

# **INDEX**

- 1. MC Training Manual**
- 2. Additional Simple Cycles**
- 3. USB Interface**
- 4. Installation**
- 5. Electrical Drawings**

**FAGOR**



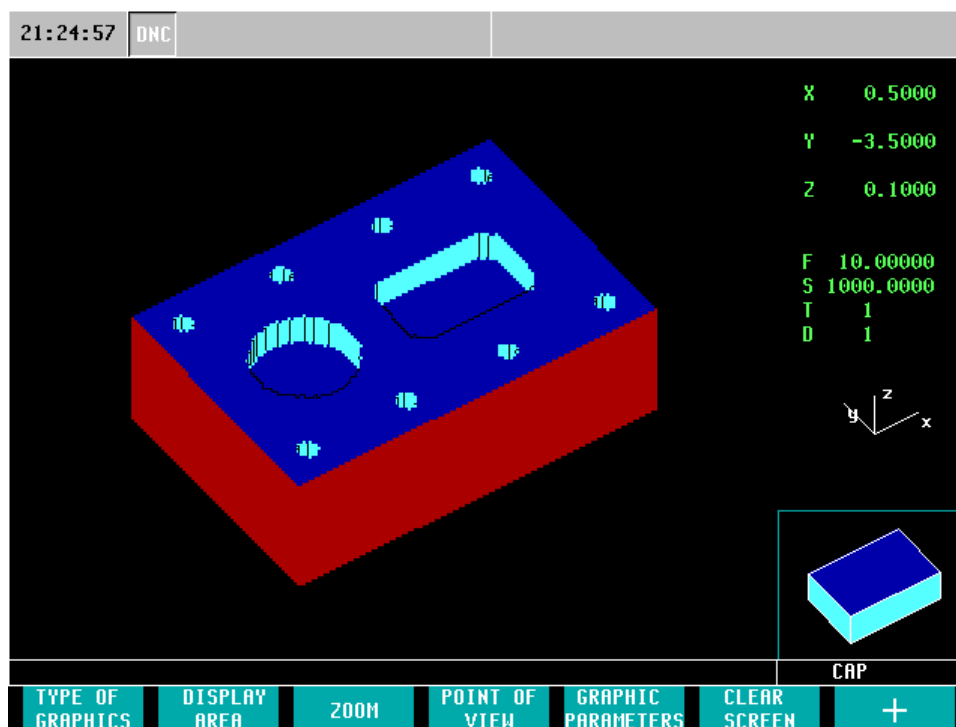
**1 800 4A FAGOR**

**\* This information package also includes 8055 CNC Training DVD.**

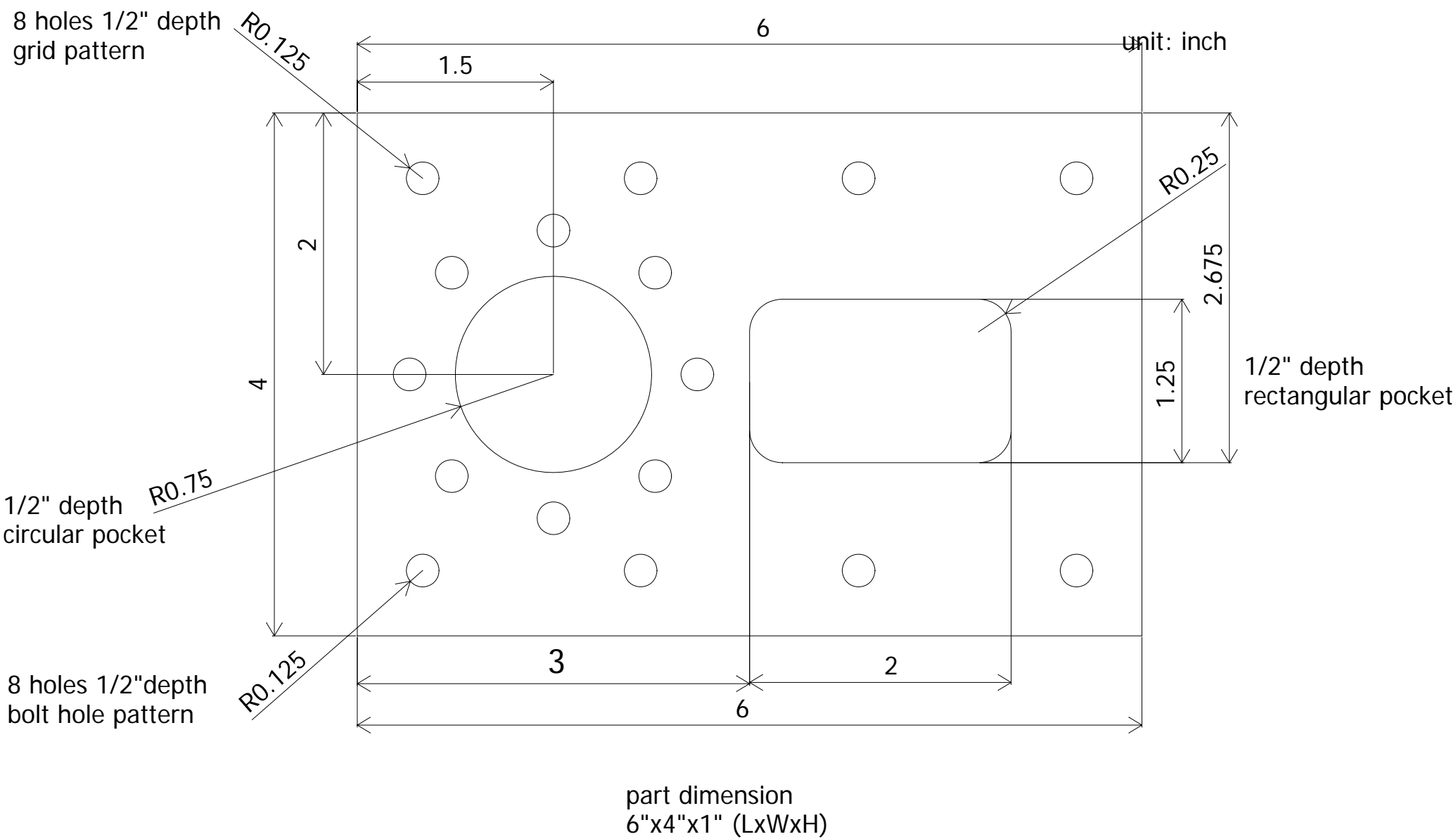
**\*\* For more information please refer to Operating/Programming Manuals.**



# FAGOR AUTOMATION MC TRAINING MANUAL



# ACER MC TRAINING MANUAL



# INDEX

<b>Introduction.....</b>	<b>39</b>
<b>Reference Search.....</b>	<b>40</b>
<b>Tool Calibration.....</b>	<b>41</b>
<b>Presetting Part Zero.....</b>	<b>43</b>
<b>Surface Milling.....</b>	<b>44</b>
<b>Simulation.....</b>	<b>46</b>
<b>Storing Part Program.....</b>	<b>47</b>
<b>Circular Pocket.....</b>	<b>49</b>
<b>Rectangular Pocket.....</b>	<b>52</b>
<b>Drilling + Rectangular Position.....</b>	<b>55</b>
<b>Drilling + Arc Position.....</b>	<b>58</b>
<b>Zero Offsets.....</b>	<b>62</b>
<b>ISO Mode.....</b>	<b>64</b>
<b>Inserting Code In-Between Canned Cycles.....</b>	<b>65</b>
<b>Tool Inspection.....</b>	<b>68</b>
<b>Quick Reference Guide.....</b>	<b>71</b>

# INTRODUCTION


**This manual has been generated to be a guideline for your assistance for the conversational programming of the Fagor control. In this manual we will be generating a part-program using the canned cycles that will be the most beneficial for your programming needs. In this manual we will be exercising the following operations:**

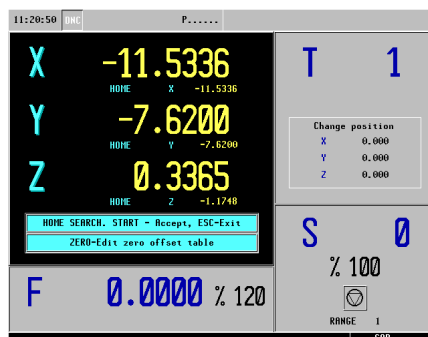
- **Homing of the Axes**
- **Tool Calibration**
- **Presetting your Zero**
- **Surface Milling**
- **Pockets**
- **Drilling**
- **Simulation/Graphics**
- **File Maintenance**

**After the completion of these operations, you will begin to understand all the steps that are necessary in programming the Fagor CNC Control. Once you grasp each of these concepts you will be able generate any part in which you desire. Using this manual as your guideline will also help you generate part-programs in a timely manner. You will soon realize once you learn the programming of one canned cycle, you will understand the programming concept of each and every one of the other canned cycles. Whether it is your first time using a CNC or if you are a CNC programming expert, the conversational programming of the Fagor CNC Control will benefit all users.**


# REFERENCE SEARCH

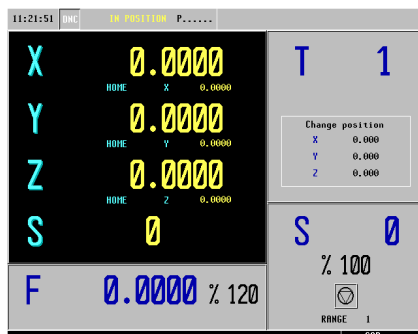
On power-up, it is always recommended to home your machine before performing any specific task. This is more of a safety precaution. When referencing your machine the CNC will know exactly where it is located. To reference your machine perform the following operation:

1. From the Jog Screen select the Zero Icon 
  - The following screen will appear



*\* Note: Pressing the ESC Key instead of Cycle Start will cancel the operation*


2. To execute the Home Search press cycle start 
  - At this time the CNC will execute the subroutine associated with the home search
  - The Z axis will be referenced first followed by the X and Y

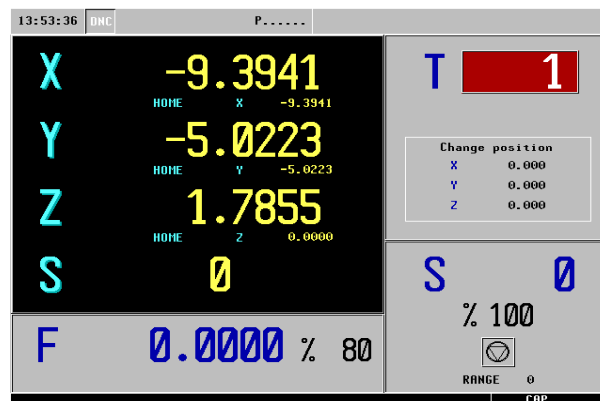


# TOOL CALIBRATION

For this specific program we will be using three tools. Before we begin any type of programming we will calibrate all three of our tools. For this particular program we will preset our Z starting point at .01 The reason for this is our first programming cycle will be Surface Milling. In our Surface Milling cycle we will remove .01 from the surface allowing us to program our remaining cycles starting from Z zero.

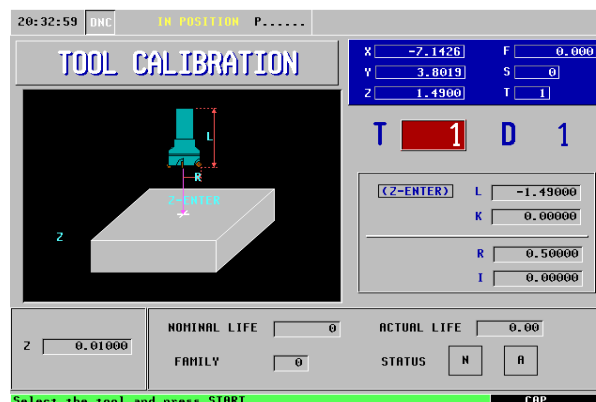
1. From the Jog Screen enter the following commands:

- T (the box will highlight in red)
- 1 (Tool 1)
- Cycle Start 




2. The axes will now move to its Tool Change Position.

3. Select the F1 Icon. 



4. Jog your axes accordingly until the tip of your tool is touching off on the top of your part.
5. Enter the value of .01000 for your Z and then press Enter. This box is located in the lower-left corner of your screen. This will preset your Z starting point at .01000
6. Define your tool:
  - L- By selecting Z and then hitting the Enter key, this will define the length of your tool from its home position.  
*Make sure the Z-ENTER box is highlighted in red when performing this operation.*
  - K- This is the length wear of your tool. At this time it is unnecessary to put any value in this field.
  - R- This is the Tool Radius. For this example we will be using a tool with a ½' radius.
  - I- This is the Radius Wear. At this time it is unnecessary to put any value in this field.

### TOOLS 2 AND 3

- Select T (Tool)
- Enter 2 or 3 (Tool Number)
- Press Cycle Start 
- Repeat steps 4-6

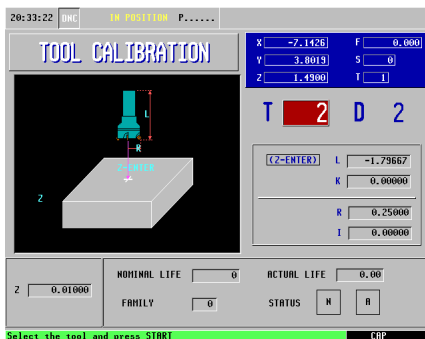


Figure 1

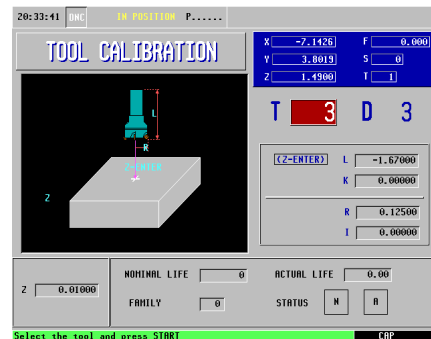


Figure 2



# PRESETTING PART ZERO

## X AXIS

1. From the Jog Mode, jog your X axis until you are to the position you wish to call zero.  
Enter the following keys:

- X (select your axis)
- The number 0 (your preset value)
- Enter (confirming your position)
- Enter (executing your command)

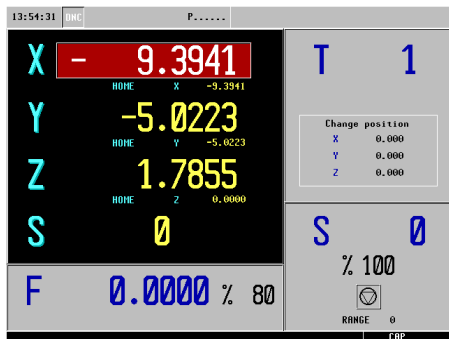


Figure 1

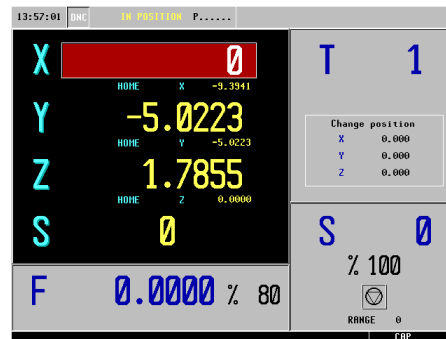


Figure 2

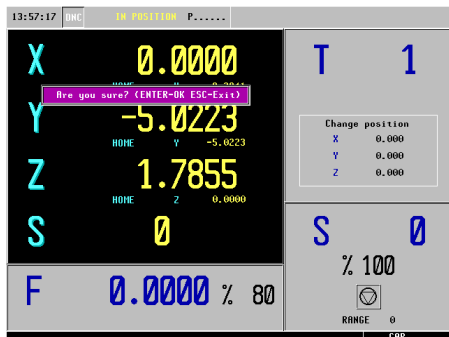


Figure 3

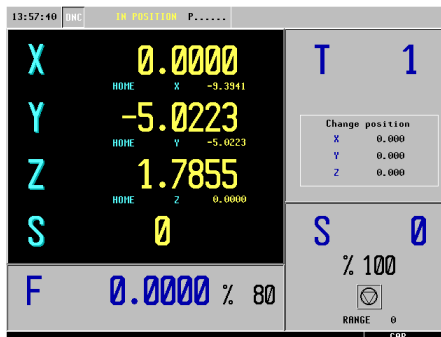
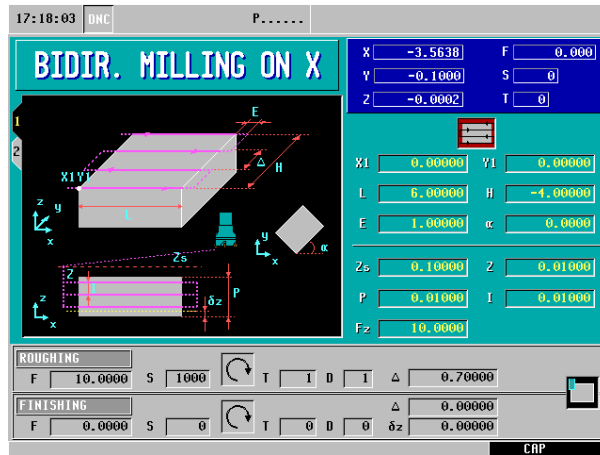



Figure 4

- Repeat these steps for your Y and Z axis. Remember to preset your Z axis to .01 as discussed earlier.

# SURFACE MILLING







1. Select the F4 Icon. 
2. For this operation we will be using the Bi-Directional Milling along X. To toggle between different Surface Milling cycles select the “half-key” over the icon while the box is highlights in red.



3. Define your part:
  - (X1, Y1)- This is your starting point of your X and Y coordinates. In this example we will be using X0 and Y0.
  - L- This is the length of your part along the X axis. In this example we will be using the value of 6.
  - H- This is the height of your part along the Y axis. In this example we will be using the value of -4.
  - E- This is the amount the tool will overshoot the programmed part after it has reached its destination. In this example we will be using the value of 1.
  - " $\alpha$ " - The value in this field must be entered in as degrees. A value is entered into this field when we are machining a part on an inclination angle. In this example we will be using a value of 0.
4. Define your machining conditions:
  - Zs- This is your Z safety distance. This is the amount your tool will be above your starting point before beginning its operation. In this example we will be using a value of .1
  - Z- This is the starting point on the Z axis. In this example we will be using a value of .01


- P- This is the total depth we will be removing off the surface of the material. In this example we will be using the value of .01
- I- This is the amount we will be removing off the surface of the material per pass. In this example we will be using the value of .01 meaning it will only take one pass to get to its final depth.
- Fz- This is the penetration feedrate. This value is entered in inches per minute. In this example we will be using the value of 10.

5. Define your Roughing Pass:

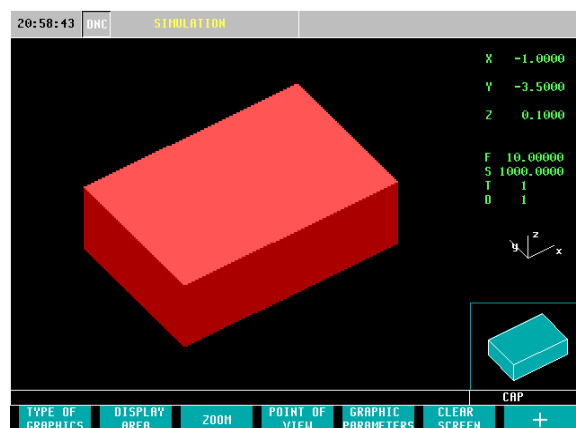
- F- This is your roughing feedrate along the axis. In this example we are using a value of 10 inches per minute.
- S- This is your spindle speed. In this example we are using a value of 1000 rpm.
- Select your spindle direction.  To toggle between directions push the  “half-key”. In this example we will be using a Clockwise Rotation.
- T- Enter is your tool number. In this example we are using T1.
- D- Enter in your Tool Offset Number. In this example we are using D1.
-  This is your milling step-over. To ensure nice clean cuts enter in a value less than the diameter of your tool.
- Finally, select the corner you would like to begin milling. To toggle between options push the half-key. In this example we will be beginning in the upper-left corner. 

