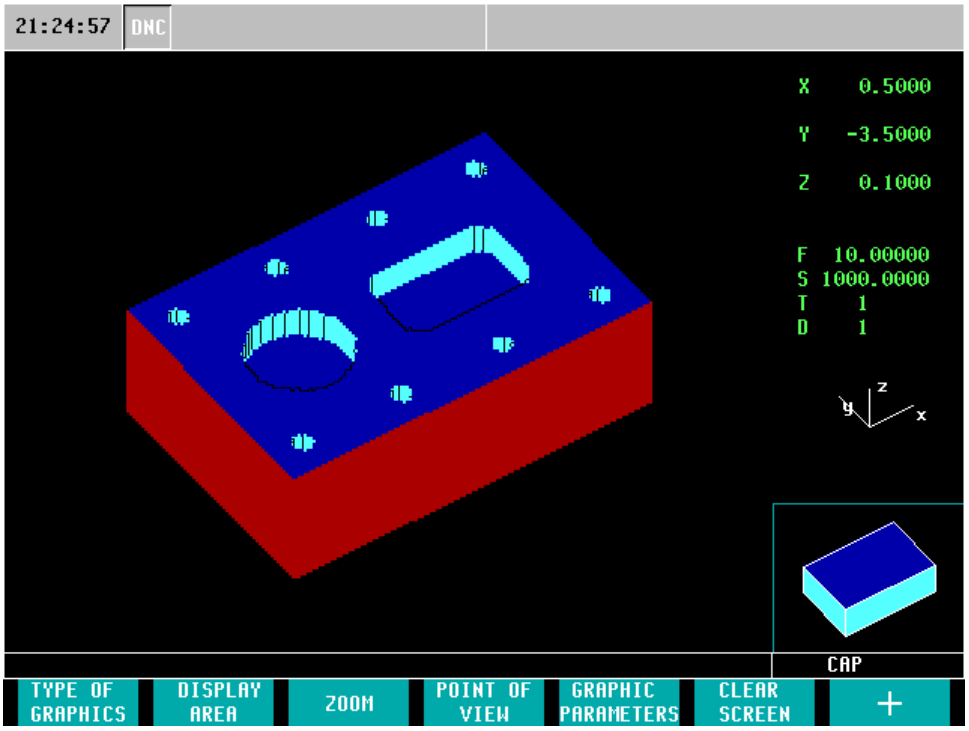
Chapters

1. MC Training Manual
2. Optional PCALL Cycles
3. USB Interface
4. Installation

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

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

FAGOR AUTOMATION
MC TRAINING MANUAL



Contents

Chapter 1

Chapter 2

Chapter 3

Chapter 4

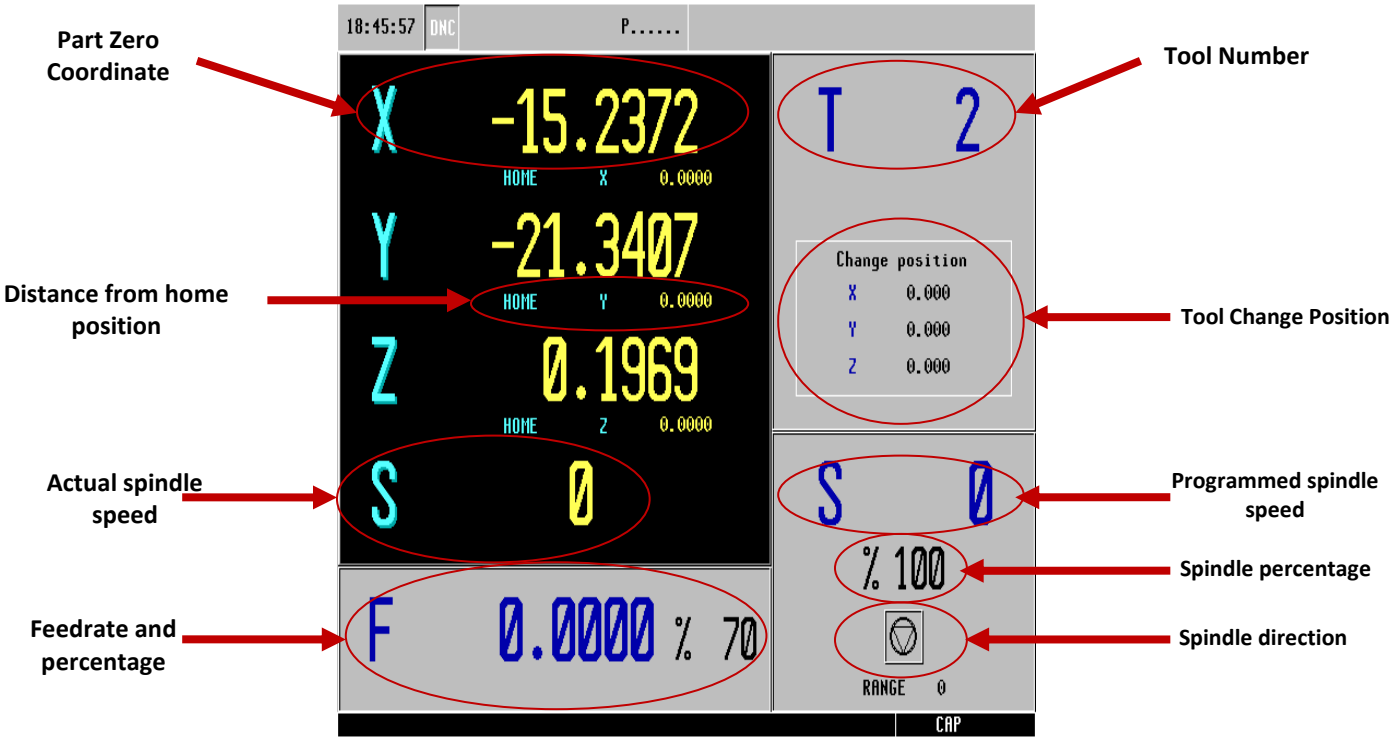
INTRODUCTION

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.

