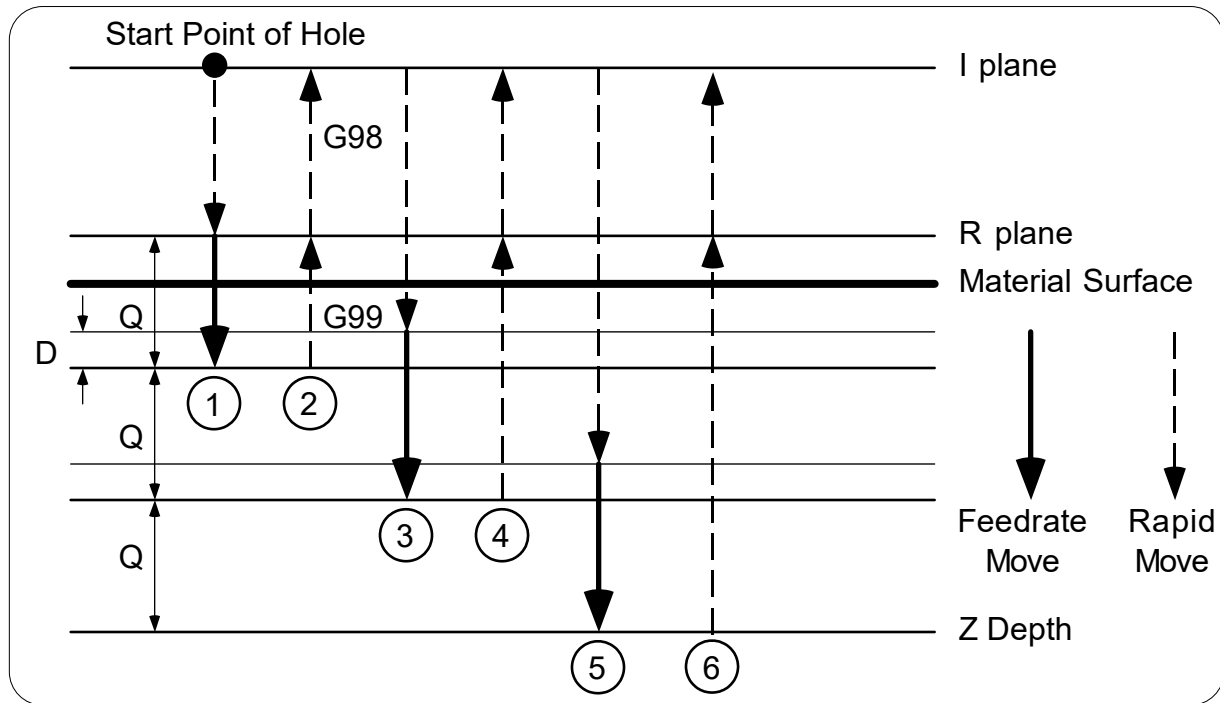


Fig. 18 - G83 Peck Drill Cycle Operation

### G90

Set absolute positioning mode

### G91

Set incremental positioning mode

### G92

Set new (X,Y,Z) reference position.

Description:

G92 X1.0Y1.0Z0.0

This sets the current machine tool position coordinate to (1,1,0). This is the new absolute reference position that all moves will be referred to now.

Example:

The G92 command can be very useful for repeating a part program at numerous locations. Suppose you are manufacturing small diamond inlays from wood and wish to cut out 4 per each piece of wood block you have. You have defined the bottom vertex as the reference position for the start of each piece. Figure 19 shows the diamond patterns with the starting coordinates of each piece you wish to produce.

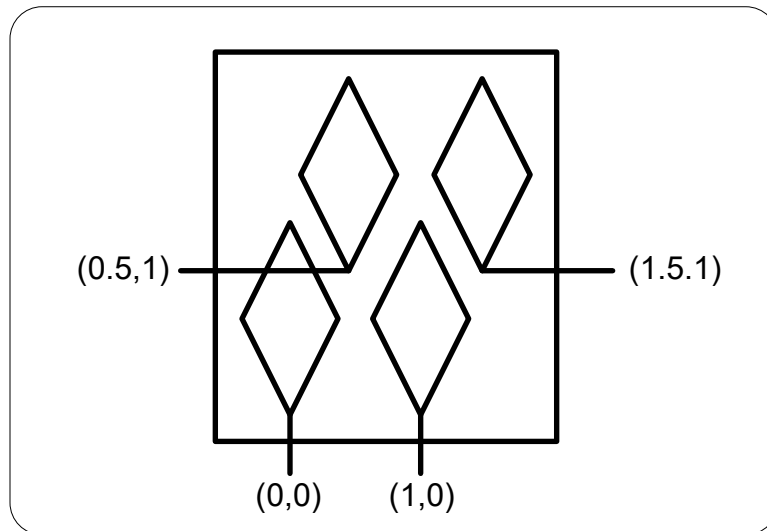


Fig. 19 - Machining multiple diamond inlays.

```
M97 P100
G92 X-1.0Y0.0
M97 P100
G92 X-0.5Y-1.0
M97 P100
G92 X1.0Y0.0
M97 P100
G00 X-0.5Y-1.0
G92 X0.0Y0.0
M02
```

```
N100
G00G90X0.0Y0.0Z0.1
G01 Z-0.13 F10
X0.375Y0.75
X0.0Y1.50
X-0.375Y0.75
X0.0Y0.0
Z0.1
```

M99

**G98**

Return to I plane for drill cycles.