# SIMULATION

Before the execution of any part program, running your program in simulation is strongly recommended. The following steps will guide you on how to set-up your graphics.

1. Select the Graphics Icon. 
2. Select F1- Type of Graphics
3. Select F7- Solid Graphics
4. Select F2- Display Area ----->

*The following screen will appear on your CNC.*



X Min: 0

X Max: 6

Y Min: 0


Y Max: -4

Z Min: -1.5

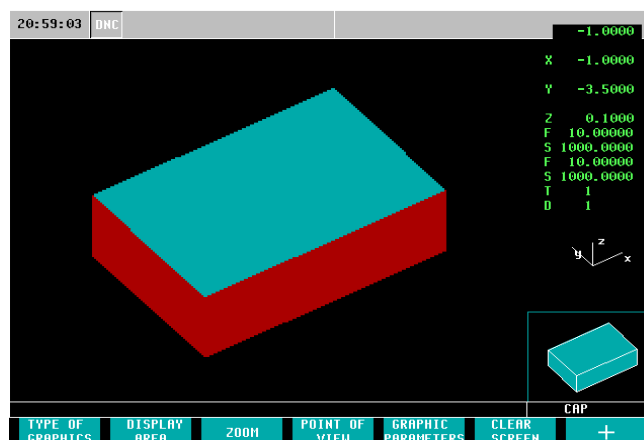
Z Max: 0

*-Press the down arrow after putting numbers in boxes.*

*-Press Enter after completed.*

5. Now to run your program in simulation press the Cycle Start Button. 

6. After simulating the cycle the following screen should appear:



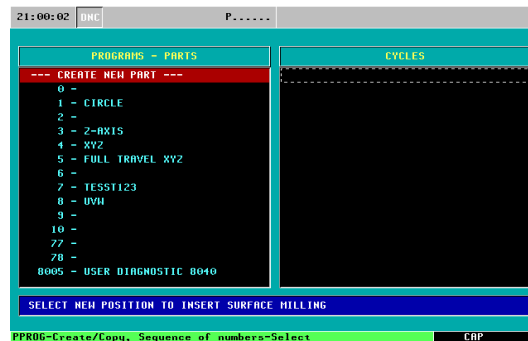
*Note: If you notice no change in your graphics while running in simulation, please be sure to check your coordinates located in your "DISPLAY AREA" (F2).*

7. You are now ready to save this cycle to your Part-Program.

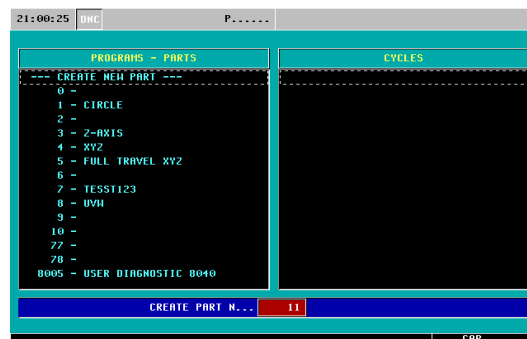
# Storing Part Programs



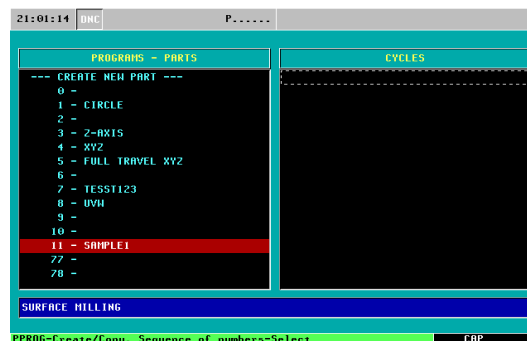
1. Select The Part-Program Key.
  - *The following screen will appear*



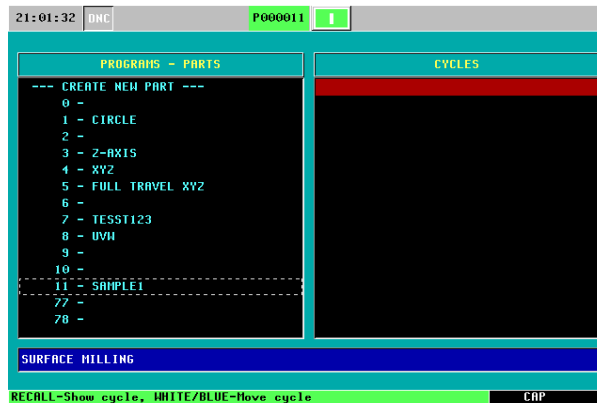
2. Select the Part-Program Key once more and enter a Part-Program Number. Hit enter when you have entered in your number. In this example we will use 11.
  - *The CNC will call up the first available number*



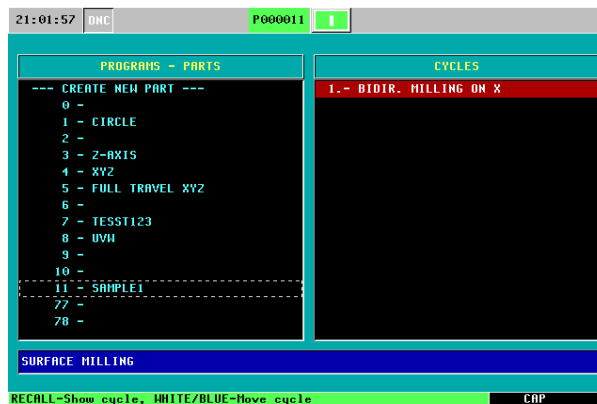
3. Enter a comment for your part-program and hit Enter. In this example we are using SAMPLE1.



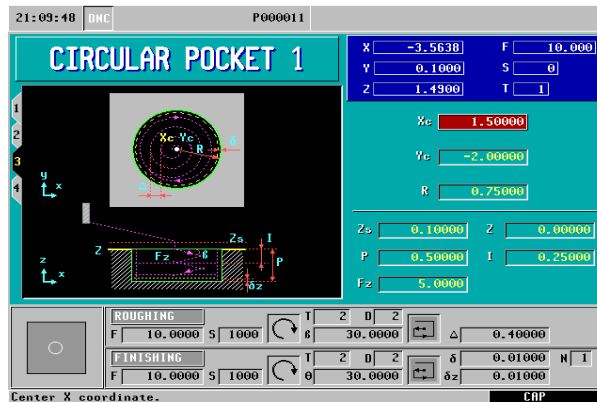
4. Select the Right Arrow Cursor.  Now the right-hand column (cycles) will be highlighted in red.








5. Once this section is highlighted in red you may now hit enter and your cycle is saved






# CIRCULAR POCKET



1. Select the F7 Icon. 
2. Press the Level Cycle key.  Press this key until you are in Level Cycle 3 (Circular Pocket 1).
3. Define your pocket:
  - Xc- This is the center point of your pocket along the X Axis. In this example we will be using a value of 1.5
  - Yc- This is the center point of your pocket along the Y Axis. In this is example we will be using a value of -2
  - R- This is the Radius of the pocket. In this example we will be using a value .75
4. Define your machining conditions:
  - Zs- This is your Z safety distance. This is the amount your tool will be above your starting point before beginning its operation. In this example we will be using a value of .1
  - Z- This is the starting point on the Z axis. In this example we will be using a value of 0
  - P- This is the total depth of the pocket. In this example we will be using a value of .5
  - I- This is the amount we will be removing depth wise per pass. In this example we will be removing a 0.25" of material per pass meaning it will take two passes until we get to our final depth.
  - Fz- This is the penetration feedrate. This value is entered in inches per minute. In this example we will be using a value of 5
5. Define your Roughing Pass:
  - F- This is your roughing feedrate along the axis. In this example we are using a value of 10 inches per minute.

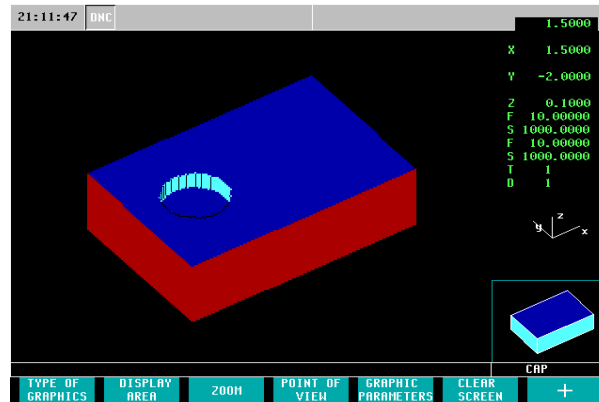
- S- This is your spindle speed. In this example we are using a value of 1000 rpm.
- Select your spindle direction.  To toggle between directions push the  “half-key”. In this example we will be using a Clockwise Rotation.
- T- Enter is your tool number. In this example we are using T2.
- D- Enter in your Tool Offset Number. In this example we are using D2.
- ( $\beta$ ) This is your Sideways Penetration Angle. In this example we will penetrate into the center of our pocket on a 30 Degree Angle.
-  This is the type of milling (Climb/Conventional Milling).
- ( $\Delta$ ) This is your step-over. In this example we will be using a value of .4

6. Define your Finishing Pass:

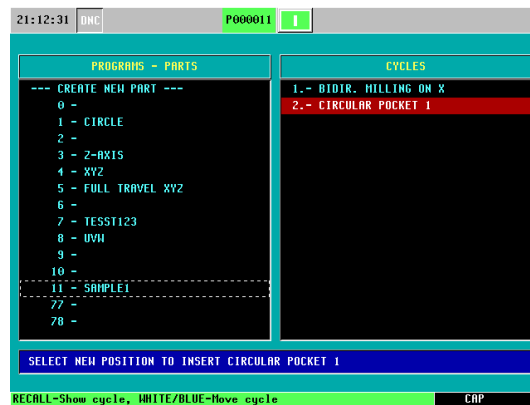
- F- This is your finishing feedrate. In this example we will be using a value of 10 inches per minute.
- S- This is your spindle speed. In this example we are using a value of 1000 rpm.
- Select your spindle direction.  To toggle between directions push the  “half-key”. In this example we will be using a Clockwise Rotation.
- T- Enter is your tool number. In this example we are using T2.
- D- Enter in your Tool Offset Number. In this example we are using D2.
- ( $\beta$ ) This is your Sideways Penetration Angle. In this example we will penetrate into the center of our pocket on a 30 Degree Angle.
-  This is the type of milling (Climb/Conventional Milling).
- $\delta$  This is the Finishing Pass. In this example we will be leaving 0.01 for the finishing pass.
- $\delta Z$  This is the Finishing Pass in Z. In this example we will be leaving 0.01 for the finishing in Z.
- ( $N$ ) This is the Number of Finishing Passes in Z. In this example we will be using only 1 finishing pass.



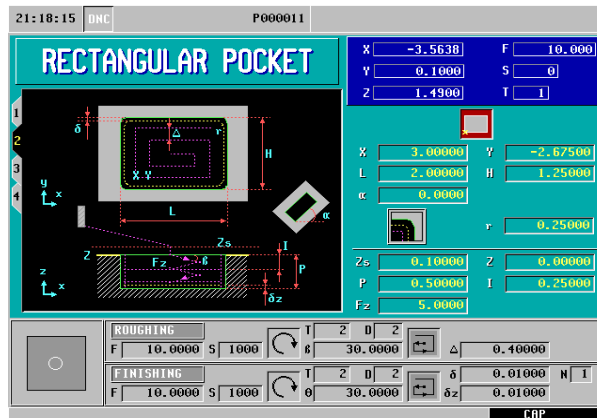
7. Simulate the program:



8. Save your part-program.



# RECTANGULAR POCKET



1. Select the F7 Icon.









2. Press the Level Cycle key. Press this key until you are in Level Cycle 2 (Rectangular Pocket).

3. Define your pocket:

- X- This is the starting point of your pocket along the X Axis. In this example we will be using a value of 3
- Y- This is the starting point of your pocket along the Y Axis. In this example we will be using a value of -2.675
- L- This is the total length of the pocket along the X Axis. In this example we will be using a value of 2
- H- This is the total height of the pocket along the Y Axis. In this example we will be using a value of 1.25
- " $\alpha$ " - The value in this field must be entered in degrees. A value is entered into this field when we are machining a part on an inclination angle. In this example we will be using a value of 0.
- r – In this cycle we are able to define the radius of the corners. You have the options of square, rounded, or chamfer. To toggle between the options press the

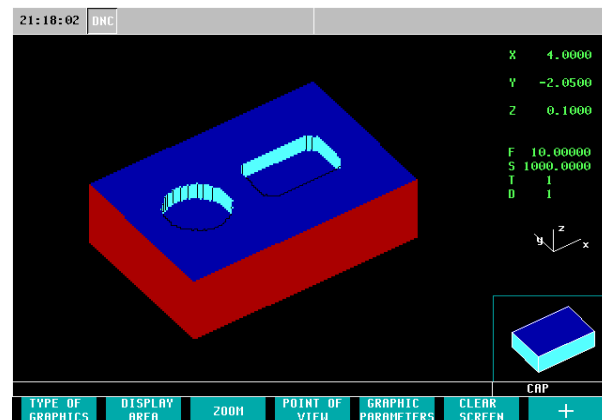


"half-key". In this example we will be using rounded corners and entering in a value of a .25 radius.

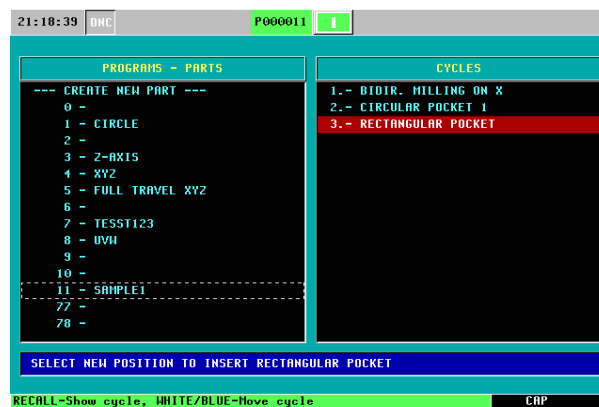
4. Define your machining conditions:
  - Zs- This is your Z safety distance. This is the amount your tool will be above your starting point before beginning its operation. In this example we will be using a value of 0.100"
  - Z- This is the starting point on the Z axis. In this example we will be using a value of 0.
  - P- This is the total depth of the pocket. In this example we will be using the value of 0.500".
  - I- This is the amount we will be removing depth wise per pass. In this example we will be removing a 0.250" of material per pass meaning it will take two passes until we get to our final depth.
  - Fz- This is the penetration feedrate. This value is entered in inches per minute. In this example we will be using a value of 5.
5. Define your Roughing Pass:
  - F- This is your roughing feedrate along the axis. In this example we are using a value of 10 inches per minute.
  - S- This is your spindle speed. In this example we are using a value of 1000 rpm.
  - Select your spindle direction.  To toggle between directions push the  "half-key". In this example we will be using a Clockwise Rotation.
  - T- Enter is your tool number. In this example we are using T2.
  - D- Enter in your Tool Offset Number. In this example we are using D2.
  - ( $\beta$ ) This is your Sideways Penetration Angle. In this example we will penetrate into the center of our pocket on a 30 Degree Angle.
  -  This is the type of milling (Climb/Conventional Milling).
  - ( $\Delta$ ) This is your step-over. In this example we will be using a value of .4
6. Define your Finishing Pass:
  - F- This is your finishing feedrate. In this example we will be using a value of 10 inches per minute.
  - S- This is your spindle speed. In this example we are using a value of 1000 rpm.
  - Select your spindle direction.  To toggle between directions push the  "half-key". In this example we will be using a Clockwise Rotation.
  - T- Enter is your tool number. In this example we are using T2.
  - D- Enter in your Tool Offset Number. In this example we are using D2.
  - ( $\beta$ ) This is your Sideways Penetration Angle. In this example we will penetrate into the center of our pocket on a 30 Degree Angle.
  -  This is the type of milling (Climb/Conventional Milling).
  - $\delta$  This is the Finishing Pass. In this example we will be leaving 0.010" for the finishing pass.
  - $\delta Z$  This is the Finishing Pass in Z. In this example we will be leaving .01 for the finishing in Z.

- (N) This is the Number of Finishing Passes in Z. In this example we will be using only 1 finishing pass.

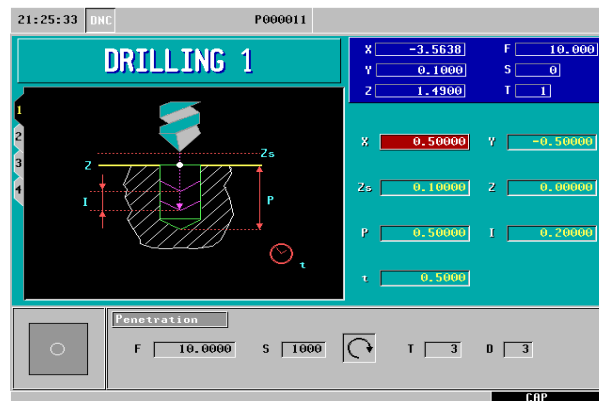
7. Simulate the program:




8. Save your Cycle to your part-program:




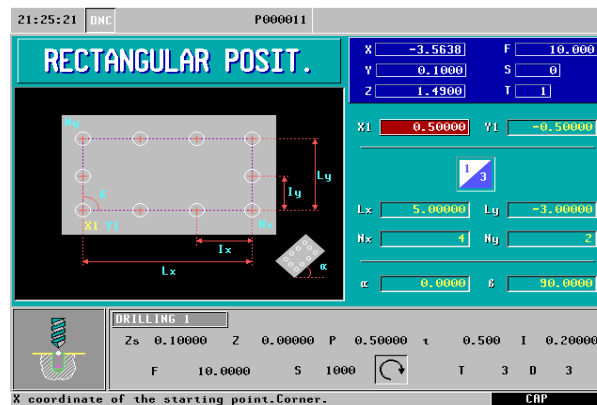
# DRILLING + RECTANGULAR POSIT.



1. Select the Drilling Icon
2. Be sure to have Drilling 1 (Level Cycle 1) selected.
3. Define the position of your first hole:
  - X- This is the position of your first hole along the X Axis. In this example we will be using a value of 0.500”.
  - Y- This is the position of your first hole along the Y Axis. In this example we will be using a value of -0.500”.
4. Define your machining conditions:
  - Zs- This is your Z safety distance. This is the amount your tool will be above your starting point before beginning its operation. In this example we will be using a value of 0.100”.
  - Z- This is the starting point on the Z axis. In this example we will be using a value of 0.
  - P- This is the total depth of the hole. In this example we will be using a value of 0.500”.
  - I- This is the drilling peck. In this example we will be drilling down 0.2 per pass.
  - t- This is the dwell time (seconds). In this example there will be a .5 second dwell time per pass.
5. Define your Penetration:
  - F- This is your feedrate. In this example we will be using a value of 10 inches per minute.
  - S- This is your spindle speed. In this example we are using a value of 1000 rpm.