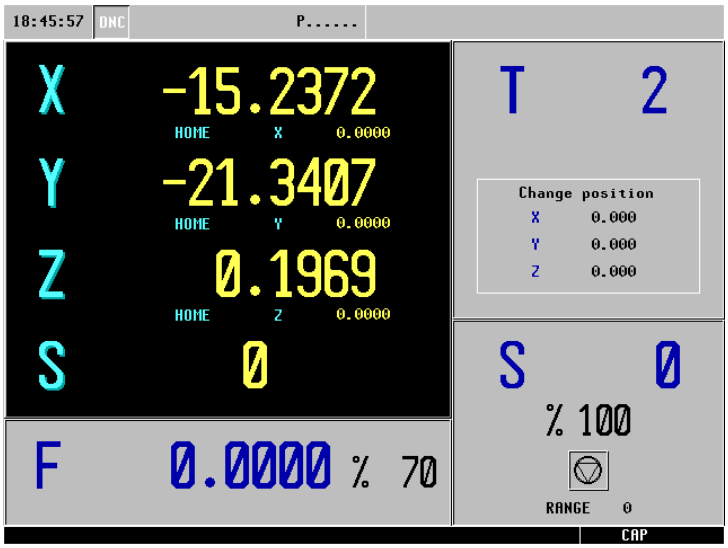
Conversational Screen Regions



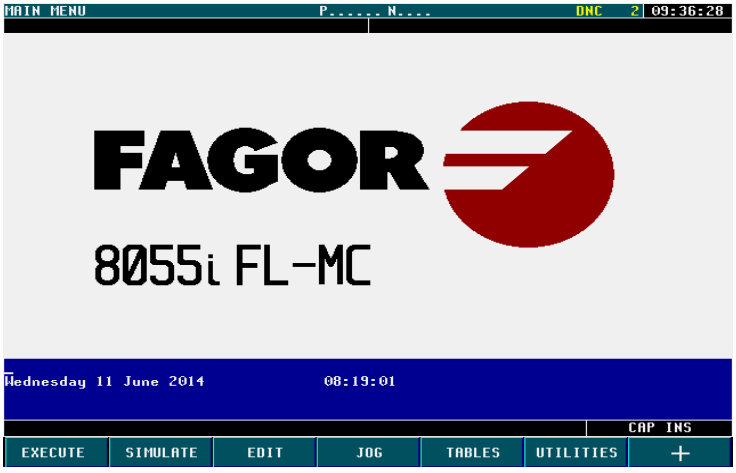
8055 Screen Modes

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  

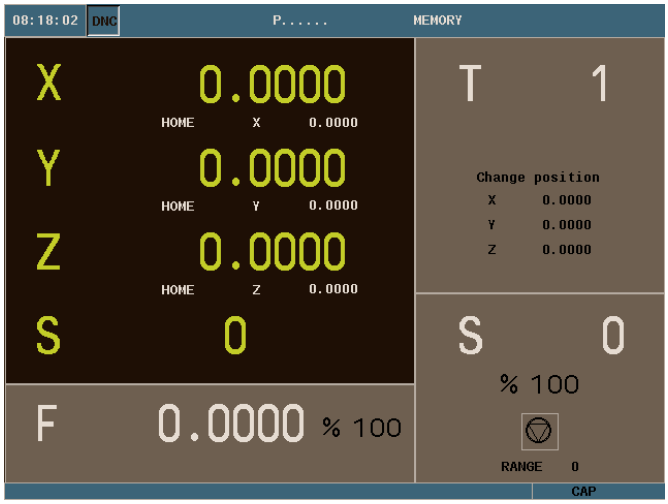


Conversational Side

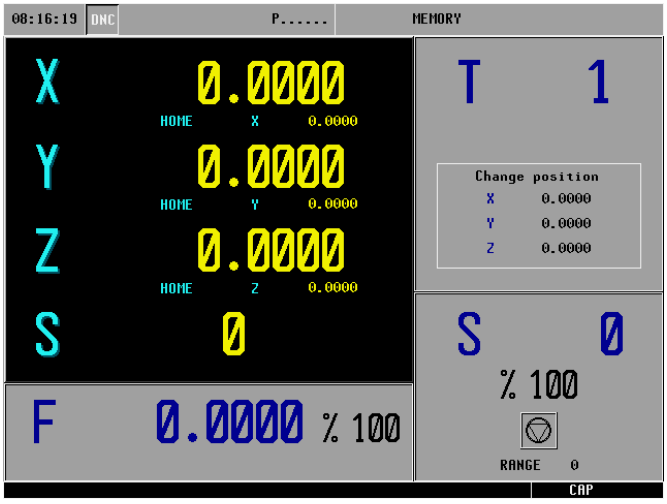


G-Code Side

New and Classic views are available by changing General Machine Parameter 193



General Machine Parameter 193 = 1 for new look.



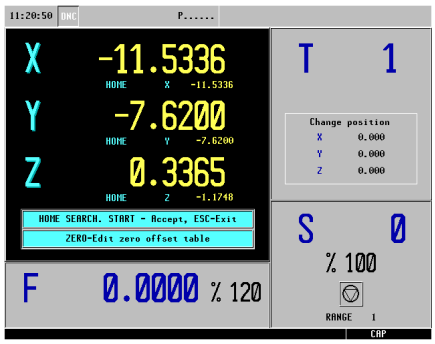
General Machine Parameter 193 = 0 for classic vivid look.