**G99**

Return to R plane for drill cycles.

## **M - Codes**

**M0**

Program Stop. Upon reading an M0 command the program stops and waits for the user to press C to continue program execution.

**M6**

Tool Change. Upon reading an M6 command the program stops and waits for the user to press C to continue program execution.



## AXIS CONFIGURATION

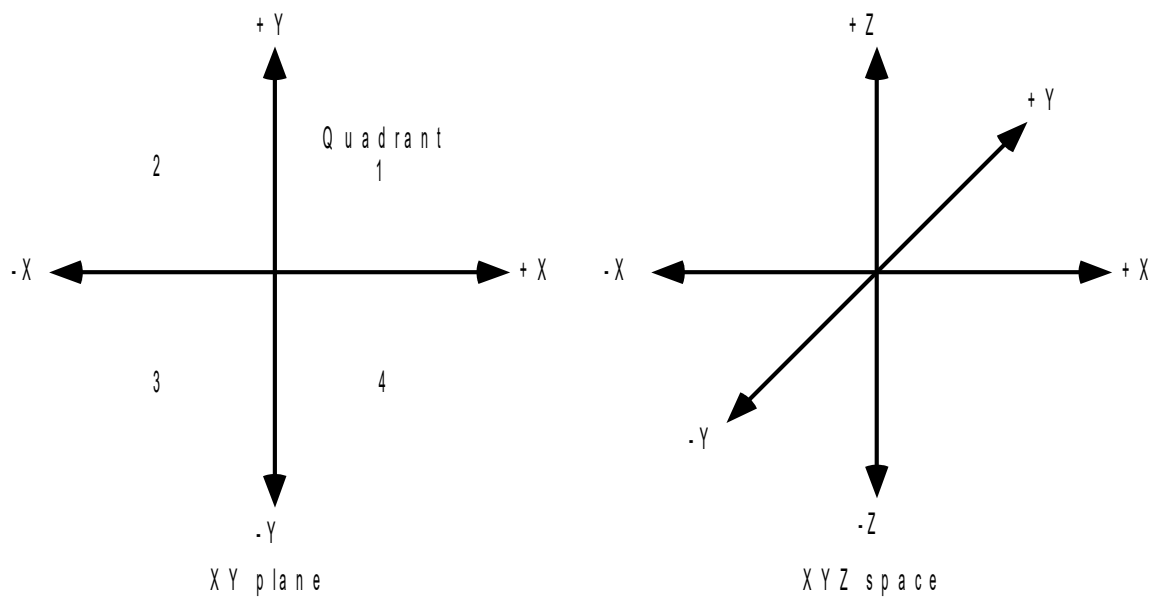


Fig.20 - The Cartesian Coordinate System

### CNC MACHINE AXIS CONFIGURATION:

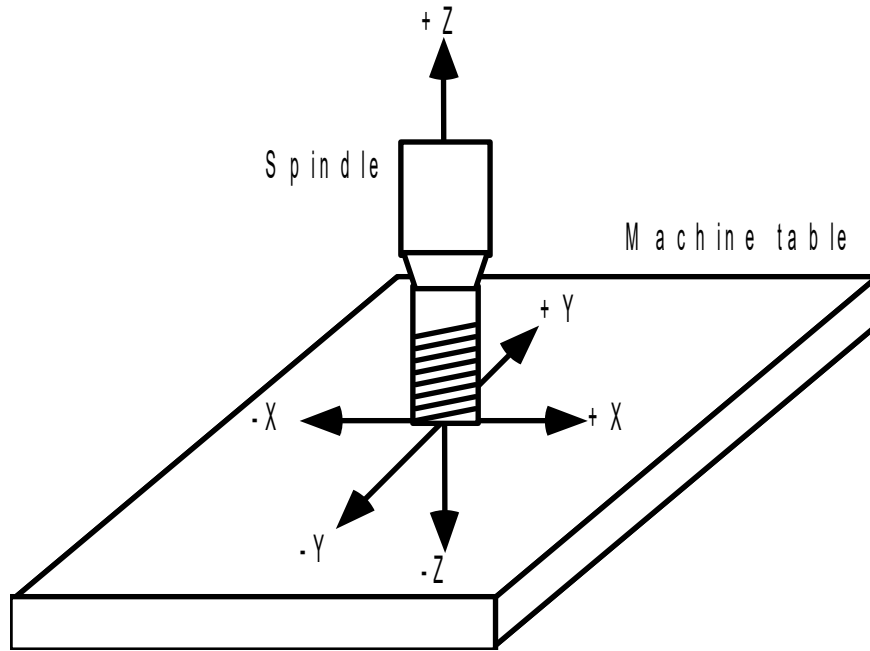


Fig.21 - Machine axis system for 3 axis Vertical CNC Machine  
(Machine axis defined by spindle movement)