- Select your spindle direction.  To toggle between directions push the “half-key”. In this example we will be using a Clockwise Rotation.
- T- Enter is your tool number. In this example we are using T3.
- D- Enter in your Tool Offset Number. In this example we are using D3.

6. Select the Rectangular Posit. Icon 

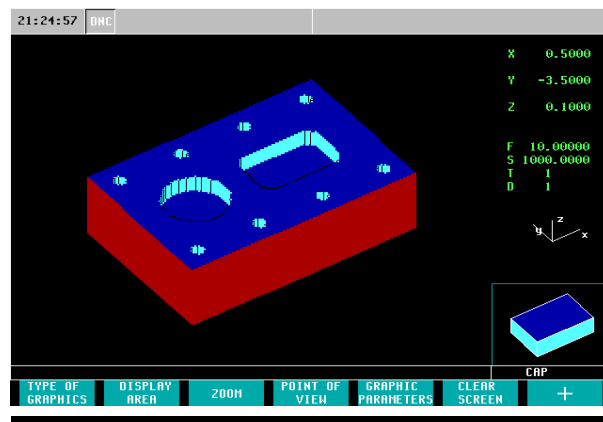


*\*Note- Your starting coordinates (X1, Y1) and machining conditions are automatically entered because they were predefined in our drilling cycle.*

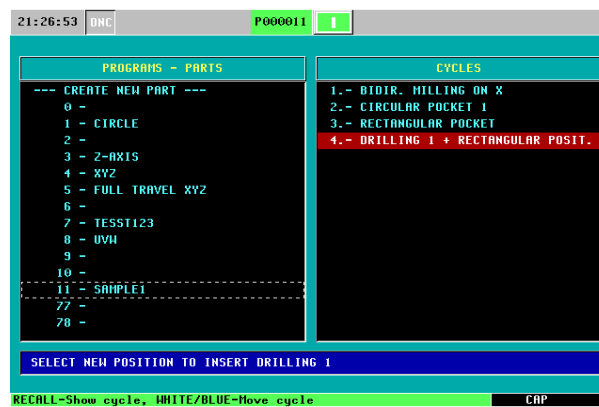
7. Define the length and number of hole along each axis:

- Lx- This is the length from your first hole on the X Axis to the last hole on your X Axis. In this example we will be using a value of 5.
- Ly- This is the length from your first hole on the Y Axis to the last hole on your Y Axis. In this example we will be using a value of -3.
- Nx- This is your total number of holes along the X Axis. In this example we will be using a value of 4.
- Ny- This is your total number of holes along the Y Axis. In this example we will be using a value of 2.
- "α" - The value in this field must be entered in degrees. A value is entered into this field when we are machining a part on an inclination angle. In this example we will be using a value of 0.
- (β) This is the angle between the sides. In this example we will be using a value of 90 degrees.

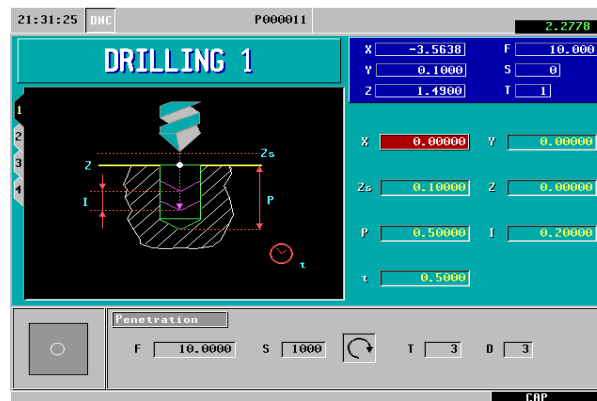
8. Simulate the program:




9. Save your Cycle to your part-program:




# DRILLING+ARC POSITIONING

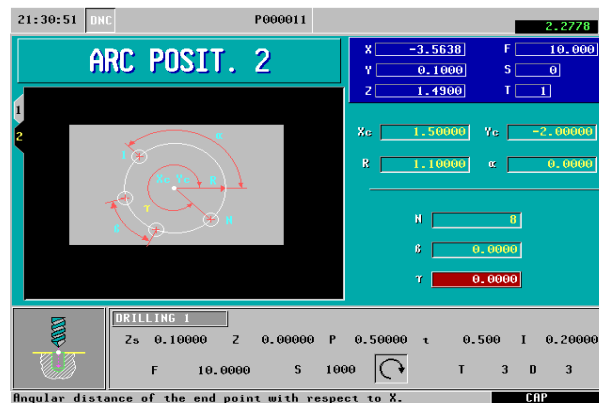


1. Select the Drilling Icon 
2. Be sure to have Drilling 1 (Level Cycle 1) selected.
3. Define the position of your first hole:
  - X- This is the position of your first hole along the X Axis. In this example we will be using a value of 0.
  - Y- This is the position of your first hole along the Y Axis. In this example we will be using a value of 0.
4. Define your machining conditions:
  - Zs- This is your Z safety distance. This is the amount your tool will be above your starting point before beginning its operation. In this example we will be using the value of 0.100”.
  - Z- This is the starting point on the Z axis. In this example we will be using a value of 0
  - P- This is the total depth of the hole. In this example we will be using a value of 0.500”
  - I- This is the drilling peck. In this example we will be drilling down .2 per pass.
  - t- This is the dwell time (seconds). In this example there will be a .5 second dwell time per pass.
5. Define your Penetration:
  - F- This is your feedrate. In this example we will be using a value of 10 inches per minute.
  - S- This is your spindle speed. In this example we are using a value of 1000 rpm.



- Select your spindle direction.  To toggle between directions push the “half-key”. In this example we will be using a Clockwise Rotation.
- T- Enter is your tool number. In this example we are using T3.
- D- Enter in your Tool Offset Number. In this example we are using D3.

6. Select the Arc Positioning Icon 

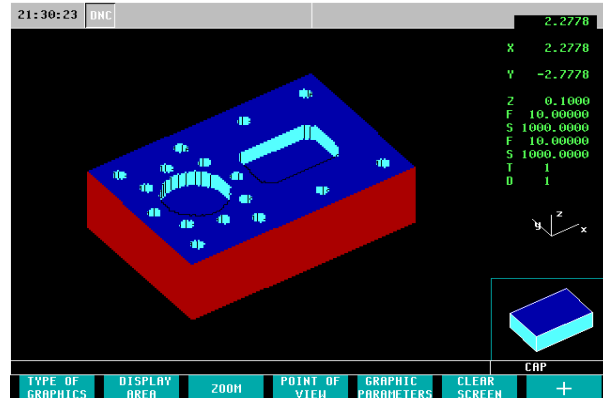


7. Go into Level Cycle 2 (ARC POSIT. 2) 

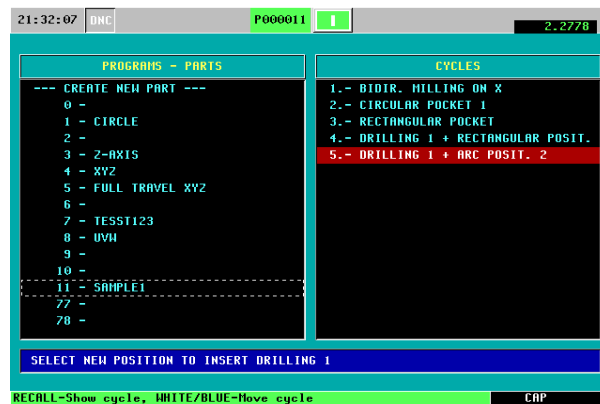
8. Define the Center Coordinates and Machining Conditions:

- Xc- This is the center point of your bolt-hole position on the X Axis. In this example we will be using a value of 1.5
- Yc- This is the center point of your bolt-hole position on the Y Axis. In this example we will be using the value of -2
- r- this is the radius of your bolt-hole pattern. In this example we will be using a value of 1.1
- " $\alpha$ " - The value in this field must be entered in as degrees. A value is entered into this field when we are machining a part on an inclination angle. In this example we will be using a value of 0.
- N- This is the total number of holes in the arc. In this example we will be using a value of 8.
- ( $\beta$ ) This is the angular distance between points. In this example we will be using a value of 0
- ( $\tau$ ) This is the angle of the end point. In this example we will be using a value of 0

9. Simulate the program:



10. Save your Cycle to your part-program



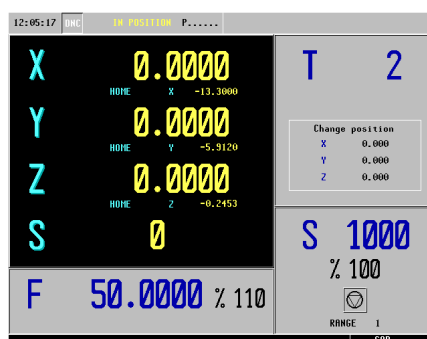
You have now created and simulated your part-program and are ready for execution.

## **Miscellaneous Features**

# ZERO OFFSETS

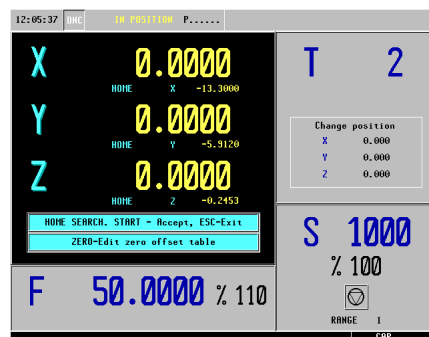
G54 through G57 are our absolute zero offsets. This feature allows us to store a part zero position into a table. With this option you are able to recall your part zero position without having to touch off on your part again. This feature is useful when having parts that are commonly machined. To setup your Zero Offsets perform the following task:


1. From the Jog Screen setup your part zero position



*\*Note: Refer to page 4 of the manual on how to preset your part zero*

2. Select the Zero Icon 



3. Select the Zero Icon  for a second time to enter the Zero Offset Table

12:06:10 [UNC] 10 POSITION P.....

**ZERO OFFSET TABLE**

X [ 0.0000 ] F [ 55.000 ]  
Y [ 0.0000 ] S [ 0 ]  
Z [ 0.0000 ] T [ 2 ]

	X	Y	Z
PLC	0.00000	0.00000	0.00000
ROI	0.00000	0.00000	0.00000
G54	0.00000	0.00000	0.00000
G55	0.00000	0.00000	0.00000
G56	0.00000	0.00000	0.00000
G57	0.00000	0.00000	0.00000
G58	0.00000	0.00000	0.00000
G59	0.00000	0.00000	0.00000

CRP

*\* Note: All of the coordinate values for G54 are currently at zero*

4. Select the Recall Key 

12:06:56 [UNC] 10 POSITION P.....

**ZERO OFFSET TABLE**

X [ 0.0000 ] F [ 55.000 ]  
Y [ 0.0000 ] S [ 0 ]  
Z [ 0.0000 ] T [ 2 ]

	X	Y	Z
PLC	0.00000	0.00000	0.00000
ROI	0.00000	0.00000	0.00000
G54	-13.29933	-5.91198	0.29933
G55	0.00000	0.00000	0.00000
G56	0.00000	0.00000	0.00000
G57	0.00000	0.00000	0.00000
G58	0.00000	0.00000	0.00000
G59	0.00000	0.00000	0.00000

CRP

*\* Note: The coordinates are now the exact value of their position away from home*

5. Press Enter

12:07:12 [UNC] 10 POSITION P.....

**ZERO OFFSET TABLE**

X [ 0.0000 ] F [ 55.000 ]  
Y [ 0.0000 ] S [ 0 ]  
Z [ 0.0000 ] T [ 2 ]

	X	Y	Z
PLC	0.00000	0.00000	0.00000
ROI	0.00000	0.00000	0.00000
G54	-13.29933	-5.91198	0.29933
G55	0.00000	0.00000	0.00000
G56	0.00000	0.00000	0.00000
G57	0.00000	0.00000	0.00000
G58	0.00000	0.00000	0.00000
G59	0.00000	0.00000	0.00000

CRP

*\*Note: Your values will not be stored if you do not hit enter*

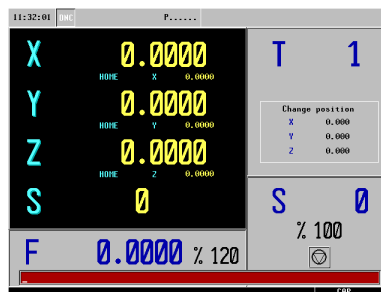
# ISO MODE

The ISO key gives you access to the MDI Mode. What this feature allows you to do is perform one or more functions at a time. It allows you to move more than one axes at a time, start the spindle, set the feed, insert G-Codes, etc. To operate in ISO Mode, execute the following instructions:

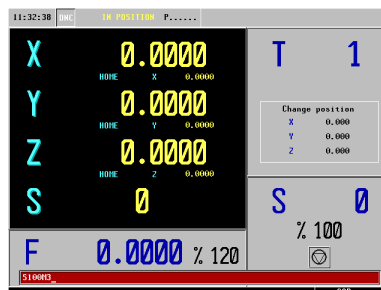
1. From the Jog Screen select the ISO key



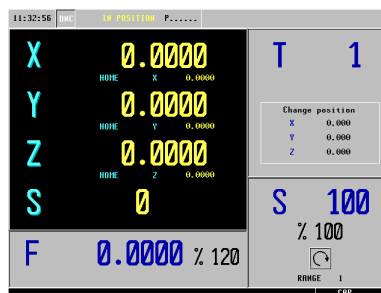
- In this example we will be turning on the spindle at a clockwise rotation at 100 rpm's
- The CNC will display a red window at the bottom of the screen



2. Type in S100 M3



3. Press Cycle Start



*\*Note: The spindle speed has changed to 100 and the icon has changed to a clock-wise position.*

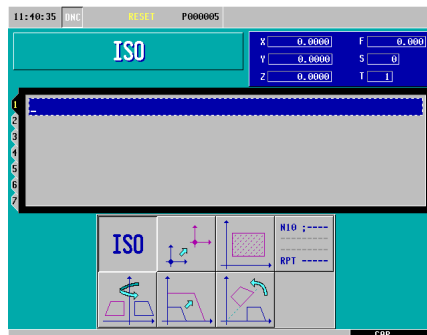
# INSERTING CODE IN-BETWEEN CANNED CYCLES

This feature allow us to insert M and G-Codes in-between our cycles in our library. In this example we will be inserting a G54 to start our program and a M00 in-between each cycle. The M00 will stop the program from executing the next programmed cycle until we hit cycle start. This feature is helpful in examining the tool after the completion of each cycle.

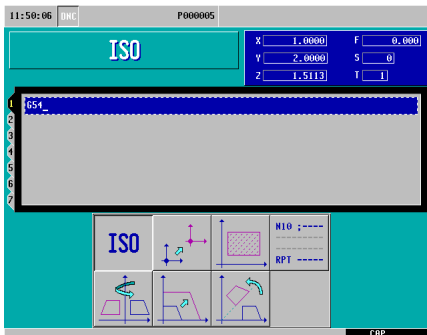
1. From the Jog Screen select the P.PROG key



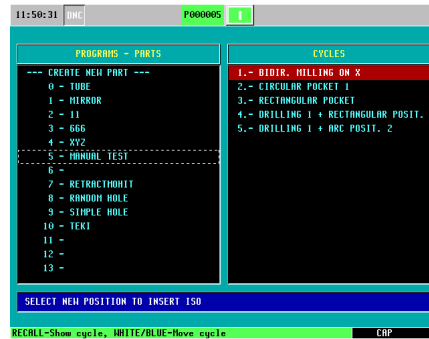
2. Select the ISO key



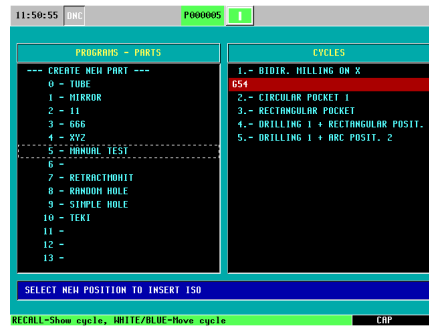
3. Type in G54



4. Select the P.PROG Key

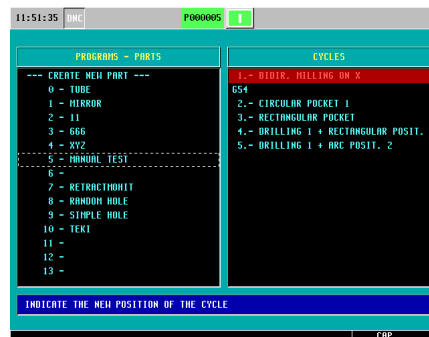


5. Press Enter



*\*Note: Make sure our first programmed cycle is highlighted in red before hitting enter.*

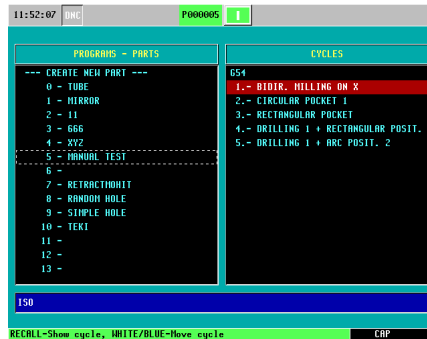
6. Highlight our first programmed cycle and select the Half-Key



*\*Note: This allows us to insert our first programmed cycle underneath the G54.*

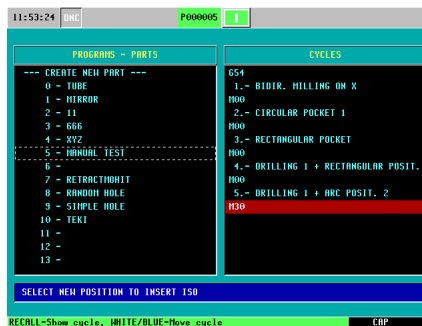


7. Arrow down to G54 and hit enter.



8. Now to insert M00 execute the following commands:

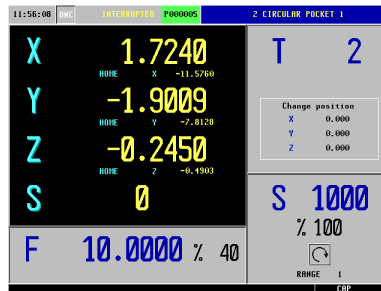
- Select the ISO Key
- Clear out the G54
- Type in M00
- Select the P.PROG Key
- Highlight the first Canned Cycle (BIDR. MILLING ON X)
- Press Enter
- Now you can highlight the remaining Canned Cycles one at a time and by pressing enter you will insert M00 in-between each of them



# TOOL INSPECTION

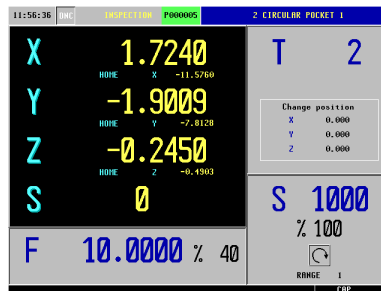
This feature allows us to examine our tool at any point while executing our program. Whether the tool breaks or if you are checking the status, this feature will allow you to stop your program, move your axes around and begin your program from the point you have left off at.