********All material from this manual here on out will be in the classic vivid look***

REFERENCE SEARCH

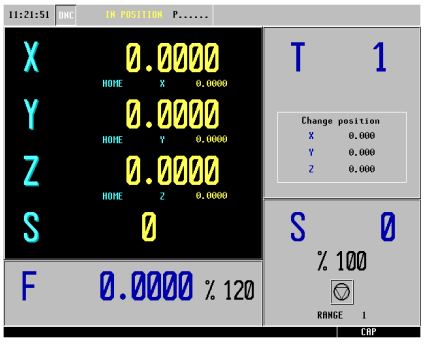
On power-up, it is always recommended to home your machine before performing any specific task. This is more of a safety precaution. When referencing your machine the CNC will know exactly where it is located. To reference your machine perform the following operation:

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*

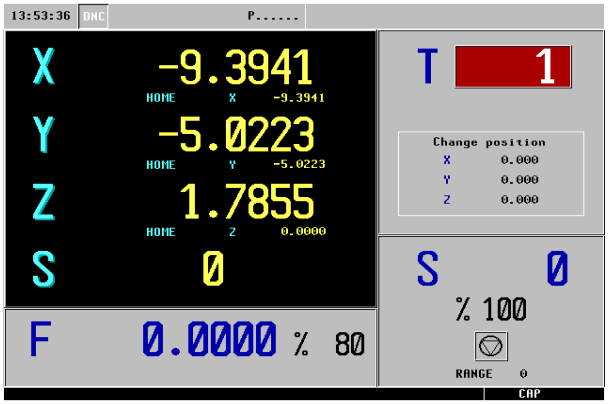
2. To execute the Home Search press cycle start 
   - At this time the CNC will execute the subroutine associated with the home search
   - The Z axis will be referenced first followed by the X and Y



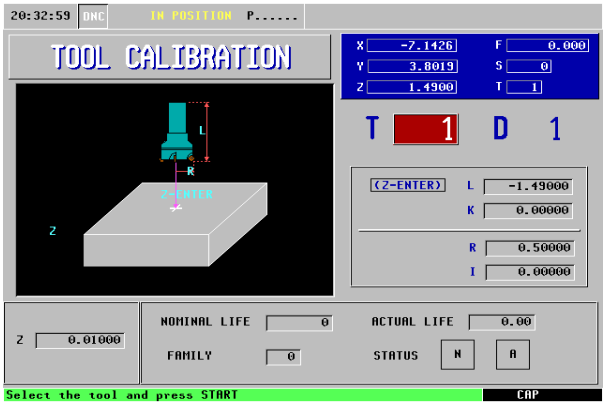
TOOL CALIBRATION

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:
   - Press "T" on the keyboard (the box will highlight in red)
   - 1 (Tool 1)
   - Cycle Start 
     ****Check Feed rate knob, otherwise machine will not move.



2. The axes will now move to its Tool Change Position.
   Insert tool
3. Press Cycle Start to complete tool change
4. Select the F1 Icon. 



5. Jog your Z axes accordingly until the tip of your tool is touching off on the top of your part. This can be done by using the hand wheel (if supplied) or the Z+ / Z- keys.
6. Enter the value of .02000 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
   See Figure1
7. Define your tool:
   - L- This will define the length of your tool from its home position. By selecting Z and then hitting the Enter key,
     Make sure the Z-ENTER box is highlighted in red when performing this operation. By pressing the Z key, then press ENTER to confirm. See Figure 2
     After pressing ENTER, you will notice in the top right corner of the screen Z will have updated value to the preset of .0100. See Figure 3
   - 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.
   - Press ESC when done