**Rotary Table or 4<sup>th</sup> Axis Option:**

The 4<sup>th</sup> Axis Rotary Table is mounted on the left side of the X-axis table as shown in fig. 22 below. The rotary table axis is specified with the letter A and is programmed in the same fashion as the X,Y and Z axes with all movements denoted in degrees. Positive rotations are in the CW direction. The resolution of the rotary table is 1/80<sup>th</sup> or 0.0125 degrees of rotation. The following command line would rotate the table 45 degrees at the specified feed rate using absolute positioning mode.

G01 A45

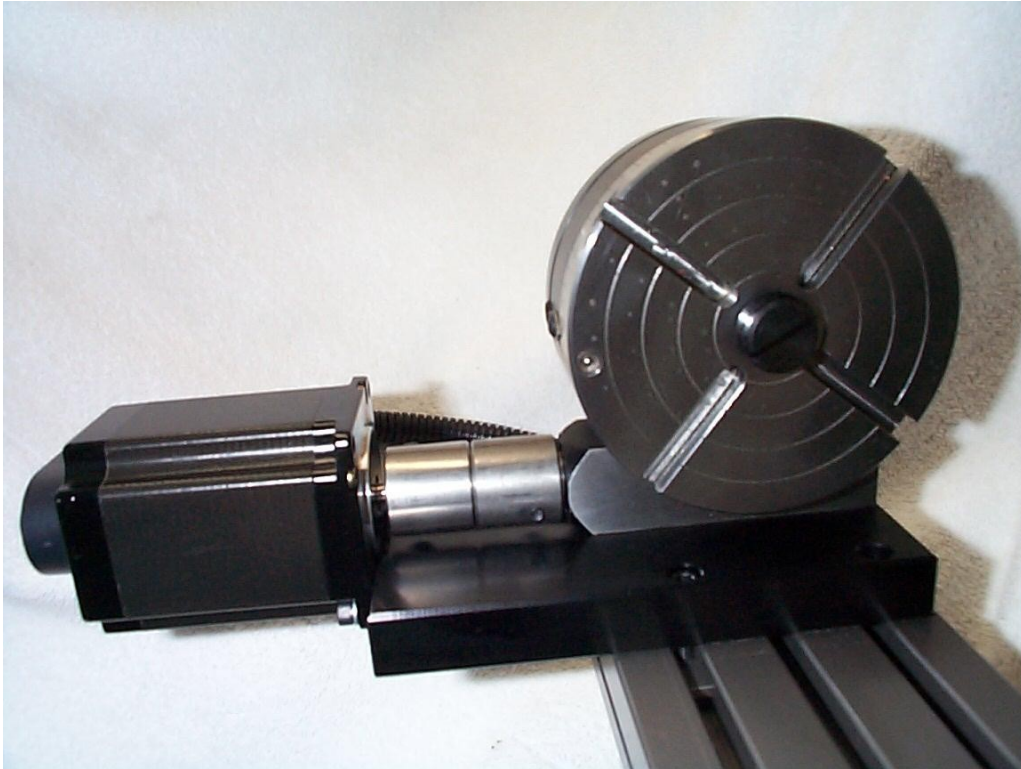


Fig. 22 - Standard mounting position of the 4<sup>th</sup> Axis Rotary Table.

## **TROUBLESHOOTING PROBLEMS**

**PROBLEM:** System does not work at all.

**CHECK:** Fuse in back of unit (2A).

**PROBLEM:** The motors don't rotate at all when jogging.

**CHECK:** If all the axis step motors don't rotate and just vibrate when jogging most likely there is a problem with the **PRINTER CABLE** or **PRINTER PORT** of your computer. Try another cable and make sure no cable or port pins are bent out of shape.