1. Press the STOP Key while the program is in execution



*\*Note: You will notice on the top of the screen it will read INTERRUPTED*

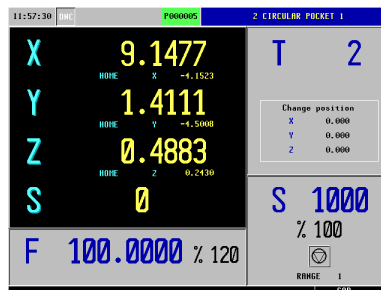
2. Press T (this is for Tool Inspection)



*\*Note: Notice the top of your screen now reads INSPECTION*

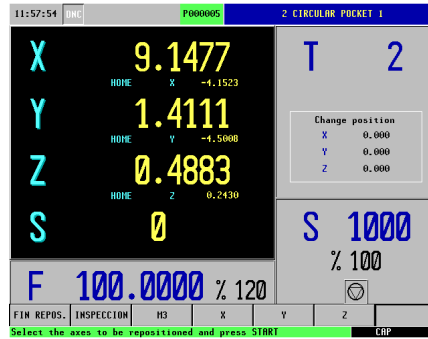
3. Execute the next step of commands:

- Jog your Z Axis up
- Turn off your spindle
- Jog your X and Y Axis out of the way

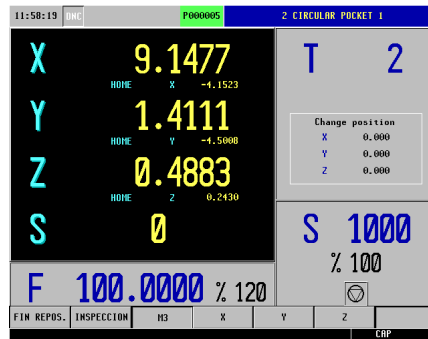


*\*Note: Notice the coordinate values are different from when we stopped the program*

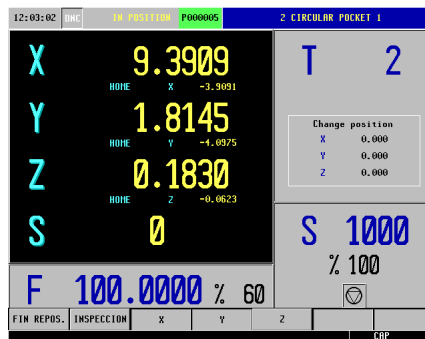
4. After examining the tool or changing it, press Cycle Start



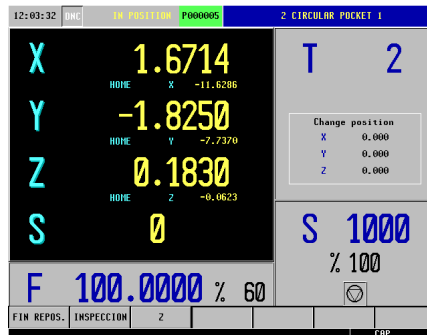
5. Select M3 and the Cycle Start to start your spindle



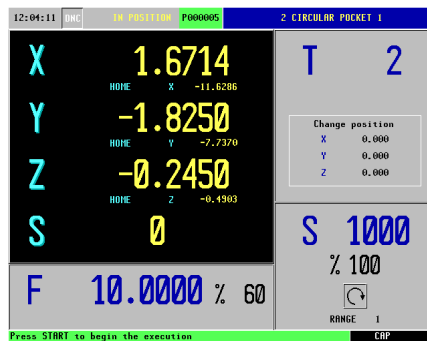
6. Select X and Y and then Cycle Start so these two axes are repositioned first



7. Select Z and then Cycle Start to reposition your Z Axis



8. Now press Cycle Start to begin the execution of your program



## QUICK REFERENCE GUIDE (MC)



**F1: TOOL CALIBRATION**



**F3: PROFILING CYCLE**



**F4: SURFACE MILLING**



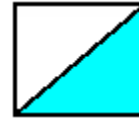
**F5: 2-D POCKET**



**F6: BOSS**  
- RECTANGULAR BOSS  
- CIRCULAR BOSS



**F7: POCKET**  
- RECTANGULAR POCKET  
- CIRCULAR POCKET



Used to change square symbols.



**POSITIONING CYCLE**



**DRILLING**



**TAPPING**



**REAMING**



**BORING**

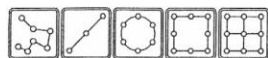
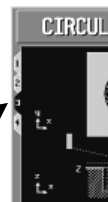


**Level cycle to select next page in an operation**

Press **SHIFT** then **ESC**.

(Used to switch from conversational to g-code)

Change Tabs





**To select different patterns for operations like Drilling, Boring, Reaming, Tapping and pockets**



## CONVERSATIONAL PARAMETERS

X	STARTING POINT ON X
Y	STARTING POINT ON Y
Z	PART Z COORDINATE
Zs	SAFETY DISTANE
P	TOTAL DEPTH
I	DEPTH OF CUT
Fz	PENETRATION FEEDRATE

### ROUGHING

F	FEEDRATE
S	SPINDLE RPM
	SPINDLE DIRECTION (CLOCKWISE / COUNTER CLOCKWISE)
T	TOOL NUMBER
D	OFFSET NUMBER
b	SIDEWAYS PENETRATION ANGLE
	CLIMB / CONVENTIONAL MILLING
D	ROUGHING PASS

### FINISHING

F	FEEDRATE
S	SPINDLE RPM
	SPINDLE DIRECTION (CLOCKWISE / COUNTER CLOCKWISE)
T	TOOL NUMBER
D	OFFSET NUMBER
Q	SIDEWAYS PENETRATION ANGLE
	CLIMB / CONVENTIONAL MILLING
d	FINISHING PASS
N	NUMBER OF FINISHING PASS
dz	FINISHING PASS IN Z

---

# FAGOR



## SIMPLE MC CYCLES

### SLOT MILLING

X	0.00
Y	
Z	
X	0.00
L	4.00
H	0.00
Zs	1.00
P	0.50
Fz	10.00

### INTERNAL THREAD

X	0.000	F	0.000
Y	0.000	S	0
Z	0.000	T	0
Xc	0.0000	Yc	0.0000
D	1.5000	H	2.0000
P	0.0500		
Zs	0.1000	Z	0.0000

### RECTANGULAR GROOVE

X	0.0000	F	0.000
Y	0.0000	S	0
Z	0.0000	T	1
Xc	0.0000	Yc	0.0000
L	12.0000	H	10.0000
D	1.0000	Corner	R
R	2.0000		
Zs	0.1000	Z	0.0000
P	1.0000	I	0.1000

### RING MILLING

X	0.0000	F	0.000
Y	0.0000	S	0
Z	0.0000	T	1
Xc	0.0000	Yc	0.0000
Re	4.0000		
Zs	0.1000	Z	0.0000
P	0.5000		
Fz	10.0000		

### ENGRAVING

X	0.0000	F	0.000
Y	0.0000	S	0
Z	0.0000	T	1
X	0.0000	SAFE-Z	0.1000
Y	0.0000	DEPTH(D)	1.0000
HEIGHT(H)	1.0000	Z	0.0000
LENGTH(L)	0.7500		
SPACE(E)	0.5000		
ANGLE(A)	0.0000		

### EXTERNAL THREAD

X	0.000	F	0.000
Y	0.000	S	0
Z	0.000	T	0
Xc	0.0000	Yc	0.0000
D	1.5000	H	2.0000
P	0.0500		
Zs	0.1000	Z	0.0000



**TABLE OF CONTENTS**

---

Slot Milling ..... 37

Ring Milling ..... 39

Thread Milling..... 41

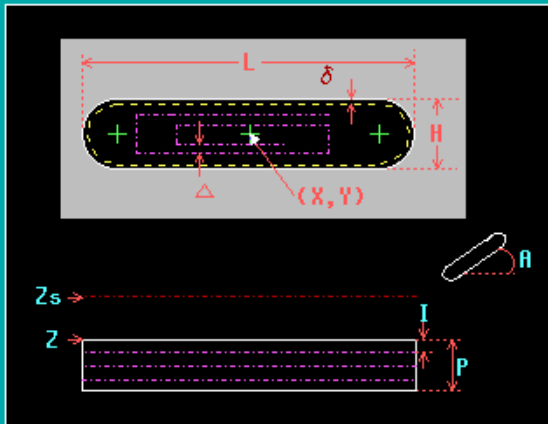
Rectangular Groove ..... 42

Part Skew ..... 44



## » Slot Milling

### SLOT MILLING



**ROUGHING**

Feed Rate

RPM

Tool

$\Delta$

Machining Direction

Spindle Direction

X

Y

Z

F

S

T

---

X

L

A

Y

H

---

Zs

P

Fz

Z

I

**ROUGHING**

Feed Rate

RPM

Tool

$\Delta$

Machining Direction

Spindle Direction

**FINISHING**

Feed Rate

$\delta$

$\delta z$

N

### Programming a slot:

**X,Y** - Enter the X and Y center coordinates.

**L** - Slot length.

**H** - Slot width.

**A** - If slot needs to be on an angle enter in degrees.

**Zs** - Safety distance for Z.

**Z** - Part Z coordinate.

**P** - Slot depth.

**I** - The amount you want to take off for each pass.

**Fz** - Feed rate for Z axis.

X

L

A

Y

H

---

Zs

P

Fz

Z

I

**Roughing setup:**

**Feed Rate** - Inches/minute.


**RPM** - Spindle speed.

**Tool** - Tool number.

$\Delta$  - Roughing pass. (Step over)

**Machining Direction** - CW is for clockwise and CCW is for counter clockwise.

**Spindle Direction** - Forward or reverse.

ROUGHING	
Feed Rate	80.00
RPM	800
Tool	1
$\Delta$	0.1000
Machining Direction	CW
Spindle Direction	

**Finishing setup:**

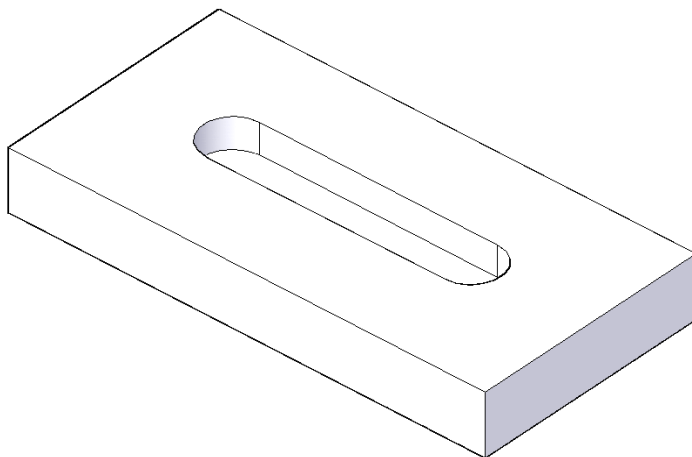
**Feed Rate** - Inches/minute.

$\bar{\delta}$  - The amount you want to leave on the sides of slot for your finishing pass.

**N** - Number of finishing passes.

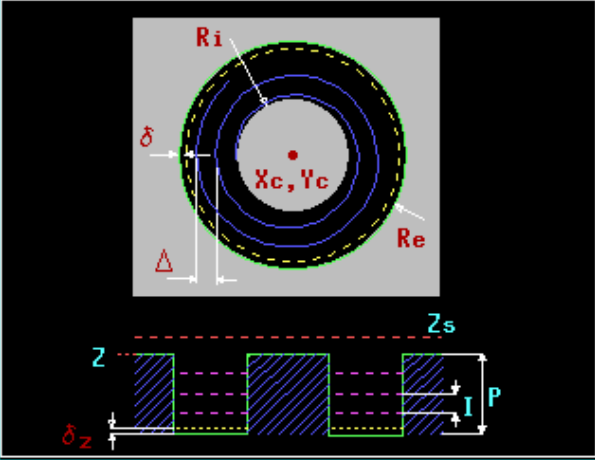
$\bar{\delta}z$  - The amount you want to leave on the floor of the slot for your finishing pass.

FINISHING	
Feed Rate	80.00
$\bar{\delta}$	0.0100
$\bar{\delta}z$	0.0100
N	2



## » Ring Milling

### RING MILLING



X	<input type="text" value="0.000"/>	F	<input type="text" value="0.000"/>
Y	<input type="text" value="0.000"/>	S	<input type="text" value="0"/>
Z	<input type="text" value="0.000"/>	T	<input type="text" value="0"/>

Xc	<input type="text" value="0.0000"/>	Yc	<input type="text" value="0.0000"/>
Re	<input type="text" value="4.0000"/>	Ri	<input type="text" value="1.0000"/>

---

Zs	<input type="text" value="0.1000"/>	Z	<input type="text" value="0.0000"/>
P	<input type="text" value="0.5000"/>	I	<input type="text" value="0.1250"/>
Fz	<input type="text" value="10.0000"/>		

**ROUGHING**  
Feed Rate   
RPM   
Tool   
Delta

**FINISHING**  
Feed Rate   
delta   
delta\_z

Machining Direction   
Spindle Direction

N

### Programming a ring:

**Xc,Yc** - Enter the X center and Y center coordinates.

**Re** - Outside radius.

**Ri** - Inside radius.

**Zs** - Safety distance for Z.

**Z** - Part Z coordinate.

**P** - Ring depth.

**I** - The amount you want to take off for each pass.

**Fz** - Feed rate for Z axis.

Xc	<input type="text" value="0.0000"/>	Yc	<input type="text" value="0.0000"/>
Re	<input type="text" value="4.0000"/>	Ri	<input type="text" value="1.0000"/>

---


Zs	<input type="text" value="0.1000"/>	Z	<input type="text" value="0.0000"/>
P	<input type="text" value="0.5000"/>	I	<input type="text" value="0.1250"/>
Fz	<input type="text" value="10.0000"/>		

**Roughing setup:**

**Feed Rate** - Inches/minute.

**RPM** - Spindle speed.

**Tool** - Tool number.

ROUGHING			
Feed Rate	80.00	Machining Direction	CW
RPM	800		
Tool	1	Spindle Direction	
$\Delta$	0.1000		

$\Delta$  - Roughing pass. (Step over)

**Machining Direction** - CW is for clockwise and CCW is for counter clockwise.

**Spindle Direction** - Forward or reverse.

**Finishing setup:**

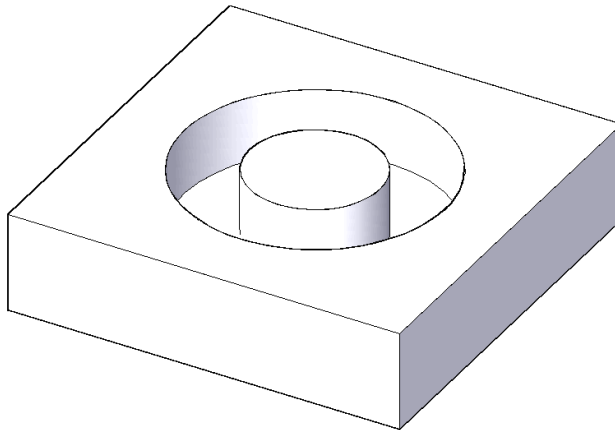
**Feed Rate** - Inches/minute.

$\delta$  - The amount you want to leave on the sides of ring for your finishing pass.

**N** - Number of finishing passes.

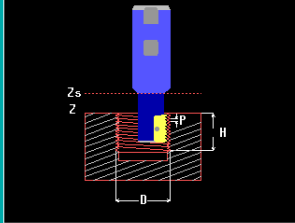
$\delta z$  - The amount you want to leave on the floor of the ring for your finishing pass.

FINISHING			
Feed Rate	80.00		
$\delta$	0.0100	N	2
$\delta z$	0.0100		



## » Thread Milling

### INTERNAL THREAD



X	0.000	F	0.000
Y	0.000	S	0
Z	0.000	T	0

Xc	0.0000	Yc	0.0000
D	1.5000	H	2.0000
P	0.0500		

---

Zs	0.1000	Z	0.0000
----	--------	---	--------

**ROUGHING**

Feed Rate  Machining Direction

RPM  Spindle Direction

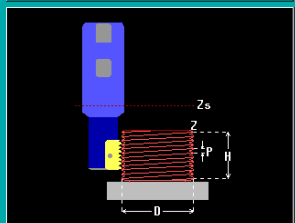
Tool

**FINISHING**

Feed Rate

$\delta$

### EXTERNAL THREAD



X	0.000	F	0.000
Y	0.000	S	0
Z	0.000	T	0

Xc	0.0000	Yc	0.0000
D	1.5000	H	2.0000
P	0.0500		

---

Zs	0.1000	Z	0.0000
----	--------	---	--------

**ROUGHING**

Feed Rate  Machining Direction

RPM  Spindle Direction

Tool

**FINISHING**

Feed Rate

$\delta$

### Programming a ring:

**Xc,Yc** - Enter the X center and Y center coordinates.

**D** – Thread diameter.

**H** – Thread height.

**P** – Thread pitch.

**Zs** - Safety distance for Z.

**Z** - Part Z coordinate.

Xc	0.0000	Yc	0.0000
D	1.5000	H	2.0000
P	0.0500		

---

Zs	0.1000	Z	0.0000
----	--------	---	--------

### Roughing setup:

**Feed Rate** - Inches/minute.

**RPM** - Spindle speed.

**Tool** - Tool number.

**ROUGHING**

Feed Rate  Machining Direction

RPM  Spindle Direction

Tool

**Machining Direction** - CW is for clockwise and CCW is for counter clockwise.

**Spindle Direction** - Forward or reverse.

### Finishing setup:

**Feed Rate** - Inches/minute.

**$\delta$**  - The amount of material you want to leave left for your finishing pass.

**FINISHING**

Feed Rate

$\delta$

## » Rectangular Groove

### RECTANGULAR GROOVE

X	0.0000	F	0.000
Y	0.0000	S	0
Z	0.0000	T	1

Xc	0.0000	Yc	0.0000
L	12.0000	H	10.0000
D	1.0000	Corner	C
C	2.0000		

Zs	0.1000	Z	0.0000
P	1.0000	I	0.1000
Fz	100.0000		

#### ROUGHING

Feed Rate	100.00
RPM	100
Tool	1
Δ	0.1000

Machining Direction  
Spindle Direction

#### FINISHING

Feed Rate	100.00
δ	0.1000
δz	0.1000

N 3

### RECTANGULAR GROOVE

X	0.0000	F	0.000
Y	0.0000	S	0
Z	0.0000	T	1

Xc	0.0000	Yc	0.0000
L	12.0000	H	10.0000
D	1.0000	Corner	R
R	2.0000		

Zs	0.1000	Z	0.0000
P	1.0000	I	0.1000
Fz	100.0000		

#### ROUGHING

Feed Rate	100.00
RPM	100
Tool	1
Δ	0.1000

Machining Direction  
Spindle Direction

#### FINISHING

Feed Rate	100.00
δ	0.1000
δz	0.1000

N 3

## Programming a Rectangular Groove:

**Xc,Yc** – Enter the X and Y center coordinates.

**L** – Rectangle length.

**H** – Rectangle height.

**D** – Rectangle width.

**C** – Enter chamfer.

**R** – Enter radius.

**Zs** – Safety distance for Z.

**Z** – Part Z coordinate.

**P** – Rectangle depth.

**I** – The amount you want to take off for each pass.

**Fz** – Feed rate for Z axis

Xc	0.0000	Yc	0.0000
L	12.0000	H	10.0000
D	1.0000	Corner	R
R	2.0000		

---

Zs	0.1000	Z	0.0000
P	1.0000	I	0.1000
Fz	100.0000		


**Roughing setup:**

**Feed Rate** - Inches/minute.

**RPM** - Spindle speed.

**Tool** - Tool number.

$\Delta$  - Roughing pass. (Step over)

ROUGHING	
Feed Rate	80.00
RPM	800
Tool	1
$\Delta$	0.1000
Machining Direction	CW
Spindle Direction	

**Machining Direction** - CW is for clockwise and CCW is for counter clockwise.

**Spindle Direction** - Forward or reverse.

**Finishing setup:**

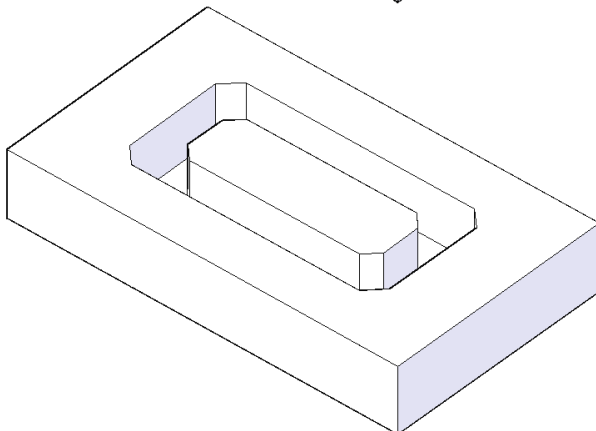
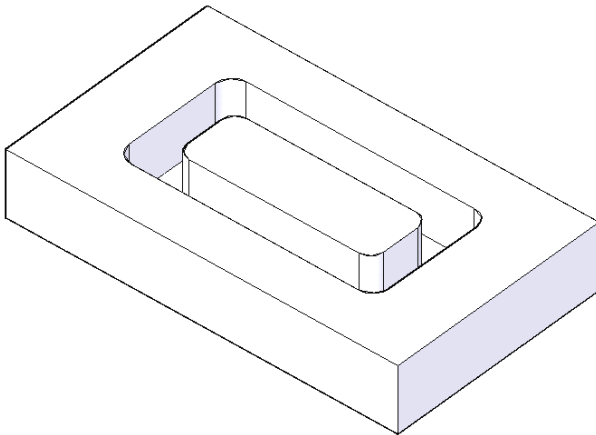
**Feed Rate** - Inches/minute.