TOOLS 2 AND 3

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

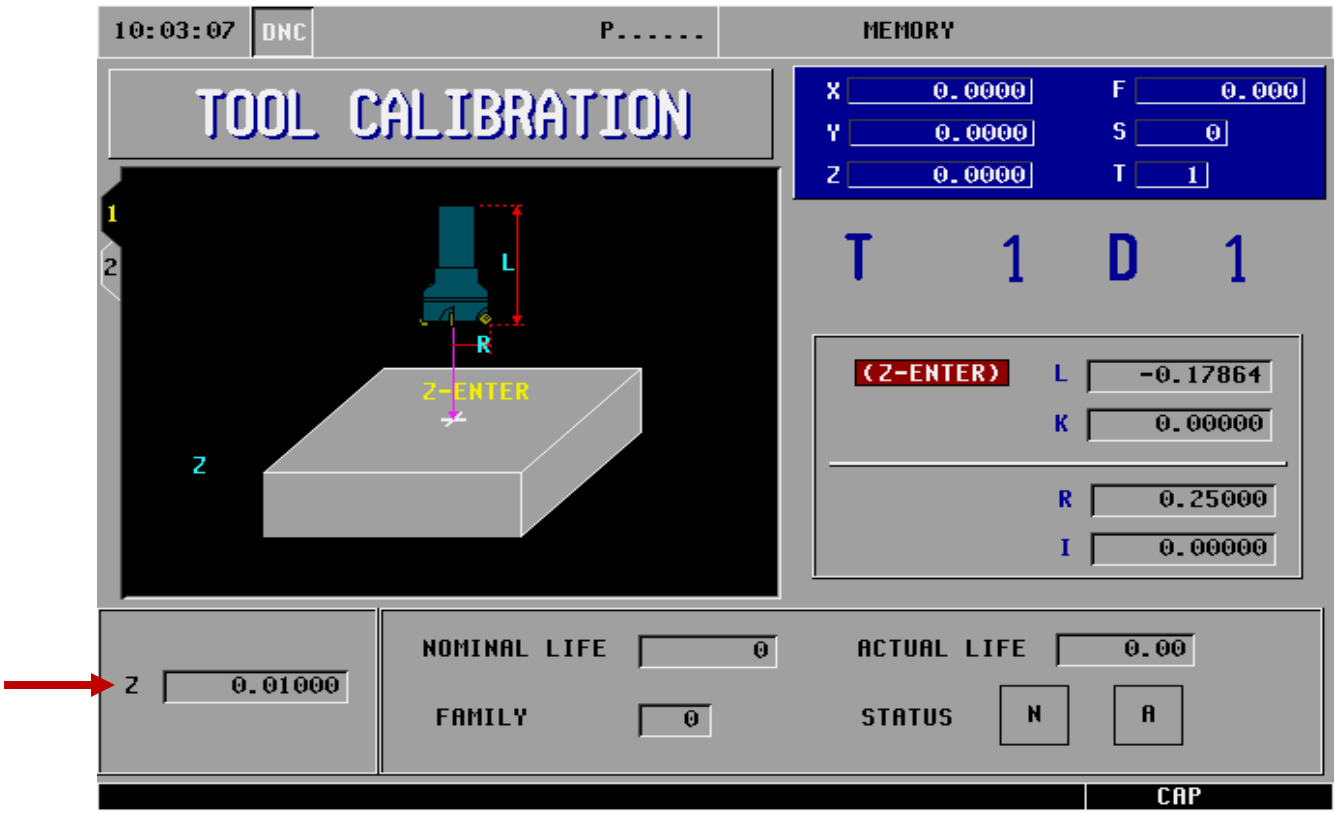


Figure 1

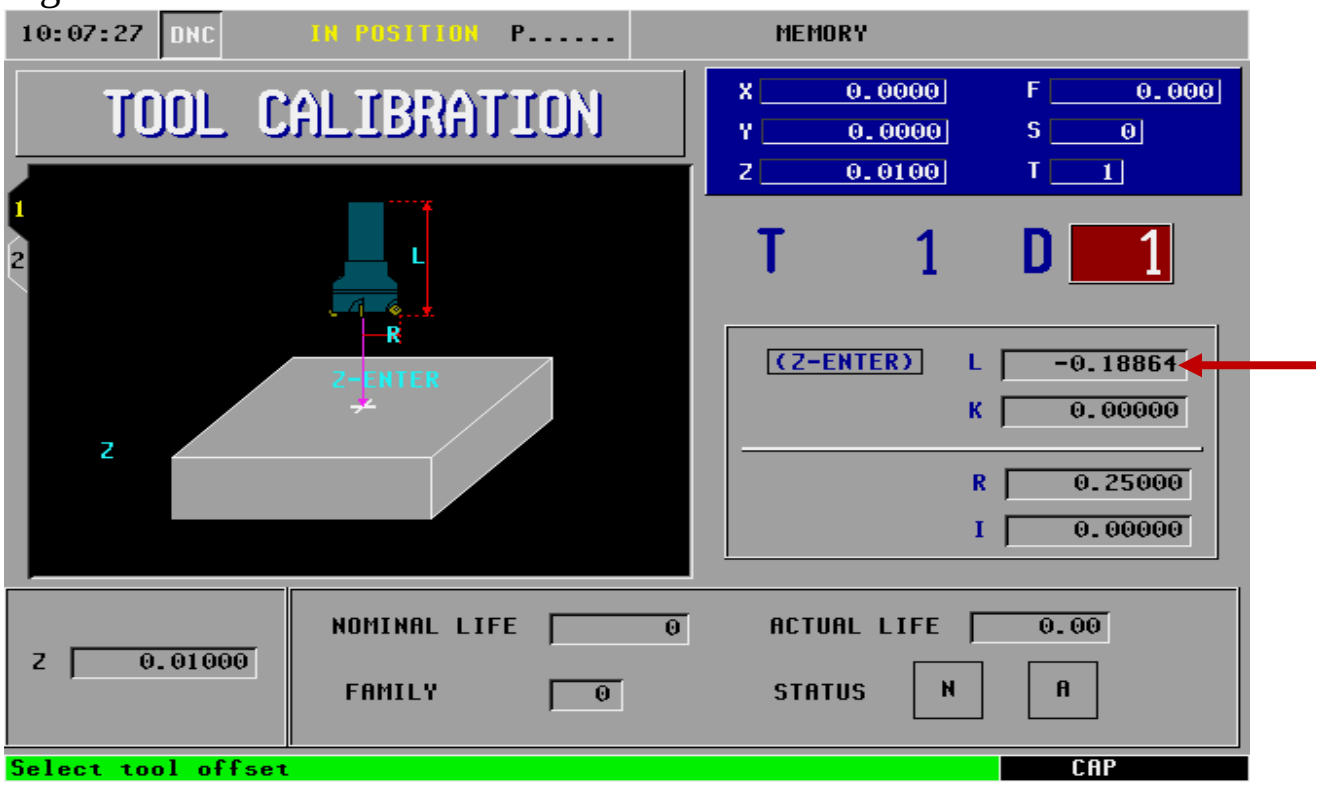
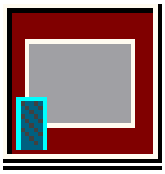


Figure 2

PRESETTING PART ZERO



In this example
***X 0 and Y 0** will be in the lower left hand corner. All numbers will be positive for programming based on the center of the tool.*

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:

    - Press X (select your axis) See Figure 1
    - The number 0 (your preset value), or the radius of the tool based on how you would like to program. The face of part will be 0.See Figure 2
    - Enter (confirming your position) See Figure 3
    - Enter (executing your command) See Figure 4