- PROBLEM: Can't get table to position at all - the motors just don't work.  
CHECK: The PORT BASE is probably configured to the wrong address. Change the port address in the Ports and Pins section and change the port #1 PORT BASE to either 0x278, 0x378 or 0x3bc.
- PROBLEM: Motors run at slow speeds with low torque, but not at high speeds at all.  
CHECK: Open up drive unit and check the 3 Amp in-line fuse (Fig. 23 below) first. If the fuse is good, measure the voltage across the shorter capacitor with RED wires (Fig. 23) to see if there is approx. 55 Vdc. If not the in-line fuse may not be making a good connection - correct if necessary.
- PROBLEM: Motors have low torque or are stepping incorrectly  
CHECK: Voltage Line Power to see if voltage is 110 to 120 Vac.
- PROBLEM: The step motors are humming at a low frequency pitch.  
CHECK: This is the high voltage chopping frequency of 60 Hz. This is normally no problem, but may become a problem if the AC line Voltage drops below 110 VAC. If the in-line fuse keeps blowing this is the case and the trimmer potentiometers on both circuit boards must be adjusted CW looking at the unit from the front panel.
- PROBLEM: The X, Y or Z axis is not stepping correctly.  
CHECK: This could be a problem with the parallel port of the computer. Check first if all the bit lines of the computer port are working correctly. If this checks OK then there may be a problem with the driver unit. Call the factory in this case.
- PROBLEM: The motors will not step at high speeds - they just keep jamming.  
CHECK: This is inherent in any system pushed beyond its capacity. The load on the table may be too large, the screw or ways may not be properly lubricated, the lead screw nut is too tight or the gib adjustment may be too tight. Under ideal conditions a single axis should achieve a maximum translation of 60 inch/min. Simultaneous two axis movement should also approach this, but may have to be a little slower.
- PROBLEM: Table is not moving what it is programmed to move.  
CHECK: You may have switched to INCH or mm mode unknowingly. Type U and select the units you wish to use.
- PROBLEM: Driver Unit gets fairly hot after prolonged idle period.  
CHECK: This is normally no problem, but if it is going to sit idle for extended periods of time it is best to DESELECT the table or turn the unit off.
- PROBLEM: The step motors get very hot after prolonged use or idle time.  
CHECK: This is most evident when running the motors at high speed which

drives them with the maximum power. Even under these conditions the motors are dissipating less than their maximum power rating and no danger should be present. To be safe, though, the motors should be DESELECTed if a prolonged idle period is anticipated.

**PROBLEM:** The motors jam while jogging at high speed or when executing a G0 rapid movement.

**CHECK:** The velocity is typically set to 60 In/min. while the acceleration is set between 2 5 In/sec/sec.

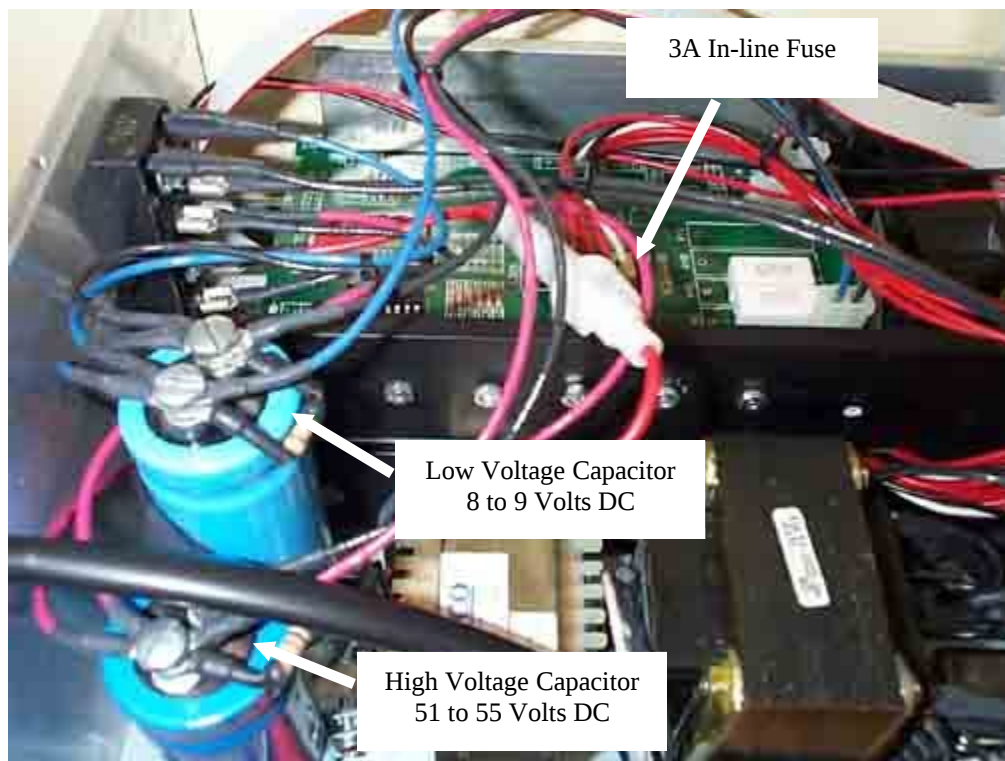


Fig. 23 - Checking Voltages inside the Electronic Driver Box. Shown are the range of voltages that will be present for a normally operating system.

# MACHINE MAINTENANCE

Proper maintenance and care will assure that your MicroMill system will work at peak performance for many years of operation. Following the simple maintenance procedures below is all it takes to keep your machine in good running condition.

## **Lubricating the Ways and Lead Screws:**

The steel ways and lead screws of the machine should be well lubricated at **All Times** to provide smooth, free movement. Standard AUTOMOTIVE TRANSMISSION FLUID ( figure A1 ) is readily available and is an excellent lubricant for this purpose. Using a standard oil can, apply generous amounts to the Steel Sliding Ways and Lead Screws before and after machining operations as shown in figure A2 below.