FINISHING	
Feed Rate	80.00
$\delta$	0.0100
$\delta z$	0.0100
N	2

$\delta$  - The amount you want to leave on the sides of rectangle for your finishing pass.

N - Number of finishing passes.

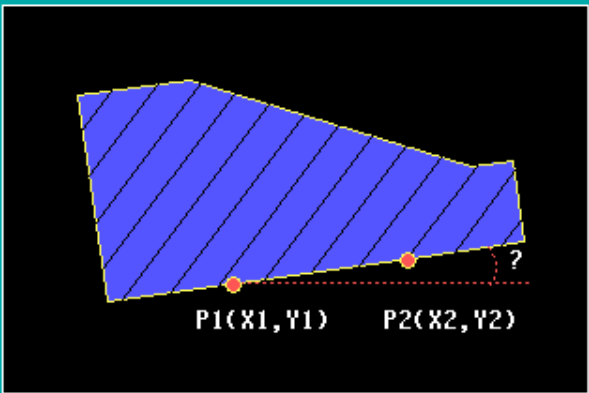
$\delta z$  - The amount you want to leave on the floor of the rectangle for your finishing pass.



## » *Part Skew*

10:58:13	DNC	IN POSITION P.....	MEMORY
----------	-----	--------------------	--------

### PART SKEWNESS



P1(X1,Y1)    P2(X2,Y2)

X	3.9708	F	24.000
Y	2.2631	S	0
Z	0.0000	T	1

**ACTIVE**

**A** 10.6197

1st POINT

X1 0.0000    Y1 0.0000

2nd POINT

X2 4.0000    Y2 0.7500

Active Part Skewness	CAP
----------------------	-----

### Programming a Part Skew:

**ACTIVE** – On or off.

**A**- Calculated skew angle.

**X1**- First X point coordinate.

**Y1**- First Y point coordinate.

**X2**- Second X point coordinate.

**Y2**- Second Y point coordinate.



**USB Specifications:** Use Blank USB stick.  
Do not use USB stick with encryption files on it.



**Programs:**

- 1) Use the format from CNC program to edit all future programs.
- 2) Programs must have six digits with .pim or .pit extensions. *Example: 000123.pim*
  - a) .pim for mills
  - b) .pit for lathes
- 3) The 1<sup>st</sup> line of the program should have %programcomment,MX,



**Initialize Connection:**

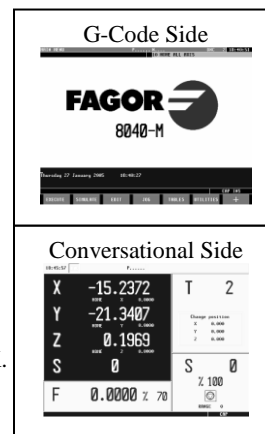
- 1) You must be on the G-code side of the CNC (Press Shift, then ESC).
- 2) Plug USB stick into USB port.
- 3) Press Main Menu, Utilities (**F6**), and then Explorer (**F7**).
- 4) Use the arrow keys to scroll down to USB DISK.
- 5) Press right arrow key above reset and all of your folders will appear under USB DISK.

**Transfer files from CNC to USB stick:**

- 1) Use arrow keys to scroll up to MEMORY.
- 2) Press transfer symbol  (**F7**).
- 3) Scroll down to program that you want to cut or copy.
- 4) Press ACTIONS (**F3**).
- 5) Highlight cut or copy and press enter.
- 6) Press the transfer symbol  (**F7**).
- 7) Use the arrow keys to scroll back down to USB DISK.
- 8) Go down to the folder you want to place the file in.
- 9) Press ACTIONS (**F3**).
- 10) Highlight paste and press enter.

**Transfer from USB stick to CNC:**

- 1) Locate the folder that your file is in.
- 2) Press the transfer symbol  (**F7**).
- 3) Highlight the file you want to cut or copy.
- 4) Press ACTIONS (**F3**).
- 5) Highlight cut or copy and press Enter.
- 6) Press the transfer symbol  (**F7**).
- 7) Scroll up to MEMORY.
- 8) Press ACTIONS (**F3**) and go to paste.



### Backup PLC, parameters, etc using a USB stick:

(Prior to backup, create a new directory in your USB stick. *Ex. My CNC Backup.*)

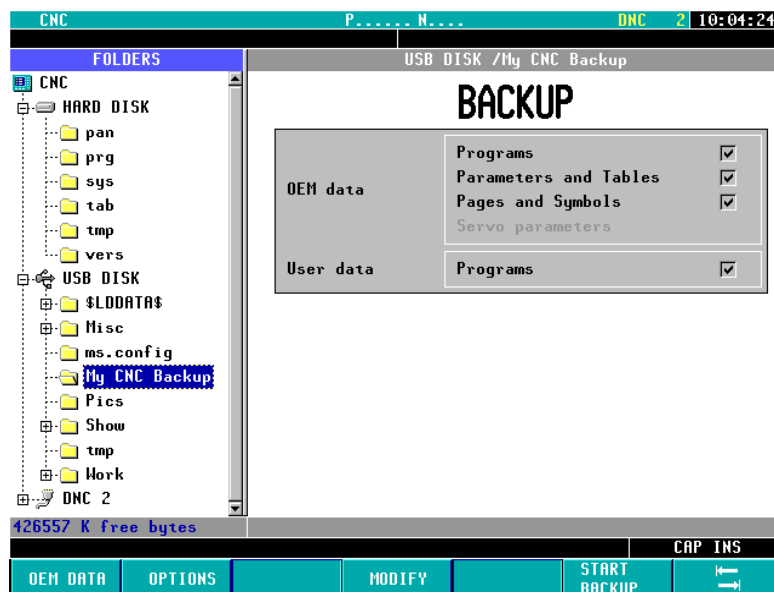
1. Must be on the G-code side of the controller. (Shift-Esc)
2. Press  $\pm$  (F7).
3. STATUS (F1).
4. CNC (F1).
5. BACKUP (F1).
6. Scroll down to USB DISK.
7. Press the right arrow key to open USB DISK.
8. Scroll to backup folder. (*Ex. My CNC Backup*)
9. Confirm that all parts of the backup are checked off.
10. Press START BACKUP (F6).

This feature is available on the following version software:

Ver.13 + for mills  
Ver.14 + for lathes



- To toggle from side to side.
- Right/left arrow to check/uncheck boxes.
- Use ENTER to save box.



\* Programs that are hidden will be encoded when opened in notepad \*

# FAGOR



## 8055 Turnkey Package



**Fagor Automation Corp.**  
2250 Estes Avenue  
Elk Grove Village, IL 60007

Tel.: 847-981-1500  
Fax.: 847-981-1311

e-mail: [fagorusa@fagor-automation.com](mailto:fagorusa@fagor-automation.com)  
**1-800-4A-FAGOR (1-800-423-2467)**

# INDEX

Z axis Quill kit instructions.....	49
X & Y axis assembly instructions.....	63
Limit switch kit.....	71
Limit switch kit installation	
Mounting arm kit.....	76
Introduction to 8055.....	80

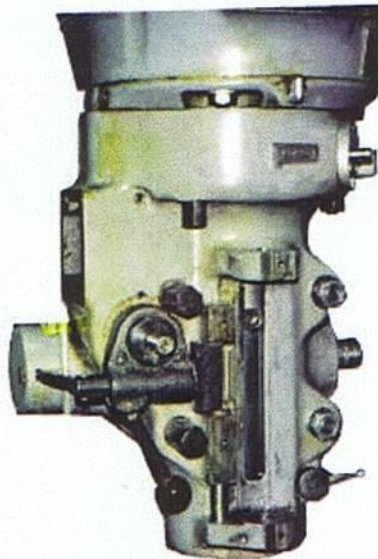
**Patented  
Elrod Z-Axis Quill Kit  
Universal  
Model# QU2C4**

Elrod Machine & Mfg. Inc.  
Patent 5,941,663 Rev 012400DE

**Machine Preparation**

1. Put the quill feed engagement lever in the disengaged position.
2. Remove the quill feed handle.
3. Leave the spindle in the disengaged position.
4. Remove the 3 10-24 SHCS on the feed disengage boss (@ 4 , 8 & 12 o'clock).
5. Remove the detent knob on the top of the 1/2-20 depth stop screw.
6. Remove the engagement lever on the bottom of the same rod.
7. Remove the circlip on the bottom of the screw and slide it out through the bottom while screwing the depth ring off from the top.
8. Remove the original depth stop block from the quill.
9. Slide the feed kick out rod out.
10. Remove the quill feed transmission cover on the left side of the head this is the cover you use to adjust your quill feed (.0015,.003,.006)
11. Tram the head of your mill.

Remove the paint from the bottom of the flange where the depth rod was removed from and also the wall that is perpendicular with it. (see figure 1)



**Figure1**



12. Using an indicator rotate the turret and indicate using the X axis until this wall is parallel with the X axis. (see figure 2)

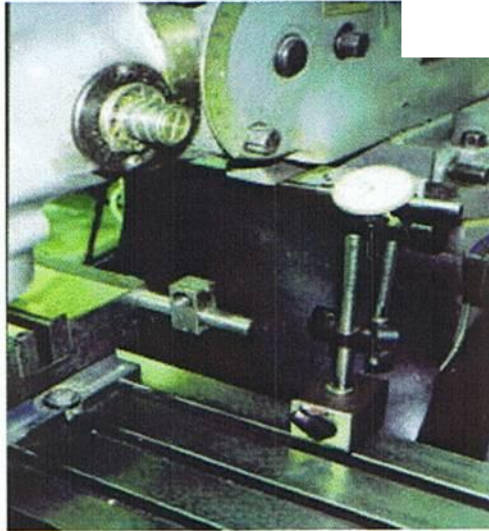


Figure 2

13. Retighten the turret.

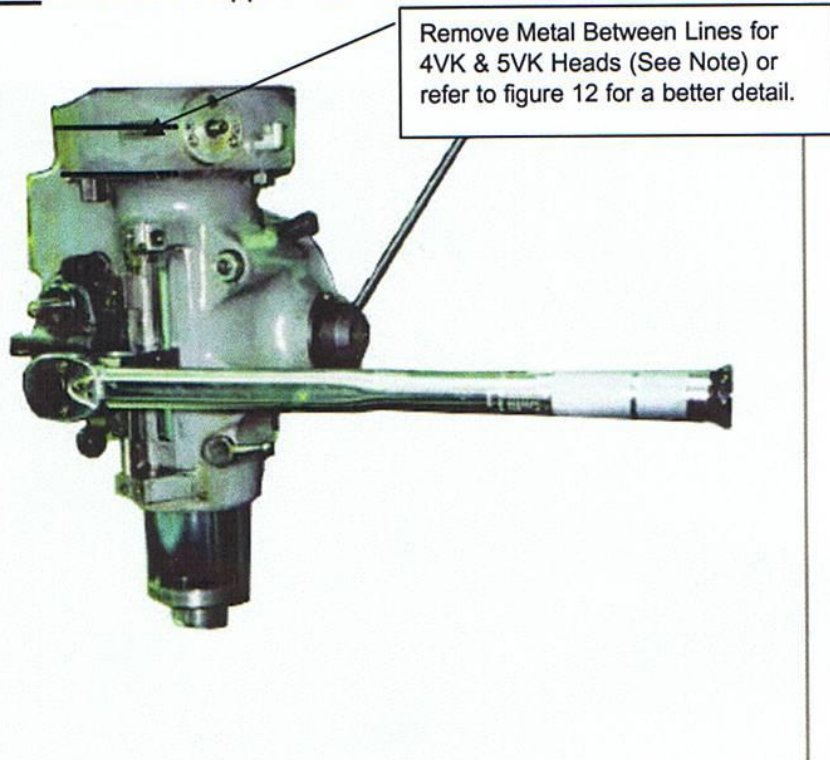
**Note:** Your quill drive can be installed without an installation tool. However Elrod Machine & Mfg. Inc. supplies a print for the tool and we feel it significantly reduces the installation time. This tool should take the average lathe hand 15-30 min. to complete or Elrod Machine & Mfg. Inc. can supply you one for \$25.00.

**Note to 5HP Oriental Mill Owners (4&5 VK).**

*In order for your Quill Drive to remain as rigid as possible we require you to remove some metal on the lower casting of your head, this is to allow clearance for the pulley and belt. After installing the ballnut block the dimension from the front of the block to the face of the head that was ground is 1.875" The spot to grind is specified on Figure 12. Please make sure you grind enough to install and remove the pulley with the screw installed in the quill drive! There is plenty of stock on the casting in this area do not be concerned!*

## Quill Unit Installation

1. Fasten the supplied ball nut block to the quill using the supplied 3/8-24 S.H.C.S. **torque to 40-44 ft lbs.** the 5/16-24 tapped hole is at the bottom. **(see figure 3)**



**Figure 3**

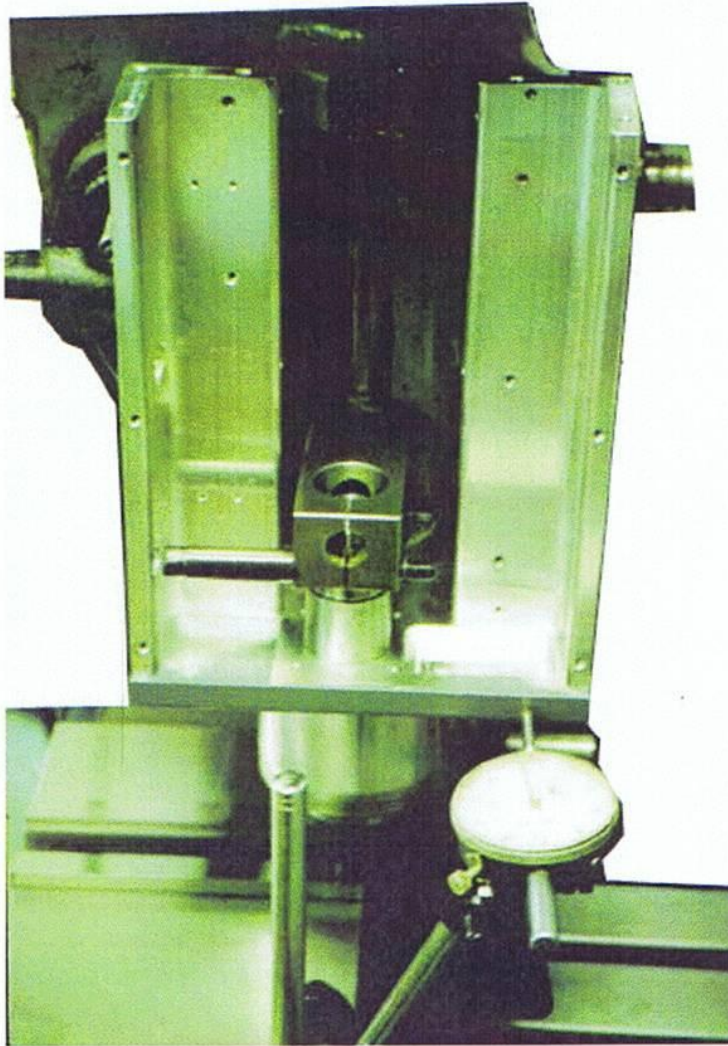
2. Place the lower assembly underneath the flange you scraped.
3. Place the 7/16-14 x 2" FHCS through the lower assembly and the lower flange.
4. Place the supplied hard washer and nut on the exposed end of the screw and tighten it just enough to hold it in place.
5. Place the quill in the middle of it's travel.
6. Place the installation tool in the bottom of the ball nut block and tighten the 5/16-24 SHCS firmly to hold it in place.
7. Roll the quill down until the installation tool is close to the bearing bore in the bottom assy. **(see figure 4)**



**Figure 4**

3. Remove the bearing access plate in the bottom of the lower assy.
4. Carefully roll the quill down and place the installation tool into the bearing bore, this is a 2 part operation. The bottom assy. has to be position under the tool while rolling the quill down to achieve your goal.
5. Keep the quill in the lower position using the lock lever.
6. Position and indicator on the front face of the bottom plate of the lower assy.  
(See Figure 5)





**Figure 5**

2. Move the X axis back + and – and rotate the lower assy. until the plate is parallel with the table. (Or no movement on the indicator is detected.)
3. Leave the indicator on one end of the plate at “0”.
4. Tighten the 7/16-14 screw with a hex wrench on the head of the screw while holding the nut with a combination wrench. Watch for any movement on the indicator detecting the lower assy is twisting out of alignment.

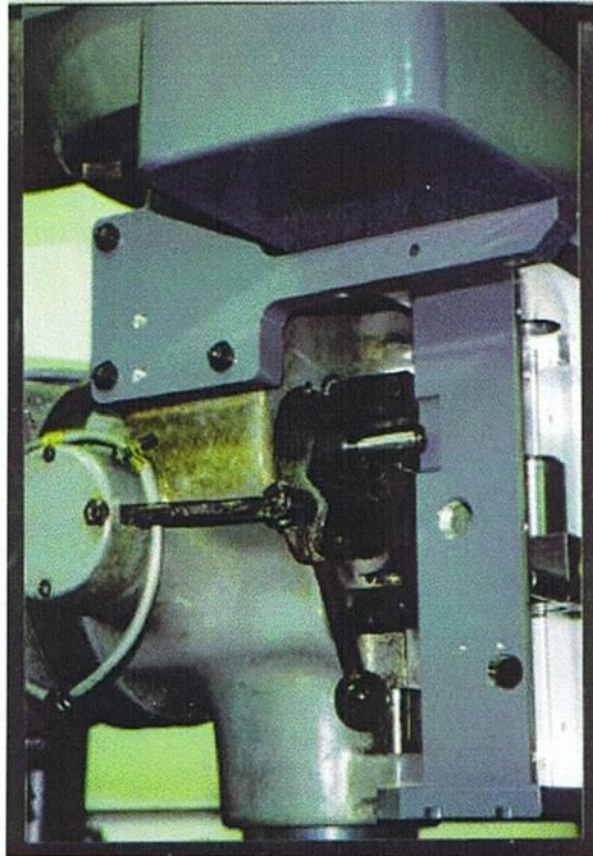
15. If this condition starts to happen unlock the quill lever and roll the quill up to the top position. Hold the screw head secure using a hex wrench and tighten the nut with a combination wrench from the top.
16. Unlock the lever and roll the quill down to the center of the travel once again.
17. Loosen the 5/16-24 SHCS on the ball nut block and place the installation tool in from the top of the ball nut block.
18. Retighten the 5/16-24 SHCS.
19. Install one 6201 bearing in the top bore of main housing.
20. Place the top housing on the top of the of the lower assy. (see figure 6)



**Figure 6**

21. Fasten the two together using the supplied 10-32 x 1" SHCS and hard washers but only enough so that the top housing can be moved around.