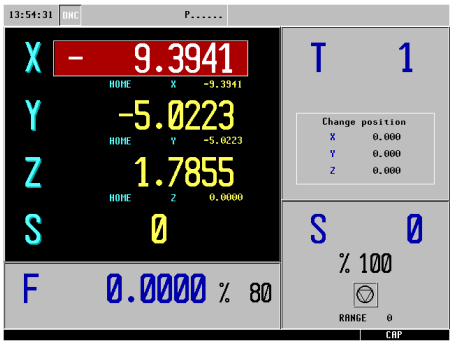


Figure 1

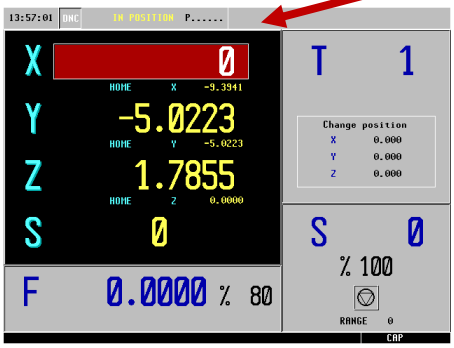


Figure 2

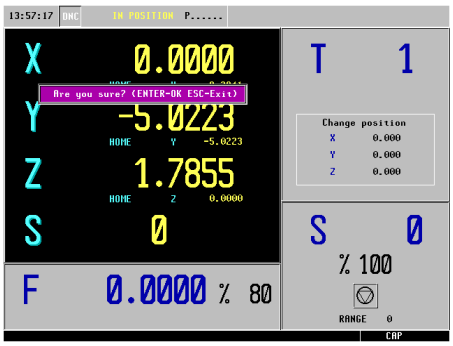


Figure 3

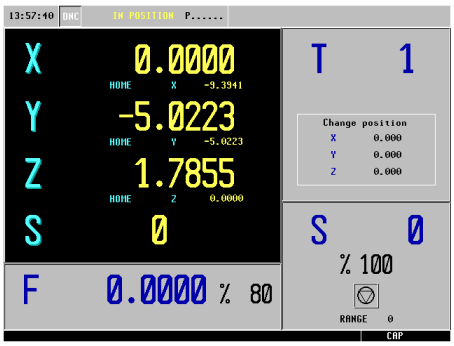
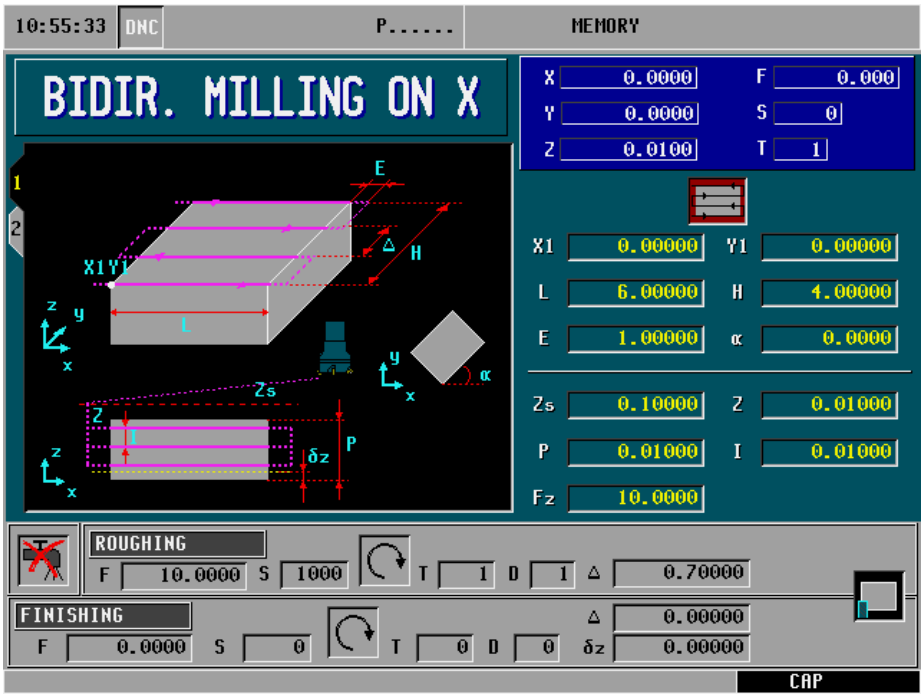


Figure 4

- *Repeat these steps for your Y and Z axis.*

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.

    Half-key

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. Press ENTER
   - L- This is the length of your part along the X axis. In this example we will be using the value of 6. Press ENTER
   - H- This is the height of your part along the Y axis. In this example we will be using the value of 4. Press ENTER
   - 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.
   - "α" - 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. Press ENTER

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 Press ENTER
    - Z- This is the starting point on the Z axis. In this example we will be using a value of .01 Press ENTER
    - 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 Press ENTER
    - 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. Press ENTER
    - Coolant ICON  You can turn coolant OFF or ON by using the  "half-key". In this example NO coolant is being used.

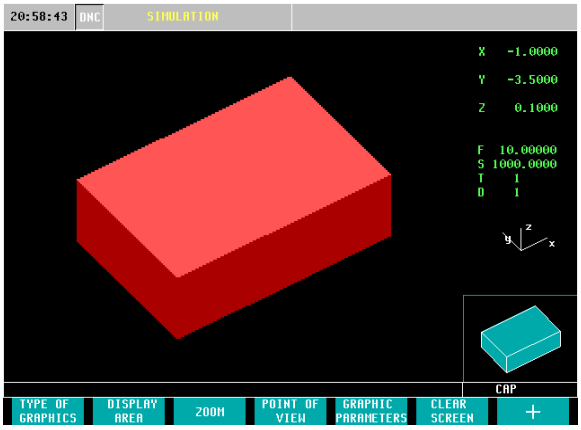
5. Define your Roughing Pass:
    - F- This is your roughing feedrate along the axis. In this example we are using a value of 10 inches per minute. Press ENTER
    - S- This is your spindle speed. In this example we are using a value of 1000 rpm. Press ENTER
    - Select your spindle direction.  To toggle between directions push the  "half-key". In this example we will be using a Clockwise Rotation.
    - T- Enter 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. In this example enter we will be using ¾ diameter of the tool, .75 Press ENTER.
    - 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.  To change location of the starting point use the arrow keys to navigate the icon.

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.



-Press the down arrow after entering numbers in boxes.