Fig. A1 - Standard Automotive Transmission Fluid and Oil Can.



Fig. A2 - Applying Lubricant to Steel Ways and Lead Screws.

**Backlash Measurement:**

The backlash of each axis can be measured by mounting a Dial Indicator to a secure base with the contact point against the moving member as shown in figure A3 below.



Fig. A3 - Measuring the X-axis Backlash.

The following procedure will determine the backlash for the X-axis:

1. Enter the JOG INC mode and set the jog increment to 0.100 inches.
2. Jog the X-axis in the negative direction 0.100 inches.
3. Position the dial indicator against the table as shown in figure A3 enough to deflect the needle approximately 0.050 to 0.100 inches.

4. Zero out the indicator.
5. Jog the X-axis in the positive direction 0.100 inches.
6. Note the difference of the dial indicator from Zero.
7. This difference is then the backlash of the X-axis.
8. Enter this backlash directly as a value in INCHES in the Config - Backlash section.
9. Enter the Backlash Speed % value between 5 and 25 %.

**Note:** Typically the backlash is between 0.00015 to 0.003 inches, but can be as high as 0.005 inches or more due to machine wear. If the backlash is this high it can be adjusted as described in the section below. Also note that G64 continuous contouring or any high speed movement that involves many direction changes will not work with the backlash compensation on.

### **Adjusting the Lead Screw Nuts:**

If your machine is running on a continuous basis and has accumulated many thousands of inches of travel on each axis throughout the years the Lead Screw Nuts may begin to show wear. This is evident if the backlash or play is larger then you think it should be. The backlash of a new system is typically from 0.0015 to 0.003 inches. If the backlash is larger than this the Nut may need to be tightened down a little. The split bronze nut assembly shown in figure A4 is used on the X and Y axes of the machine.

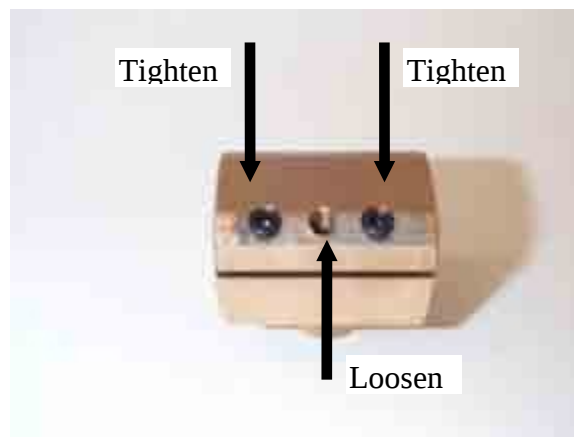


Fig. A4 - Tightening the Bronze Lead Screw Nut.

The outer two hex screws tighten or pull down the nut while the center hex screw loosens or pushes it open. This push-pull arrangement has to be adjusted very precisely. If the Nut is too tight it will cause too much friction and will be very difficult to move the axis. If it is too loose the axis will turn very freely, but the backlash will be too large for precise axis positioning. The adjustment of these hex screws is very, very sensitive to the slightest turn. They should be adjusted NO MORE than 1/30 of a Turn at a time and tightened by only the force you can produce with your thumb and forefinger on the small end of the L-shaped hex wrench.

### **Tightening Procedure:**



1. Loosen (CCW turn) the center hex screw slightly (1/30 turn or less).
2. Tighten (CW turn) down the outer two hex screws an equal amount.
3. Re-Insert the Lead Screw and test for tightness or freedom of movement and Backlash.
4. Repeat 1 to 3 until desired performance level is obtained.

**Loosening Procedure:**

1. Loosen the outer two hex screws slightly (1/30 turn or less).
2. Tighten down center hex screw an equal amount.
3. Re-Insert Lead Screw and test movement freedom and backlash.
4. Repeat 1 to 3.

**Note:**

This can be a Very Tedious process and can really throw the whole Lead Screw/Nut system out of calibration ( Much Too Tight or Too Loose) if not done in the proper fashion. If you are not very experienced with 'The Feel' a properly adjusted machine should have it is recommended that you do not attempt this procedure.

**Adjusting the Brass Gibs:**

**Box Way Assembly Gib:**

The brass gibs may also need adjustment over time. The brass gib of the Z-Axis Box Way Assembly is adjusted with Hex Screws on the top and bottom of the assembly as shown in figure A5.1 below. The top screw tightens the gib on the steel way while the bottom one loosens it.



Fig. A5.1 - Box Way Brass Gib top adjustment screw.

**Saddle Gib:**

The brass gib on the saddle or Y-axis is adjusted with the two Hex Screws shown in figure A5.2 below. The longer middle Hex Screw is used to lock the saddle in place, which would generally NEVER be done on a CNC machine. This would only be used on the Manual MicroMill.



Fig. A5.2 - Brass Gib adjustment screws for the Saddle.