22. Roll the quill up until the installation tool is close to the bearing bore in the bottom of the top housing.
23. Move the top housing until it is approx. over the installation tool.
24. Carefully try to roll the quill up and move the top housing until the installation tool slides into the bearing bore.
25. Place the left side brace over the open cavity where the transmission feed cover was removed. (see figure 7)



**Figure 7**

26. The correct position is with the 45 Deg. Angle in the front to the bottom and pushed up against the milled surface on the bottom of the top housing.
27. Fasten it to the head using (4) 10-24 x 1 1/4" SHCS for Bridgeport or 5mm x 35mm SHCS for Oriental and hard washers supplied .



28. You may have to rotate the top housing to align the slots in the floor of the housing with the slots in the left mount.
29. Place the feed disengage mount on the front of the feed disengage lever boss. (see figure 8)

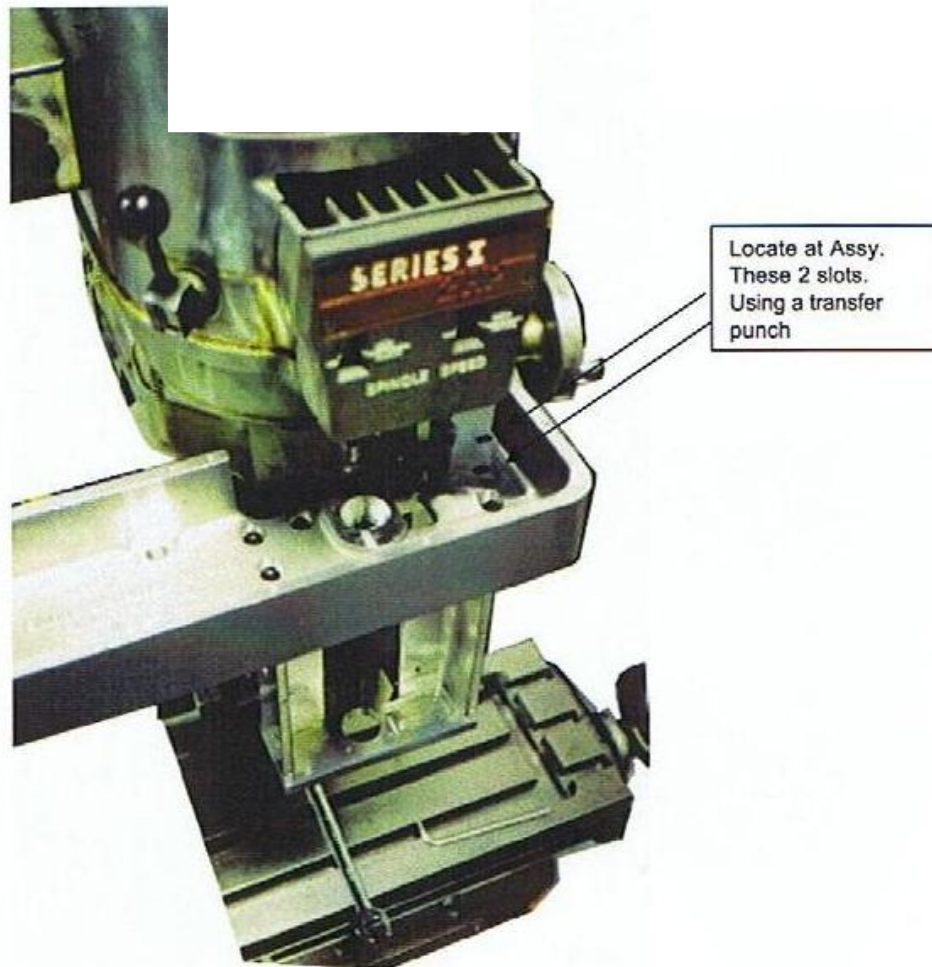


Figure 8

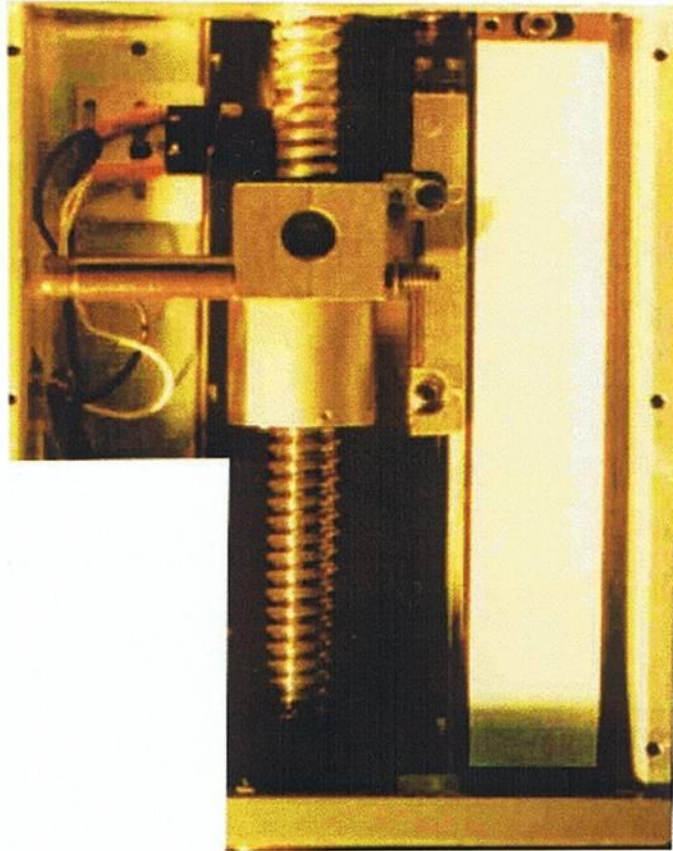
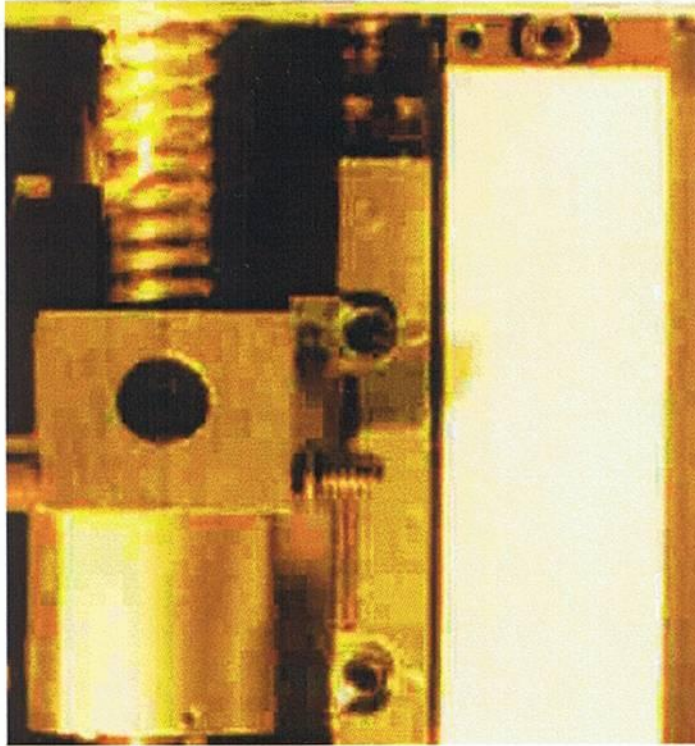


Figure 9

38. Feed the cable through the top housing and push on to the reader head do not securely fasten the cable yet.
39. Fasten the reader head mount to the ball nut block using (2) 10-32 x 5/8" S.H.C.S. supplied.
40. Roll the quill down to the bottom position.
41. Slide the reader head mount forward until it bears against the backside of the reader head on the scale.
42. Move the quill until the mounting holes are in line on the "Y" axis and slide the scale left to right until they are in line on the "X" axis.
43. At this point measure the distance between the right side of the scale and the inside wall of the angle and record this number.
44. Rotate the top of the scale until the same distance is achieved.
45. Align the scale by mounting a magnetic base with an indicator on the ball nut block and adjust the right side of the scale so it is parallel with the Z- axis within .005 T.I.R.
46. Tighten the (2) 10-32 x 5/8" SHCS to secure the reader head bracket.

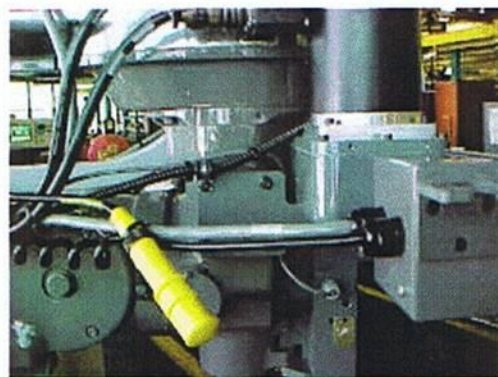


47. Fasten the reader head to the reader head bracket using the (2) 8-32 x 3/4" SHCS and 4 mm washers supplied. (see figure 10)



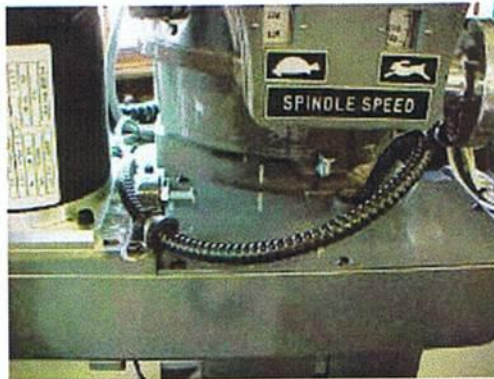
**Figure10**

48. Remove T20 screw and the red shipping bracket on the reader head.  
49. Route the scale cable around the left side of the head making sure there is a loop at the top of the cable quill drive for travel. Install the Scale Cable Clamps on the left side (See Figure 11A,B,C)

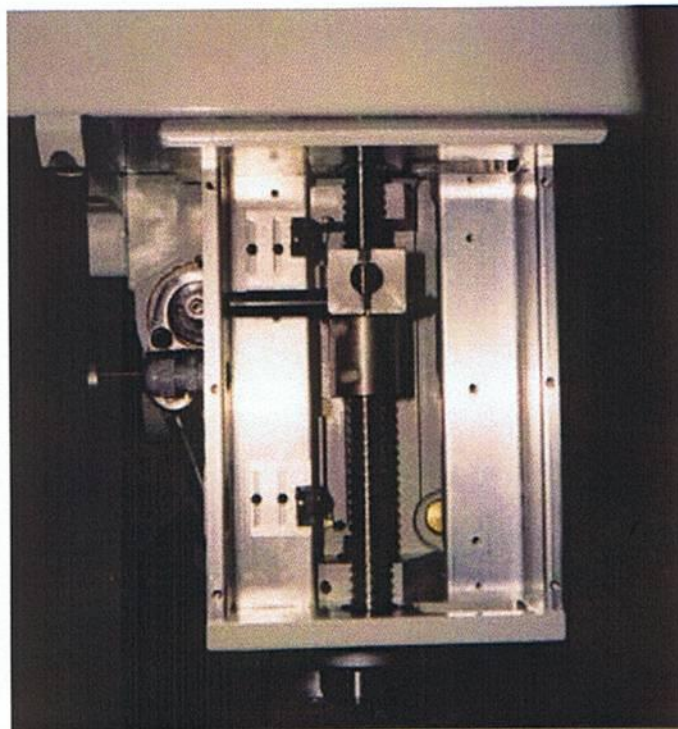


**Figure 11 A,B,C**

50. Place the other 6201 bearing in the bottom bore of the top housing.
51. Remove the installation tool from the ball nut block.
52. Place the quill in the middle of the travel.
53. Slide the ball screw assy. in from the top of the bottom plate of the lower assy. and in front of the ball nut block. Slide it through the bottom bearing hole in the lower assy. until it can be slid through the bottom of the ball nut block. Slide the ball screw assy. up until the ballnut journal engages into the bore of the ball nut. Snug the 5/16-24 SHCS to hold the screw Assy. in the ball nut block. (see figure

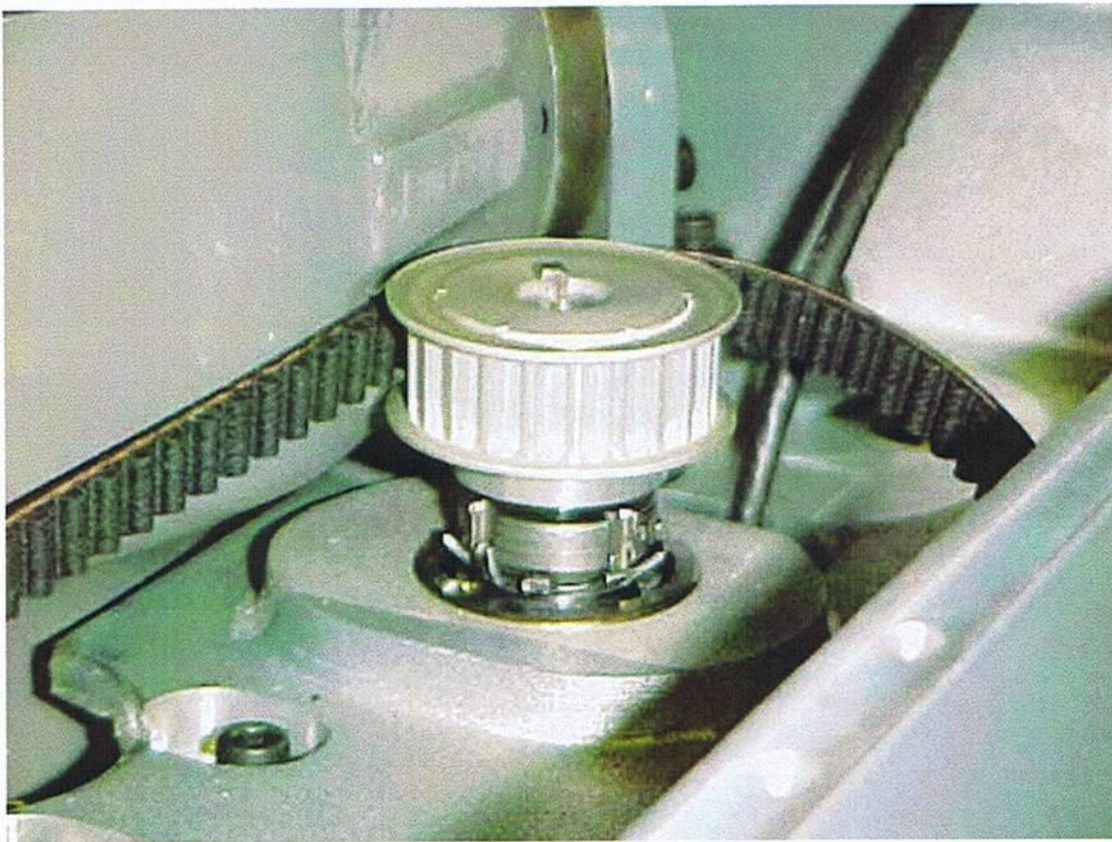


12)





- . Roll the quill up until the screw slides through the 6201 bearings completely.
- i. Place the W01 lock washer over the top of the screw and against the top of the 6201 bearing.
- j. Screw the N01 lock nut on the threads of the screw until it preloads the 6201 bearings.
- ' The main objective here is to tighten the nut enough to obtain little or no endplay in the bearings but not so tight that the bearings will not turn.
- }. Find a tooth on the washer and slot on the nut that line up and bend the tooth in the washer into the slot on the nut.
- }. Slide the 6201 bearing into the bottom of the lower assy. and on to the lower journal of the screw.
- }. Reinstall the bearing access plate on the bottom of the lower assy. fasten using the (4) 4-40 x 3/8" BHCS that were removed at dissassembly.
- l. Place the screw pulley (hub up) and key in place on the end of the ball screw and fasten securely. **(see figure 13)**

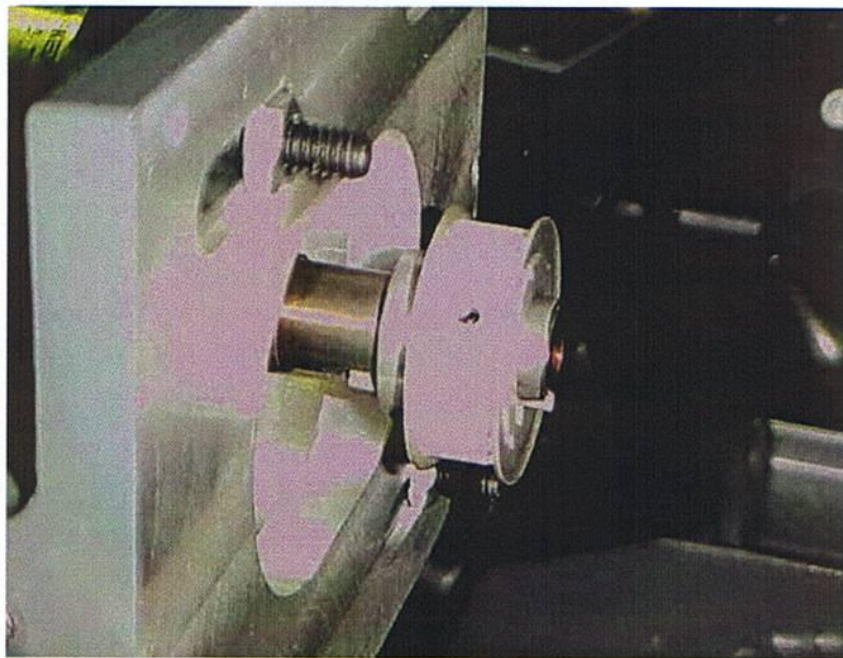


**Figure 13**

- 2. Rotate the pulley by hand until the ball nut is at the top position and adjust the top limit switch assy so it trips at full travel - .020". Slide it so the switch trips at this



- position. Precise actuation can be obtained by hearing the switch trip or by using an OHM meter to detect tripping.
63. Fasten securely using the (2) 4-40 x 3/8" BHCS supplied.
  64. Fasten the motor to the motor plate using (4) 1/4-20 x 1 1/2" SHCS and nuts. The nuts are placed in the 7/16 wide slots in the bottom of the motor plate and the screws are placed in from the top of the nema flange. **Make sure that when the assy. is placed on the top housing the connector on the motor faces the back of the machine.**
  65. Measure the distance from the top of the top housing to the first flange on the screw pulley. Record this number.
  66. Slide the motor pulley over the shaft of the servo motor until the same distance is achieved from the face of the motor plate to the first flange. Secure the pulley. (See figure 14)



**Figure 14**

67. Holding the motor assy in one hand place the 500M or 700M belt over the pulley with the other hand.
68. Set the Assy. on top of the top housing and slide the belt over the screw pulley.
69. Position the motor plate over the mounting holes on the top housing and fasten it using the (5) 10-32 x 3/4" SHCS supplied.
70. Screw the 1/4-20 pan head screw against the nema flange until the 194 belt is preloaded with tension. Tighten the 1/4-20 jam nut to lock the screw.
71. Install the strain relief in the left side of the lower assy.