X Min: 0

X Max: 6

Y Min: 0

Y Max: 4

Z Min: -1.0

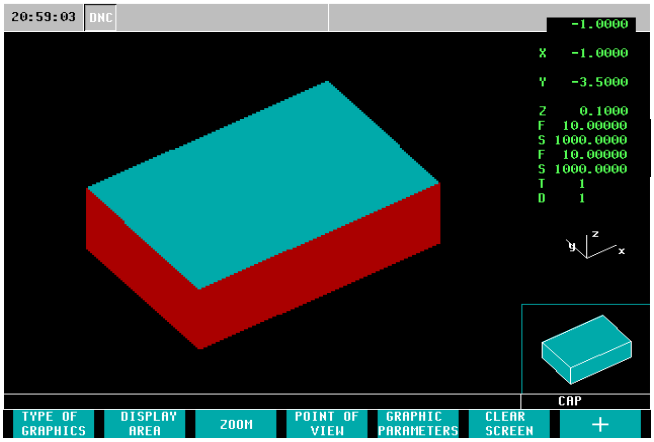
Z Max: 0.01

-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).

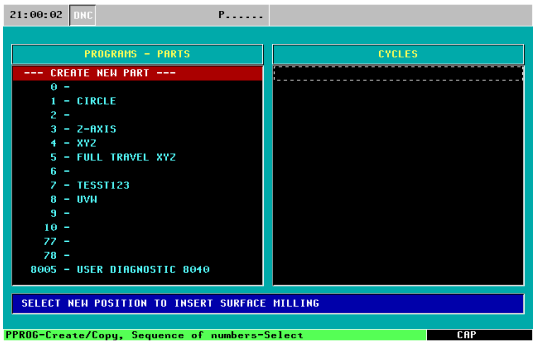
7. You are now ready to save this cycle to your Part-Program.



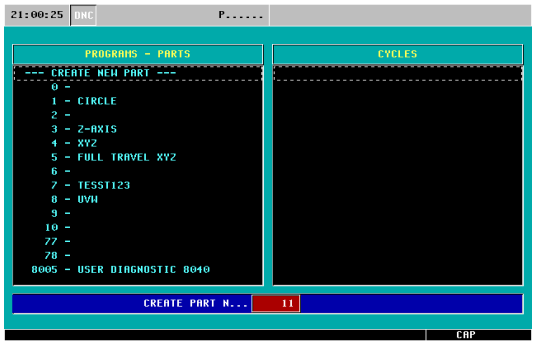
Storing Part Programs

1. Select The Part-Program Key. 

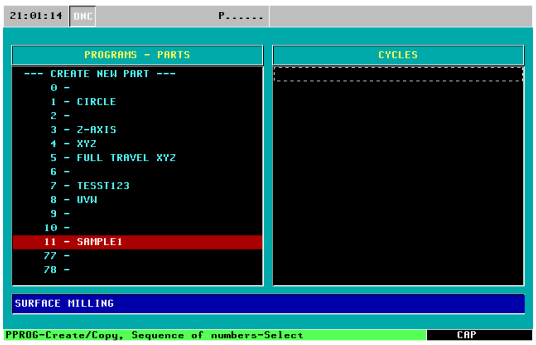
-Hit this button twice
   - *The following screen will appear*



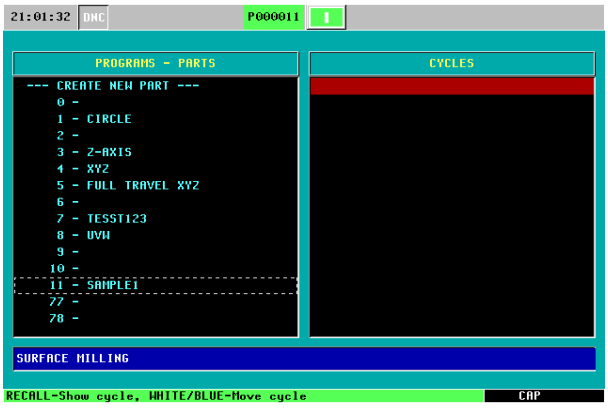
2. Select the Part-Program Key once more and enter a Part-Program Number. Hit enter when you have entered in your number. In this example we will us 11.
   - *The CNC will call up the first available number*



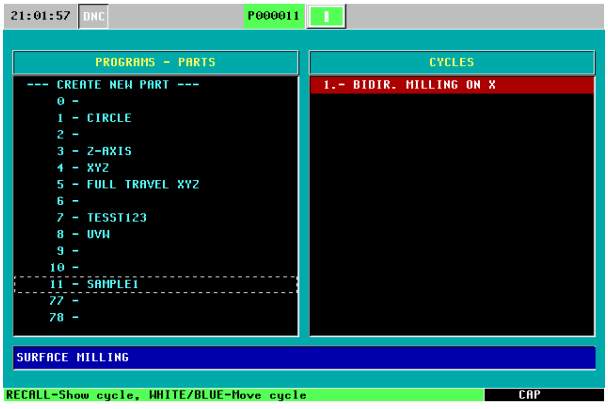
3. Enter a comment for your part-program and hit Enter. In this example we are using SAMPLE1.