#### Table Gib:

The tapered brass gib on the table or X-axis is adjusted by the left side and right side Hex Screws shown in figure A5.3 below. Adjusting the left screw inward will tighten the table while adjusting the right screw inward will loosen the table. The longer middle Hex Screw is used to lock the table in place, which would generally NEVER be done on a CNC machine. This, again, would only be used on the Manual MicroMill.



Fig. A5.3 - Tapered Brass Gib adjustment screws for Table.

### **Adjusting the HeadStock:**

The Headstock is removed or re-positioned on the dovetail by loosening up the two Hex Screws shown in figure A6-A below. It is Very Important to NOT Over-Tighten these two screws when setting it back in place. Only a small tension is required to securely fasten the dovetail. Tighten it only with the amount of force you can provide by holding the short end of a standard L-shaped Hex Wrench between your Thumb and Forefinger as shown in figure A6-B. If the tension is too tight the spindle will not turn freely and may be damaged due to over-heating. This will also prevent it from running at the top (high speed, low torque) spindle speed.

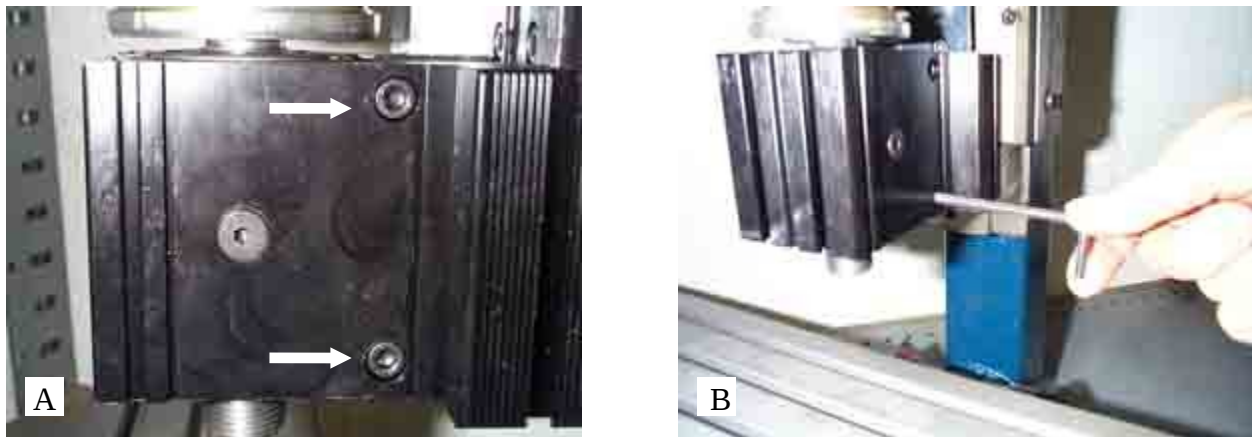


Fig. A6 - (A) Headstock Dovetail adjustment hex screws  
(B) Tightening with only the Thumb and Forefinger

**Note:** If you have accidentally over-tightened the spindle dovetail you may have ‘crunched down’ the bearing race assembly. This would be evident if it doesn’t start on the Top Spindle Speed. If this is the case you will have to ‘loosen up’ the bearings again by setting it at the 2<sup>nd</sup> Top Spindle Speed and letting it continuously run overnight for a period of 12 hours or more. The bearings should loosen up after this and it should be able to start on the Top Spindle Speed. If not, the spindle bearing race may be permanently damaged and will have to be replaced.

## IMPORTING BOBCAD FILES

Standard BobCad G-code files can be used by the MPS2003 program. The files must be post-processed with the following procedure and options in BobCad.

1. Generate a BobCad part drawing.
2. Enter the **NC-Cam** window.
3. In **Setup** make sure the following parameters are configured (These should just be the standard default values).
  - a. Coordinate Values = Absolute
  - b. Coordinate Output = Non-Modal
  - c. G-Code = Modal
  - d. G02 G03 Circular = Incremental I J
  - e. Circular Movement = Full
  - f. Spacing = ON
  - g. Coordinates set to 4 digits
4. Generate the G-Code with the **Machine All** option.
5. Save the file with the Tape or **.TAP** option.

This will produce a standard file that can then be imported into the MPS2003 program.

## DIGITIZING PROBE OPERATION

The Digitizing Probe program MPSPROB3.EXE must be run and is entered with option **E** from the main menu. It has the following options.

### PROBE PROGRAM:

**1 = DIGITIZE**

**2 = SET PARAMETERS**

**3 = EXIT**

**OPTION 1** Start the digitizing process

**OPTION 2** Set the digitizing probe parameters

DIGITIZING METHOD = 1

1 = XY RASTER, 2 = XY SPIRAL

### METHOD 1 - Figure D1:

The probe moves in the negative Z direction until it makes contact with the material surface. It then advances along in the positive X direction following the XZ surface profile. When the X Limit is reached the probe advances in the positive Y direction the Y Raster Distance and proceeds in the negative X direction. This rastering process continues until the final X Limit and Y Limit is reached as shown in figure D1.