72. Slide the limit switch cable through strain relief and fasten to the micro switch in the normally closed position or normally open position (**See your manual for this specification**).
73. Connect the motor cable to the controller.
74. Connect the limit switches to the controller.
75. Power up controller and set software travels in + and – directions after (If applicable).
76. Install Front cover using the (6) 8-32 x 3/8" BHCS supplied.

#### **QUILL DRIVE LUBRICATION**

1. Lubricate your quill drive screw with lubriplate white lithium grease.
2. Grease every 3 months or sooner if the screw looks dry.
3. Grease the screw through the 10-32 tapped hole in the ball nut.

#### **QUICK DISCONNECT OPERATION**

1. Jog the quill down to the minus "z" limit switch.
2. Swivel the hexagon tag marked 1/4 .
3. Use a 1/4 inch allen wrench and loosen the 5/16-24 s.h.c.s. in the ballnut.
4. Your quill is disconnected for manual operation.

#### **RETURN TO CNC OPERATION**

1. Roll the quill down until the ballnut block bottoms on the ball nut.
2. Retighten the 5/16-24 s.h.c.s. to 22 ft lbs (note: this screw must be tight in the quill drive to operate properly)
3. Swivel the hex tag down to cover access hole.

#### **QUICK DISCONNECT RULES**

1. **Never** jog the quill while in the manual mode!!!
- it is the operators responsibility to remove all z moves in a program when controller as a 2 axis cnc.

Copyright 2000  
Elrod Machine & M  
All rights reserved

red to  
#4 & 5  
ually  
ith  
& #40



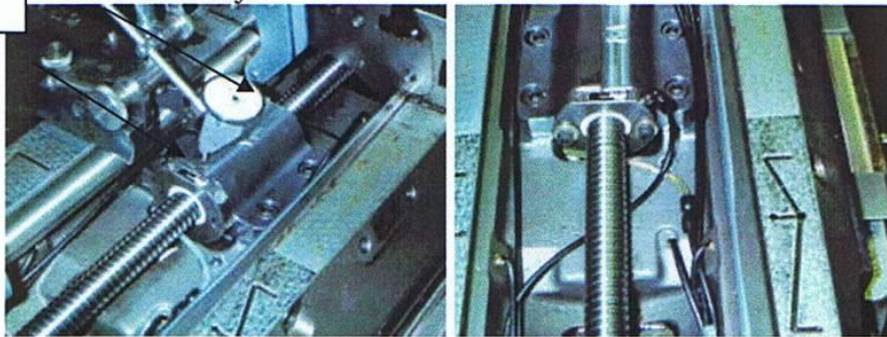


Elrod X&Y Bracket  
Model DOXLFT95  
Taper Lock Pulleys  
REV 20404

**X Axis Assy. Instructions**

1. Remove the handles from the screw along with the dials.
2. Remove the left bearing carrier on the X axis.
3. Slide the table to right as far as possible. (You might want to support the end hanging out.)
4. Remove the old yoke and slide the screw out to the left.
5. Install the supplied yoke there is and indicating register to help with alignment. Use an indicator with a surface gage to help with the alignment. Place  $\frac{3}{4}$ " to 1" Dia. rod inside the V place the surface gage on the scraped top of the saddle push the pins out in the surface gage far enough so they can ride against the protruding edge of the rod. Place the indicator needle on the alignment shoulder and slide the surface gage along the rod while holding the pins against the rod and adjust the yoke so the indicator needle is still. After alignment tighten the 4 mounting screws securely.

Indicate this shoulder parallel to back way.



6. Install the ball screw in the yoke, the long bearing journal is on the left side with most screws.
7. Locate the lube access hole on the nut flange and install the angled fitting and insert lube line in it.
8. Grease the duplex angular contact bearings and install them in the housing with the word thrust on each bearing facing each other.
9. Use the original bearing retainer ring to hold them in place.
10. Tighten the screws in the retainer equally to prevent any tilting.
11. Slide the housing on the screw journals and use the supplied S.H.C.S. to fasten it to the table.
12. Slide the 1 & 7/8 spacer on the screw until it comes in contact with the inner bearing race. For 2:1 ratio use 1 & 5/16 spacer.
13. Install the keyway on the screw.

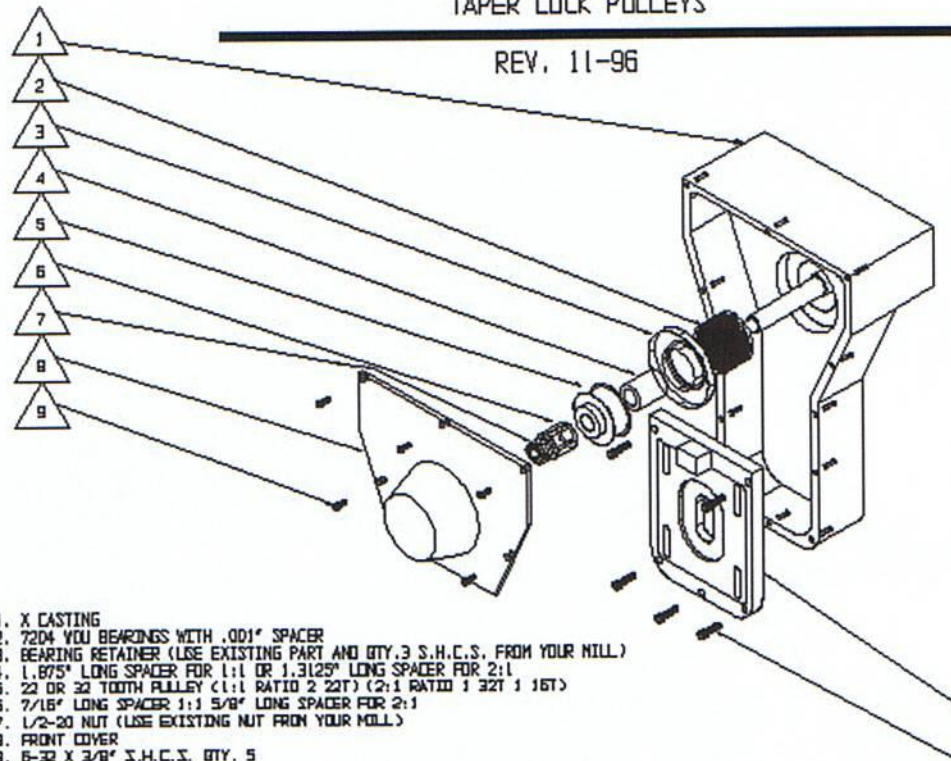
Elrod X&Y Bracket  
Model DOXLFT95  
Taper Lock Pulleys  
REV 20404

14. Slide the pulley and bushing on the screw bolts facing out.
15. Install the 7/16" long spacer and tighten it using the 1/2-20 nut for the screw end. For 2:1 ratio use 5/8" long spacer
16. Tighten the bolts on the pulley hub equally.
17. Place the motor on the motor mount plate, use the supplied 1/4-20 S.H.C.S and nuts with washers to fasten it . **Orient the motor so the cable receptacle faces down or back toward the rear of the machine.**
18. Measure the distance from the housing face to the first flange on the screw pulley and record it.
19. Slide the pulley and bush on the motor shaft bolts facing out for 1:1, small hub first for 2:1 until the same recorded distance is achieved from the mounting face of the motor plate.
20. Tighten the bolts equally on the hub for 2:1 tighten the set screws on the pulley.
21. Fasten the motor plate to the casting body using 10-32 x 3/4" S.H.C.S. Qty.(4).
22. Loosen the motor bolts and slide the motor up then install the 195L075 Belt for 1:1, 210L075 Belt for 2:1.
23. Pull the motor back creating tension on the belt and retighten the bolts. **(Careful, to much tension can prematurely wear out the motor\_bearings!)** .
24. Install the cover on the housing.



ELROD MACHINE X BRACKET ASSY VIEW AND PARTS LIST  
 MODEL 00XLFT95  
 TAPER LOCK PULLEYS

REV. 11-96



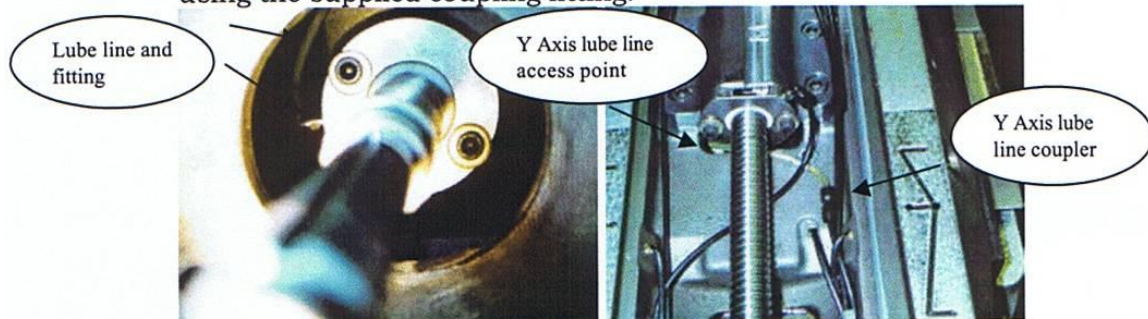
1. X CASTING
2. 7204 VDU BEARINGS WITH .001" SPACER
3. BEARING RETAINER (USE EXISTING PART AND QTY. 3 S.H.C.S. FROM YOUR MILL)
4. 1.875" LONG SPACER FOR 1:1 OR 1.3125" LONG SPACER FOR 2:1
5. 22 OR 32 TOOTH PULLEY (1:1 RATIO 2 22T) (2:1 RATIO 1 32T 1 16T)
6. 7/16" LONG SPACER 1:1 5/8" LONG SPACER FOR 2:1
7. 1/2-20 NUT (USE EXISTING NUT FROM YOUR MILL)
8. FRONT COVER
9. 6-32 X 3/8" S.H.C.S. QTY. 5
10. MOTOR PLATE
11. 10-32 X 3/4" S.H.C.S. QTY. 5
12. 1/4-20 X 1 1/2" S.H.C.S. & NUTS (FASTENS MOTOR TO MOTOR PLATE) QTY. 4
13. 3/8-16 X 1 1/2" S.H.C.S. OR 10MM X36 S.H.C.S. QTY. 4 (FASTENS CASTING TO TABLE)
14. 195L075 BELT 1:1 RATIO 210L075 BELT 2:1 RATIO



Elrod X&Y Bracket  
Model DOXLFT95  
Taper Lock Pulleys  
REV 20404

**Y Axis Assy. Instructions Using Graduated Dial**

1. Remove the handles from the screw along with the dial.
2. Remove the bearing carrier on the Y axis.
3. Slide the table to front as far as possible.
4. Install supplied angled fitting in the ball nut flange and insert the supplied lube line in the fitting.
5. Install the ball screw in the yoke while feeding the lube line in the angled hole of the yoke.
6. The lube line will come out next to the left side of the X axis ball nut, locate the original Y axis lube line and splice them together using the supplied coupling fitting.



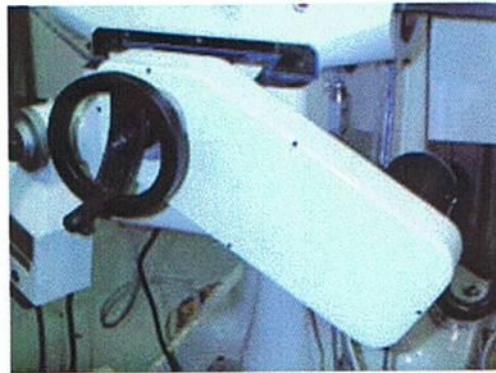
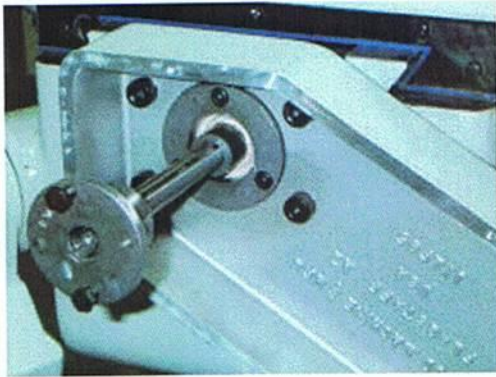
7. Use white Lithium grease to lubricate the duplex angular contact bearings and install them with the word thrust facing each other on each bearing in the Y housing.
8. Use the original bearing retainer ring to hold them in place.
9. Tighten the screws in the retainer equally to prevent any tilting.
10. Slide the housing on the screw journals and use the supplied S.H.C.S. to fasten it to the table. **(Note : On some machines with Erratic hole patterns the Y casting is supplied with no mounting holes. You should use transfer screws to locate hole centers and drill the Y casting.)**
11. Screw on the ballscrew extension. Use Loctite 609 to secure it or crosspin using a 1/16 dia. roll pin. **Permit the loctite to cure before proceeding!**
12. Slide the 1 & 3/16" long (1:1 ratio) or 15/16 (2:1 ratio) spacer on the screw until it comes in contact with the inner bearing race.
13. Install the keyway on the screw.
14. Install the 307L075 belt 1:1 or 322L075 belt for 2:1 on the pulley.
15. Slide the pulley and bush on the screw bolts facing out.
16. Slide the original dial hub on the screw next.



Elrod X&Y Bracket  
Model DOXLFT95  
Taper Lock Pulleys  
REV 20404

17. Place the cover on the bracket.
18. Place the original dial on the hub. Check for spacing between the dial and the cover and record it. If there is not enough space, shim out the dial. If there is an excess, face the dial hub off where it contacts the pulley. The ideal spacing should be .030-.050".
19. Take the cover off for now.
20. Screw the dial friction ring onto the dial hub.
21. Install the handwheel and tighten it using the 1/2-20 nut for the screw end.
22. Tighten the bolts on the pulley equally.
23. Remove the handwheel.
24. Place the motor on the NEMA flange mount, use the supplied 1/4-20 S.H.C.S. ,nuts and washers to fasten it.
25. **Orient the motor so the cable receptacle faces down.**
26. Measure the distance from the housing face to the first flange on the screw pulley and record it.
27. Slide the pulley and bush on the motor shaft with bolts facing out for 1:1, or small hub first for 2:1 until the same recorded distance is achieved.
28. Tighten the bolts equally for 1:1. For 2:1 tighten the set screws on the pulley.
29. Loosen the motor bolts and slide the motor towards the table then install the belt over the pulley.
30. Pull the motor back creating tension on the belt and retighten the bolts. **Careful, to much tension can prematurely wear out the motor bearings!**
31. Install the cover on the housing using the 6-32 S.H.C.S.
32. Slide the dial onto the hub.
33. Reinstall the handwheel.

Elrod X&Y Bracket  
Model DOXLFT95  
Taper Lock Pulleys  
REV 20404





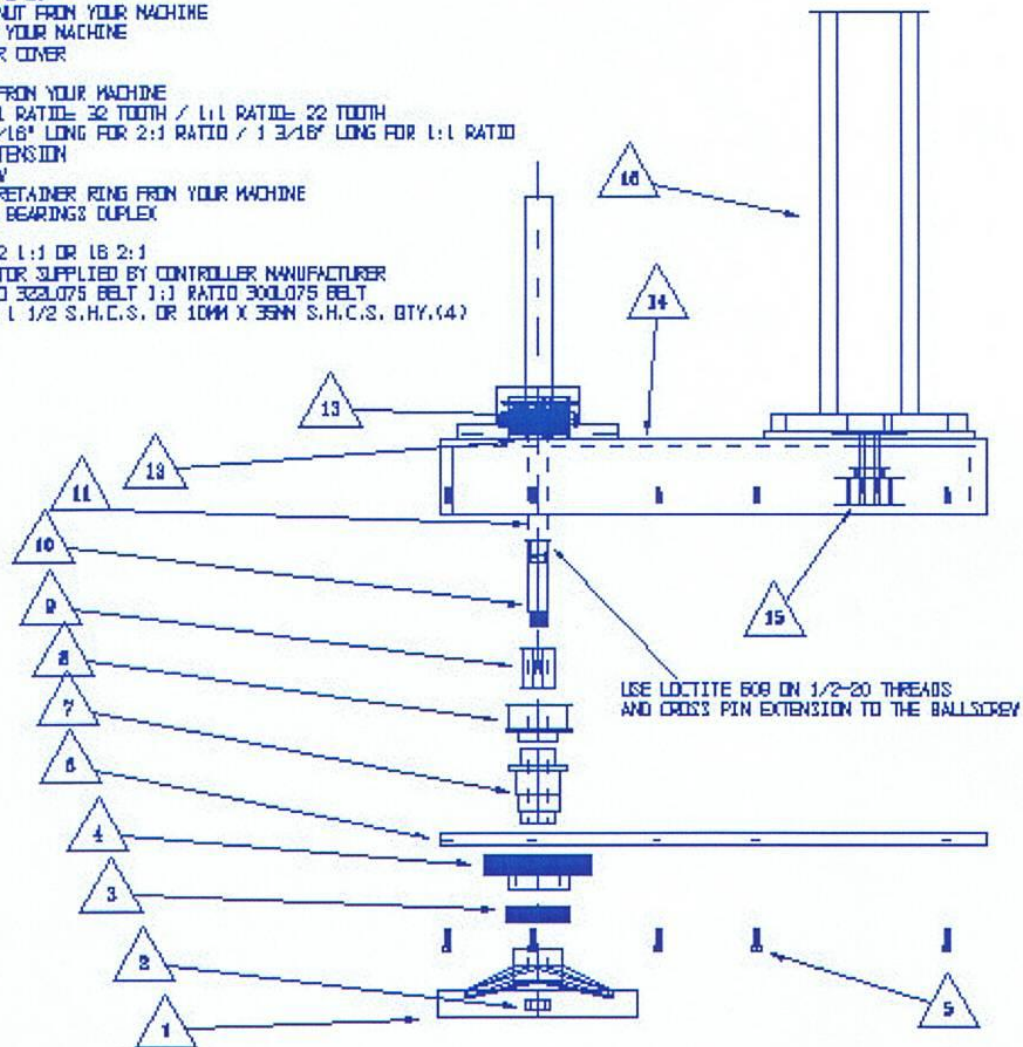


Elrod X&Y Bracket  
Model DOXLFT95  
Taper Lock Pulleys  
REV 20404

ELROD MACHINE Y AXIS BRACKET  
TAPER LOCK PULLEYS  
USING THE ORIGINAL GRADUATED DIAL  
EXPLODED ASSEMBLY

PARTS LIST

1. HANDWHEEL
2. JAN NUT 1/2-20
3. DIAL LOCKNUT FROM YOUR MACHINE
4. DIAL FROM YOUR MACHINE
5. SCREWS FOR COVER
6. COVER
7. DIAL HUB FROM YOUR MACHINE
8. PULLEY 2:1 RATIO= 32 TOOTH / 1:1 RATIO= 22 TOOTH
9. SPACER 15/16" LONG FOR 2:1 RATIO / 1 3/16" LONG FOR 1:1 RATIO
10. SHAFT EXTENSION
11. BALLSCREW
12. BEARING RETAINER RING FROM YOUR MACHINE
13. 7204 MDU BEARINGS DUPLEX
14. BRACKET
15. PULLEY 22 1:1 OR 18 2:1
16. SERVO MOTOR SUPPLIED BY CONTROLLER MANUFACTURER
17. 2:1 RATIO 300L075 BELT 1:1 RATIO 300L075 BELT
18. 3/8-16 x 1 1/2 S.H.C.S. OR 10MM X 35MM S.H.C.S. QTY.(4)