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

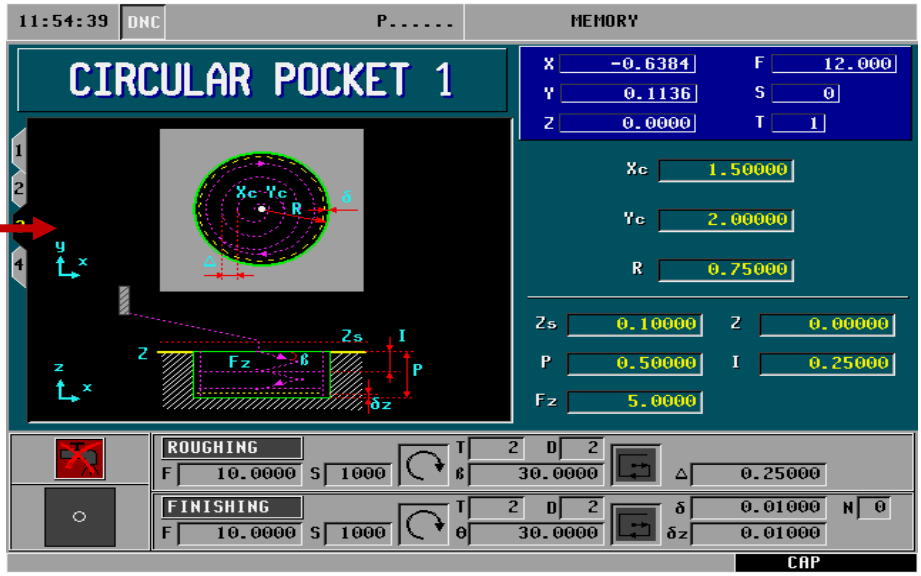


5. Once this section is highlighted in red you may now hit enter and your cycle is saved



CIRCULAR POCKET

Level Cycle 3



1. Select the F7 Icon. 
2. Press the Level Cycle key. 

Press this key until you are in Level Cycle 3 (Circular Pocket 1). See Figure above
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. Press ENTER
    - 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. Press ENTER
    - 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 0.1. Press ENTER
    - Z- This is the starting point on the Z axis. In this example we will be using a value of 0. Press ENTER
    - P- This is the total depth of the pocket. In this example we will be using a value of 0.5. Press ENTER

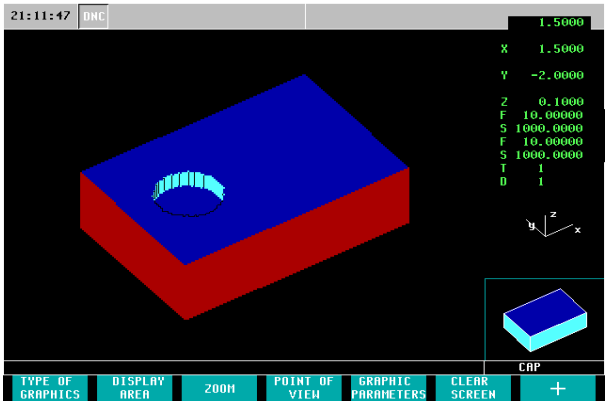
- I- This is the amount we will be removing depth wise per pass. In this example we will be removing a 0.25” of material per pass meaning it will take two passes until we get to our final depth. Press ENTER
- Fz- This is the penetration feedrate. This value is entered in inches per minute. In this example we will be using a value of 5. Press ENTER
- Coolant ICON  You can turn coolant OFF or ON by using the  ”half-key” In this example NO coolant will be used.
-

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. Press ENTER
    - S- This is your spindle speed. In this example we are using a value of 1000 rpm.
      Press ENTER
    - Select your spindle direction.  To toggle between directions push the  “half-key”. In this example we will be using a Clockwise Rotation.
    - T- Enter your tool number. In this example we are using T2.
    - D- Enter in your Tool Offset Number. In this example we are using D2.
    - (β) This is your Sideways Penetration Angle. In this example we will penetrate into the center of our pocket on a 30 Degree Angle. Press ENTER
    -  This is the type of milling (Climb/Conventional Milling).  Use “half-key” to toggle
    - (Δ) This is your step-over. In this example we will be using a value of .25 Press ENTER

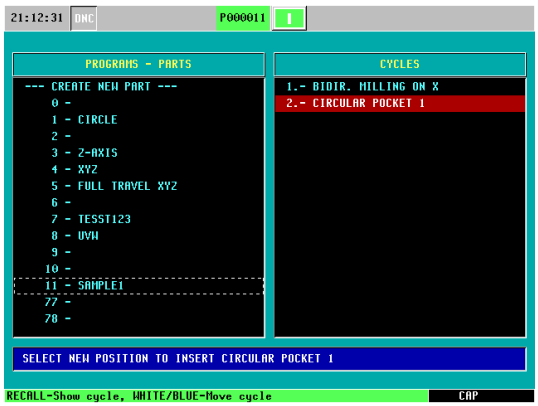
6. Define your Finishing Pass:
    - F- This is your finishing feedrate. In this example we will be using a value of 10 inches per minute. Press ENTER
    - S- This is your spindle speed. In this example we are using a value of 1000 rpm. Press ENTER
    - Select your spindle direction.  To toggle between directions push the  “half-key”. In this example we will be using a Clockwise Rotation.
    - T- Enter your tool number. In this example we are using T2.
    - D- Enter in your Tool Offset Number. In this example we are using D2.

- (β) This is your Sideways Penetration Angle. In this example we will penetrate into the center of our pocket on a 30 Degree Angle. Press ENTER
- This is the type of milling (Climb/Conventional Milling). Use "half-key" to toggle
- δ This is the Finishing Pass. In this example we will be leaving 0.01 for the finishing pass. Press ENTER
- (N) This is the Number of Finishing Passes in Z. If 0 is entered, the CNC will calculate the number of passes needed. δZ This is the Finishing Pass in Z. In this example we will be leaving 0.01 for the finishing in Z. Press ENTER