### METHOD 2 - Figure D2:

The probe moves in the negative Z direction until it makes contact with the material surface. It then proceeds outward from this center point in a square spiral pattern tracing the XZ and YZ surface as shown in figure D2.

X LIMIT = 2

Sets the maximum X dimension the probe will move to 2 inches.

Y LIMIT = 1

Sets the maximum Y dimension the probe will move to 1 inches. Not used for method 2.

MAX Z DEPTH = -1

Sets the maximum depth the probe will move in the negative Z direction until a surface is contacted. If this depth is reached without contacting a surface the probe will proceed in the X direction.

Z TEST DISTANCE = 0.0005

Set the Z axis sample distance to 0.0005 inches.

X TEST DISTANCE = 0.01

Sets the distance the X axis advances between contacts to 0.01 inches. For method 2 this is both the X and Y increment test distance.

X STORE DISTANCE = 0.05

Records the X,Y, and Z coordinates with the X resolution of 0.05 inches. For method 2 the X STORE DISTANCE = X TEST DISTANCE.

Y RASTER VALUE = 0.05

Set the Y raster or step increment between passes to 0.05 inches. Not used for method 2.

FILE NAME = X.TAP

Save the probe coordinates to the file X.TAP. A file header of G01 F10 is attached to set the file in linear interpolation mode with a feed rate of 10 Inch/Min.

**OPTION 3** Exit the probe program and return to the main menu.

### **PROBE SETUP and OPERATION:**

1. The digitizing probe is installed in the spindle using a 0.25 inch collet as shown in figure D3. **IMPORTANT NOTE** - Be sure to unplug the spindle motor so that it is not accidentally turned on when the probe is in the spindle.
2. Once the probe is installed select option E to enter the Probe Program. Select option 2 to set the probe parameters. Enter the parameters as prompted. Entering a blank retains the current value.
3. Exit the Probe program with option 3 and go into Manual Jog mode. If digitizing method 1 was selected jog the probe tip to the lower left corner of the part area you wish to digitize. If digitizing method 2 was selected jog the probe tip to the center of the part. Be sure the probe tip is higher than the highest surface of the part to be digitized. Typically the probe is set about 0.1 to 0.25 inches above this point. Once the tip is positioned exit Manual Jog mode and Zero all the axes. The part is now ready to be digitized.
4. To view the 3D digitizing process on the screen be sure the PLOT VIEW option **V** is set to 3D PLOT VIEW and the scan is visible on the screen. For instance if Method 1 or XY rastering is selected with the X LIMIT = 2 and Y LIMIT = 1 then you should set the X OFFSET = 1 and Y OFFSET = 0.5 to center it. This is selected with option **O** from the main menu. If Method 2 or XY Spiraling is selected just set the X and Y offsets to 0 in general.

4. Enter the Probe Program and select 1 to start the digitizing process. You are prompted to start the digitizing at this point. Hit ENTER to begin or the SPACEBAR to escape. NOTE: Once option 1 is selected the filename is rewritten.
5. The digitizing process can be HALTED at any time by pressing the SPACEBAR. If pressed the user is prompted to continue the process by pressing C or stopping it by pressing ENTER. If ENTER is pressed the digitized points up to that point will remain stored in the chosen filename.
6. Once the digitizing process is complete the user is prompted to press ENTER to return the probe back to the home or (0,0,0) point or press the SPACEBAR to stop at the end point. If ENTER is pressed the following code is appended to the digitized G-Code file.

```
M0  
G0 Z0  
M0  
X0 Y0  
M30
```

#### **RUNNING THE DIGITIZED PROGRAM:**

Once a digitized file has been created it can now be used to machine a duplicate part. The file is treated like any other normal G-Code file after it is created. The header of G01 F10 is attached to the top of the file to set it in linear interpolation mode with a default feed rate of 10 Inch/Min. This can be edited in a standard text editor if needed.

Note: If you wish to machine your part in hard woods or metals you most likely have to take multiple depth cuts into the material using option 6 from the main program (See SET DEPTH INC section on page 9). This is because deep profiles are digitized directly in one pass whereby the machining process must take multiple cuts to reach these depths.

Fig. D - Methods of digitizing a surface.



# LEADSCREW COMPENSATION

The leadscrew on the MicroMill system, like any other machine system, may have slight variations from the true pitch of the screw. The standard screw pitch is 20 turns/inch for all three axes on the machine. For the 1.8 deg/step drive motors and half stepping mode employed by the driver electronics and software this translates to 8000 steps/inch or a step size of 0.000125 inch/step. This is derived as follows:

$$\text{Step Size}[\text{Step}/\text{Inch}] = \frac{20 \text{ Rev}}{\text{Inch}} \cdot \frac{360 \text{ Deg}}{\text{Rev}} \cdot \frac{2 \text{ Step}}{1.8 \text{ Deg}} = 8000 \frac{\text{Steps}}{\text{Inch}}$$

or taking the inverse,

$$\text{Step Size}[\text{Inch}/\text{Step}] = \frac{1 \text{ Inch}}{8000 \text{ Steps}} = 0.000125 \frac{\text{Inch}}{\text{Step}}$$