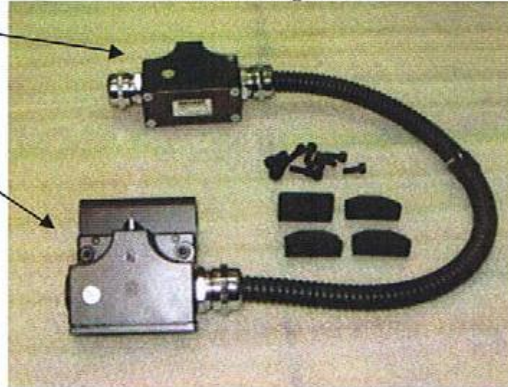


## Elrod Limit Switch Kit

Y Axis  
Switch and  
Bracket

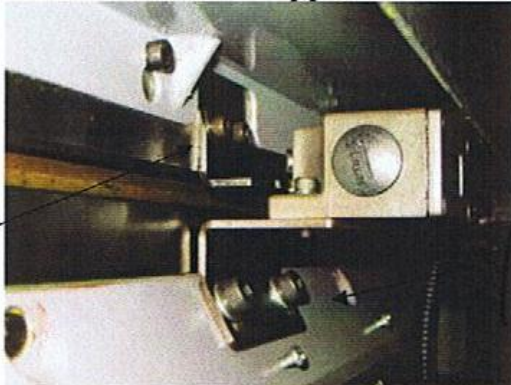
X Axis  
Switch and  
Bracket

**Kit Complete**



## X- Or Left Side Tripper Placement

Long Leg Tripper  
Bracket

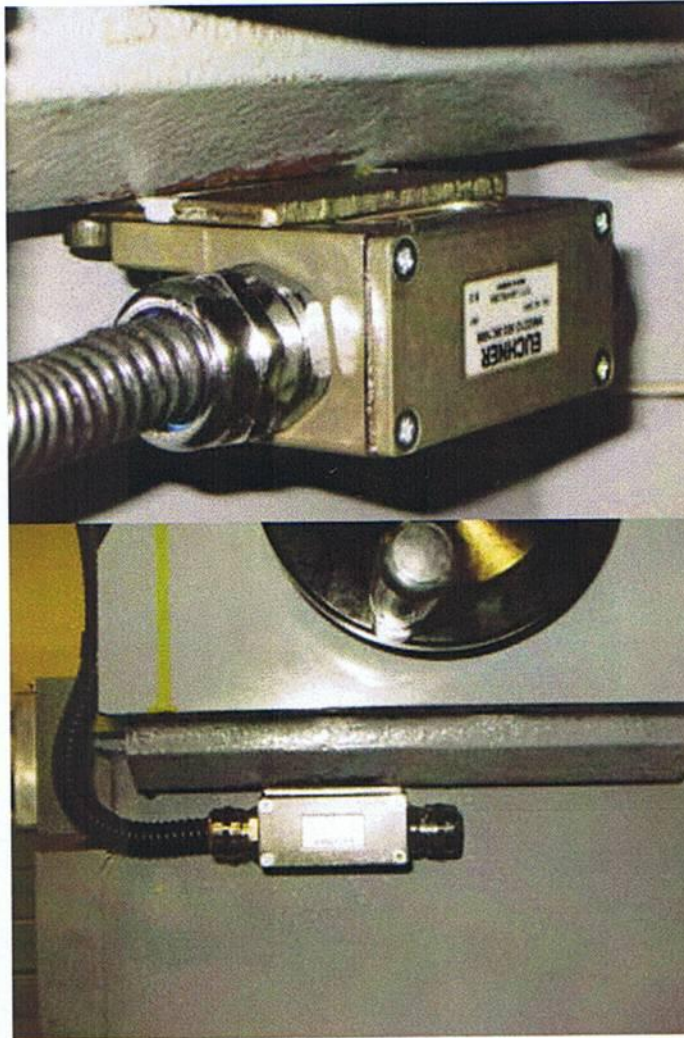


Mount X Axis  
Switch using  
original holes in the  
front face of the  
saddle your crossslide

**Use the long leg bracket for this side!**  
**Tripper positioned to trip the bottom plunger**



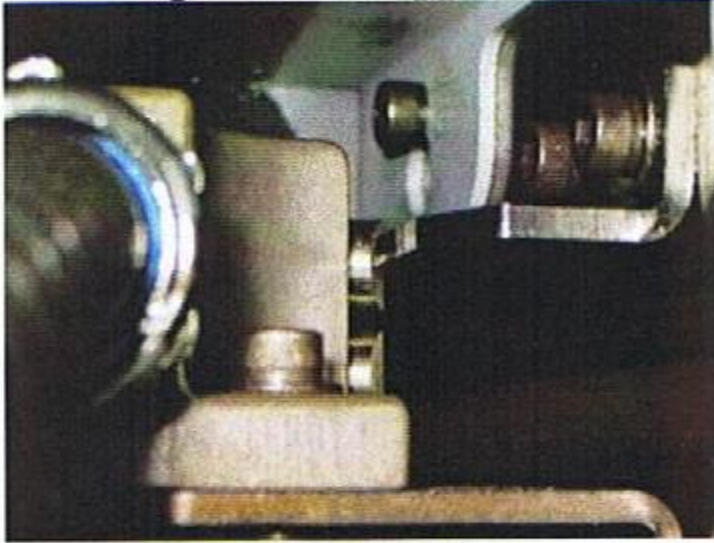
## **Y Axis Limit Switch Placement**



**Mount with 10-32 x 1/2" SHCS**

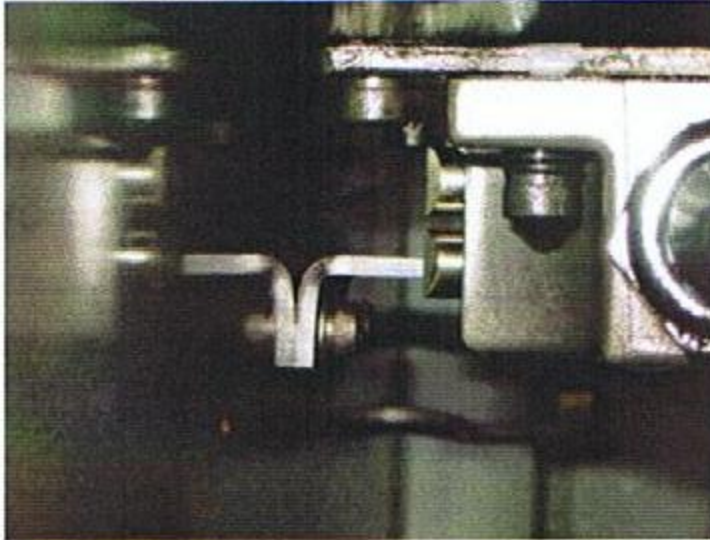
**Note: Position the switch plungers close enough to be actuated with trippers when mounted Approx. .750" to .812" from the side of the knee.**

## **X+ or Right Side Tripper Placement**



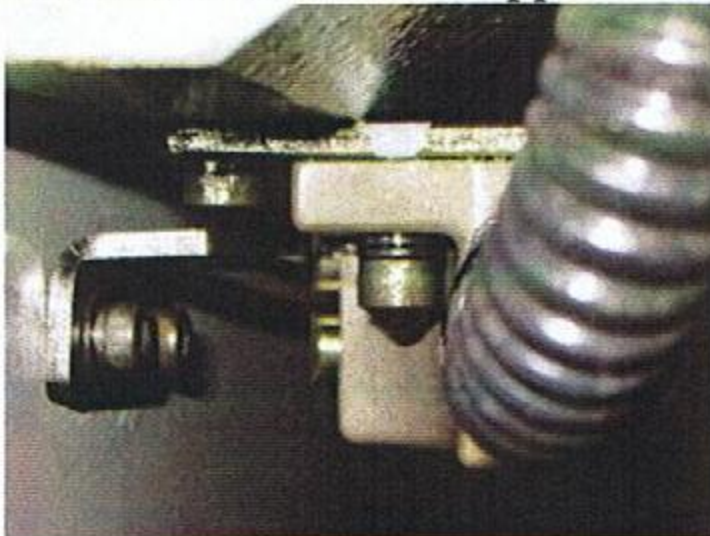
**Tripper positioned to trip top plunger**

**Y - or Saddle to the Back Tripper Placement**



**Tripper position to trip the bottom plunger**

**Y+ or Saddle to the Front Tripper Placement**



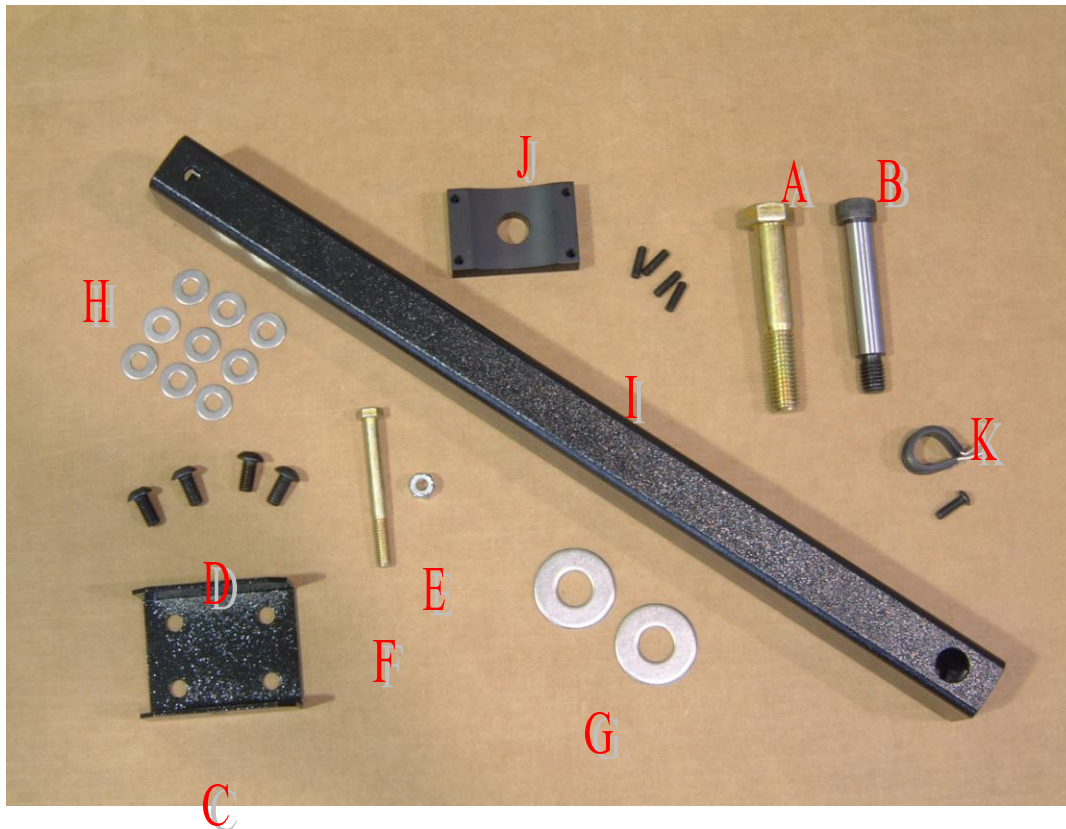
**Tripper positioned to trip the top plunger**



## Completed Installation



## TURNKEY MOUNTING ARM KIT ASSEMBLY



- 1 - Bolt hex head  $\frac{3}{4}$ -10 x 4  $\frac{1}{2}$  (A) (Note: used on oriental machines)
- 1 - Bolt SHCS shoulder bolt  $\frac{5}{8}$ -11 x 3  $\frac{1}{4}$  (B) (Note: used on Bridgeport machines)
- 1 – C shape bracket (C)
- 4 - Bolts BHCS  $\frac{3}{8}$ -16 x  $\frac{3}{4}$  (D)
- 1 - Nut nylock  $\frac{3}{8}$ -16 (E)
- 1 - Bolt hex head  $\frac{3}{8}$ -16 x 3  $\frac{1}{2}$  (F)
- 2 - Washer  $\frac{3}{4}$  ID (G)
- 9 - Washer  $\frac{3}{8}$  ID (H)
- 1 – 22” Arm (I)
- 1 – Saddle Bracket with 4 jacking screws M6 x 20mm (J)
- 1 – Motor cable clamp and BHCS  $\frac{1}{4}$  - 20 x  $\frac{3}{4}$  (Note: one per motor) (K)



## Fagor Automation Limit switch kit Installation

### X axis limit switch



## X+ or Right Side Tripper Placement



## **X- or Left Side Tripper Placement**

**Note: The X axis dog may be different as shown above.**



**Final installation of switch and two dogs on X axis**

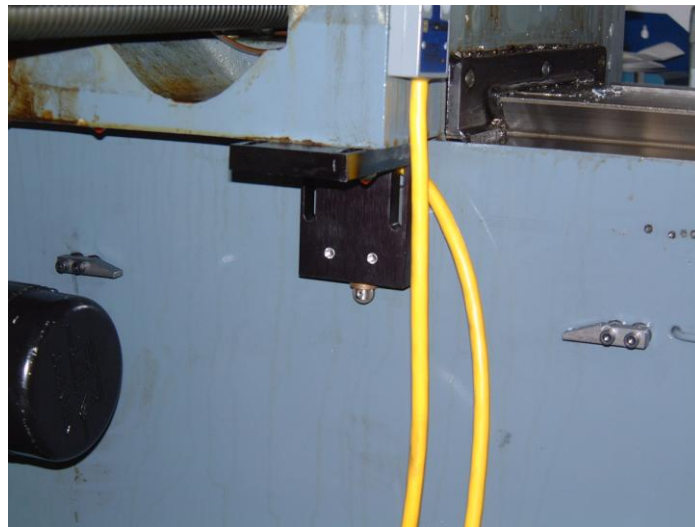
Y axis limit switch ( option 1 )



**Y+ Tripper Placement**



**Y- Tripper Placement**



**Final installation of switch and two dogs on Y axis**

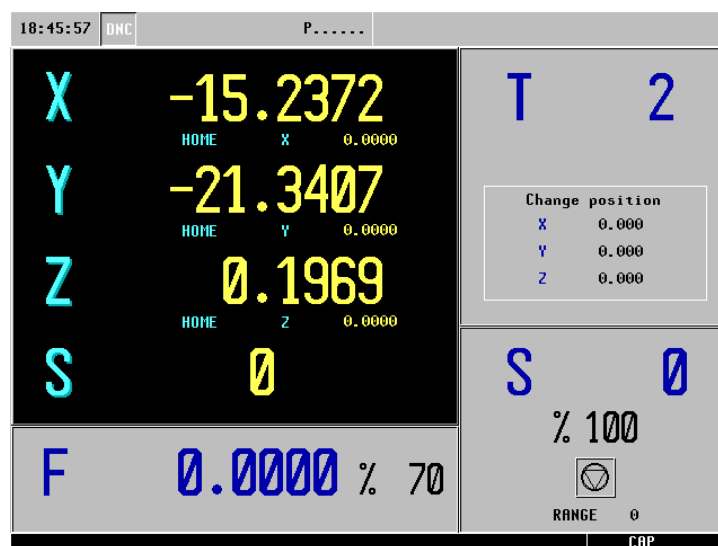
**Note: The system will use the switch first as a home switch. Once the control has found its home, It will then take and use the switch as an over travel switch for both the positive and the negative side of its travel.**

# INTRODUCTION TO CNC 8055 AND START UP

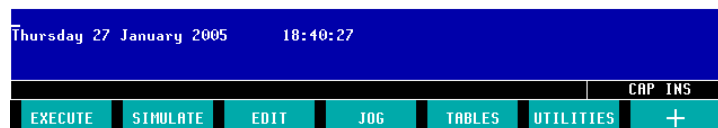
Note: The 8055 MC CNC control dual programming modes: **Conversational programming** and **G-Code programming**.

The **G-code side** of the control will be used for startup of the machine and programming in G-code language. The **conversational side** of the control will be used for icon-based programming.

To switch from one operating mode to another, press key sequence



Conversational Side



G-Code Side

# Confirmation of Movement / Backlash Compensation

To confirm that system is moving the correct distance (1" = 1") we must confirm the following: **Note: You need a dial indicator for this process**

1. Confirm that the PITCH for the axis is correct (P7) and that the encoder line count is correct (P8).
  - Select the MAIN MENU key
  - Select the + key (F7)
  - Select the MACHINE PARAMETERS key (F4)

2. Take the dial indicator place it on the table and use the spindle as a reference point to confirm the movement on the X axis.

Note: See figure 1

3. Go to the JOG mode and turn the jog switch from continuous mode to incremental mode (x1000). Every time the X+ or X- is pressed you will move .1000" at a time.
4. Press the X+ key the CNC screen displays that the X axis has moved .1000". Confirm with the dial indicator that it has moved the correct distance. Repeat the process until the CNC shows that you have moved 1". Confirming every time that the dial indicator shows the correct distance moved.
5. Repeat the same process as above in the X- direction. At this point you might have notice that the system shows some lost motion (backlash). Take note of the backlash, and Follow these steps to compensate for the backlash.
  - Select the MAIN MENU key
  - Select the + key (F7)
  - Select the MACHINE PARAMETERS key (F4)
  - Select the AXIS PARAMETERS key (F2)
  - Select the X axis key (F1)

- Using the down arrow key , cursor down and highlight parameter 14 (BACKLASH)
  - Select the MODIFY key (F2)
  - Enter the value that you would like to compensate for backlash and press ENTER
  - Confirm the parameter changes. Press RESET for the control to apply the changes.
  - Repeat steps 4 and step 5
6. Once the whole process has been completed you should be able to move both in the positive direction and negative direction .1000" at a time and confirm this with the dial indicator. The system should show that 1" = 1".

Figure 1



# Setting Software Limits

Before setting software limits we must confirm that the following items have been accomplished:

- Confirmed that all axes are under control
- Confirmed that all axes are jogging in the correct direction. In Axis Parameter AXISCHG (P13) and LOOPCHG (P26). Either Yes or No
- Confirmed that all axes are set for the correct resolution. 1' is equal to 1'
- Confirmed that all axes have been checked for backlash and compensated for backlash

Once we have confirmed that the task have been accomplished. Follow these steps:

- Home all axes ( please refer to the section in this manual “Homing the Control”)
- Once the control has been homed jog the X axis towards the positive direction stopping short of hitting its positive over travel dog.
- Make note of the X axis position on the screen.
- Jog the X axis towards the negative direction stopping short of hitting its negative over travel dog.
- Make note of the X axis position on the screen.
- Select the MAIN MENU key .
- Select the + key (F7).
- Select the MACHINE PARAMETERS key (F4).
- Select the AXIS PARAMETERS key (F2).
- Select the X axis key (F1).
- Using the down arrow key , cursor down and highlight parameter 5 (limit +).
- Select the MODIFY key (F2).
- Enter the value of the positive travel that was recorded earlier and press ENTER .
- Using the down arrow key , cursor down and highlight parameter 6 (limit -).
- Select the MODIFY key (F2).
- Enter the value of the negative travel that was recorded earlier and press ENTER.
- Confirm the parameter changes. These parameters require that you do a SHIFT RESET for the control to apply the changes.

**Note:** You can repeat the same process for Y and Z axis before you do a SHIFT RESET. It is not necessary to do one axis at a time.