If you find that the machine is not moving the proper distance along the lead screw there may be a slight variation of the screw pitch. In this case you can reprogram the step size to compensate for the leadscrew pitch error. For example if you program the machine to move 1.000 inches and measure the actual movement to be 1.001 inches there is a leadscrew error of + 0.001 inch/inch. In this case the true pitch is slightly less than 20 turns/inch. The step size or step/inch setting would then need to be decreased to compensate for this as follows:

$$\text{Step Size}[\text{Step}/\text{Inch}] = \frac{\text{Programmed Distance}}{\text{True Distance}} = \frac{1.000}{1.001} \cdot \frac{8000 \text{ Steps}}{\text{Inch}} = 7992 \frac{\text{Steps}}{\text{Inch}}$$

This value of **7992** can then be entered as the new step size using the **CHANGE UNITS** option from the main menu. Type **U** and choose option **3** for Custom units. Each axis can then be reprogrammed with the true step size. Hitting Enter with no value retains the previous value.

**Note:** This assumes the leadscrew pitch error to change linearly over the length of the screw. This is typically the type of leadscrew error that occurs, but the error may be a more complex variation in certain cases. In this case this leadscrew compensation technique will not completely eliminate the inherent pitch error.

# SPEEDS AND FEEDS FOR DRILLING

### DRILLING SPEEDS (HIGH SPEED STEEL DRILLS):

<b><u>Material</u></b>	<b><u>Average Drill Speed (sfm)</u></b>
Magnesium	300
Aluminum	250
Brass and Bronze	200
Copper	70
Cast Iron (soft)	120
Cast Iron (hard)	80
Mild Steel	110
Cast Steel	50
Alloy Steels (hard)	60
Tool Steel	60
Stainless Steel	30
Titanium	30
High Manganese Steel	15

Note: For carbide drills, double the average speeds.

### DRILLING FEEDS:

<b><u>Drill Diameter (inch)</u></b>	<b><u>Drill Feed (ipr)</u></b>
< 1/8	0.001 - 0.002
1/8 - 1/4	0.002 - 0.004
1/4 - 1/2	0.004 - 0.007
1/2 - 1	0.007 - 0.015
> 1	0.015 - 0.025

Note: Use a slightly lower feed when drilling hard materials such as carbon steel, tool steel, or stainless steel. Use a slightly higher feed when drilling softer materials such as magnesium, aluminum, or soft cast iron.

## SPEEDS AND FEEDS FOR MILLING

### MILLING SPEEDS (HIGH SPEED STEEL END MILLS):

<b><u>Material</u></b>	<b><u>Average Tool Speed (sfm)</u></b>
Magnesium	300
Aluminum	250
Brass and Bronze	150
Copper	100
Cast Iron (soft)	80
Cast Iron (hard)	50
Mild Steel	90
Cast Steel	80
Alloy Steels (hard)	40
Tool Steel	50
Stainless Steel	60
Titanium	50
High Manganese Steel	30

Note: For carbide cutting tools, double the average speeds.

### MILLING FEEDS:

<b><u>Material</u></b>	<b><u>TOOL FEED (inch/tooth)</u></b>		
	<b><u>Face Mills</u></b>	<b><u>Side Mills</u></b>	<b><u>End Mills</u></b>
Magnesium	0.005 - 0.020	0.004 - 0.010	0.005 - 0.010
Aluminum	0.005 - 0.020	0.004 - 0.010	0.005 - 0.010

Brass and Bronze	0.004 - 0.020	0.004 - 0.010	0.005 - 0.010
Copper	0.004 - 0.010	0.004 - 0.007	0.004 - 0.008
Cast Iron (soft)	0.004 - 0.016	0.004 - 0.009	0.004 - 0.008
Cast Iron (hard)	0.004 - 0.010	0.002 - 0.006	0.002 - 0.006
Mild Steel	0.004 - 0.010	0.002 - 0.007	0.002 - 0.010
Alloy Steel (hard)	0.004 - 0.010	0.002 - 0.007	0.002 - 0.006
Tool Steel	0.004 - 0.008	0.002 - 0.006	0.002 - 0.006
Stainless Steel	0.004 - 0.008	0.002 - 0.006	0.002 - 0.006
Titanium	0.004 - 0.008	0.002 - 0.006	0.002 - 0.006
High Manganese Steel	0.004 - 0.008	0.002 - 0.006	0.002 - 0.006

### **IMPORTANT MACHINING FORMULAS**

Cutting Speed (sfm)  $V_c = 0.262 \times D_m \times \text{rpm}$

Feed Rate ( inch/min.)  $F_m = F_t \times n \times \text{rpm}$

Feed per tooth (inch)  $F_t = F_m / ( n \times \text{rpm} )$

$D_m$  = Milling cutter diameter (inch)

rpm = Revolutions per minute

sfm = Surface feet per minute

n = Number of teeth in cutter

$0.262 = \pi / 12$