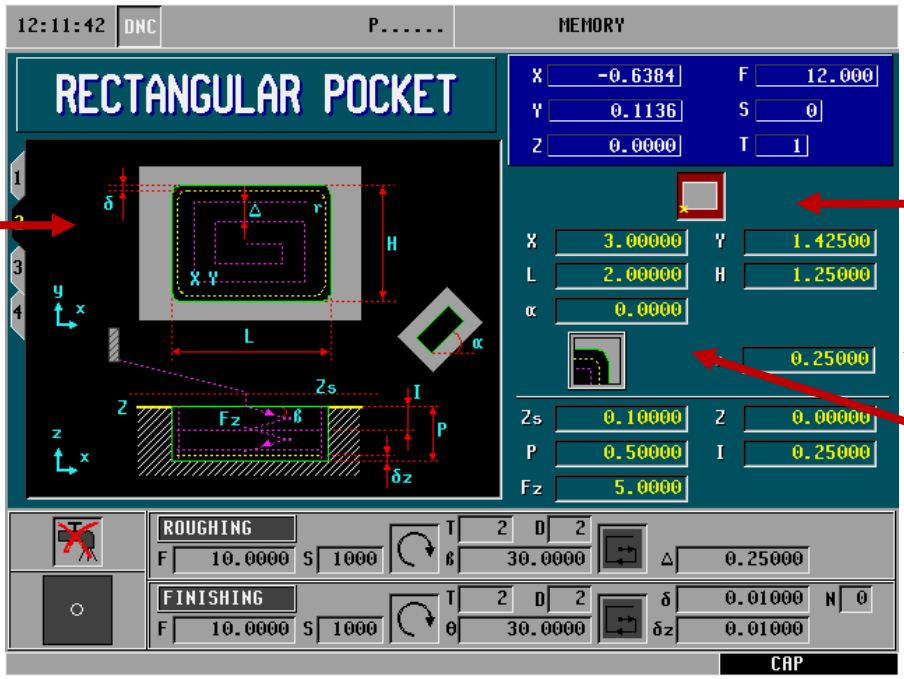
7. Simulate the program: Press Press Cycle Start.



8. Save your part-program. Press -Hit this button twice, then press ENTER to save.



RECTANGULAR POCKET



Level Cycle 2

Select corner

Radius

Type of corner

1. Select the F7 Icon. 

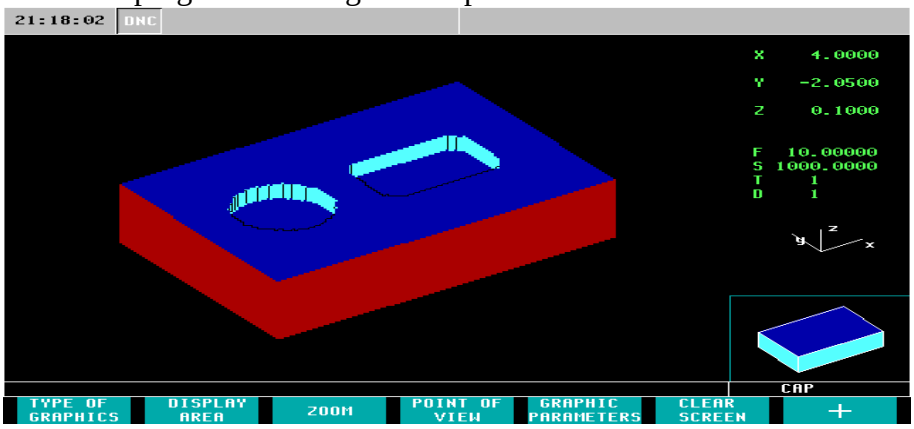
2. Press the Level Cycle key.  Press this key until you are in Level Cycle 2 (Rectangular Pocket). See above

   - Define the starting location:  It can be the center or corner of the part. In this example we will be using the corner.
   - X- This is the starting point of your pocket along the X Axis. In this example we will be using a value of 3. Press ENTER
   - Y- This is the starting point of your pocket along the Y Axis. In this example we will be using a value of 1.425. Press ENTER
   - L- This is the total length of the pocket along the X Axis. In this example we will be using a value of 2. Press ENTER
   - 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. Press ENTER

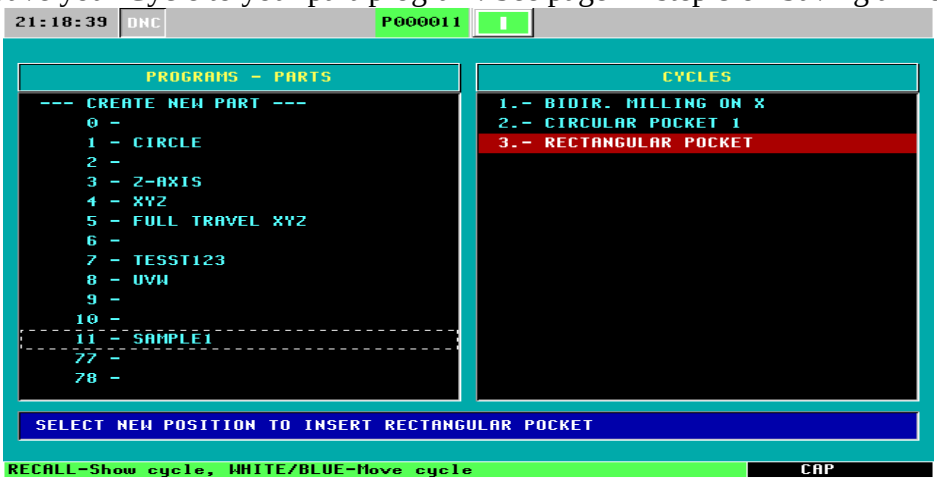
- "α" - The value in this field must be entered in degrees. A value is entered into this field when we are machining a part on an inclination angle. In this example we will be using a value of 0. Press ENTER
- r – In this cycle we are able to define the radius of the corners. You have the options of 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. Press ENTER

3. 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". Press ENTER
   - 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. Press ENTER
   - 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. Press ENTER
4. 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. Press ENTER
   - S- This is your spindle speed. In this example we are using a value of 1000 rpm. Press ENTER
   - Select your spindle direction.  To toggle between directions push the  "half-key". In this example we will be using a Clockwise Rotation.
   - T- Enter your tool number. In this example we are using T2.
   - D- Enter in your Tool Offset Number. In this example we are using D2.
   - (β) This is your Sideways Penetration Angle. In this example we will penetrate into the center of our pocket on a 30 Degree Angle. Press ENTER
   -  This is the type of milling (Climb/Conventional Milling).
   - (Δ) This is your step-over. In this example we will be using a value of .4
   - Coolant ICON  You can turn coolant OFF or ON by using the  "half-key". In this example NO coolant will be used.

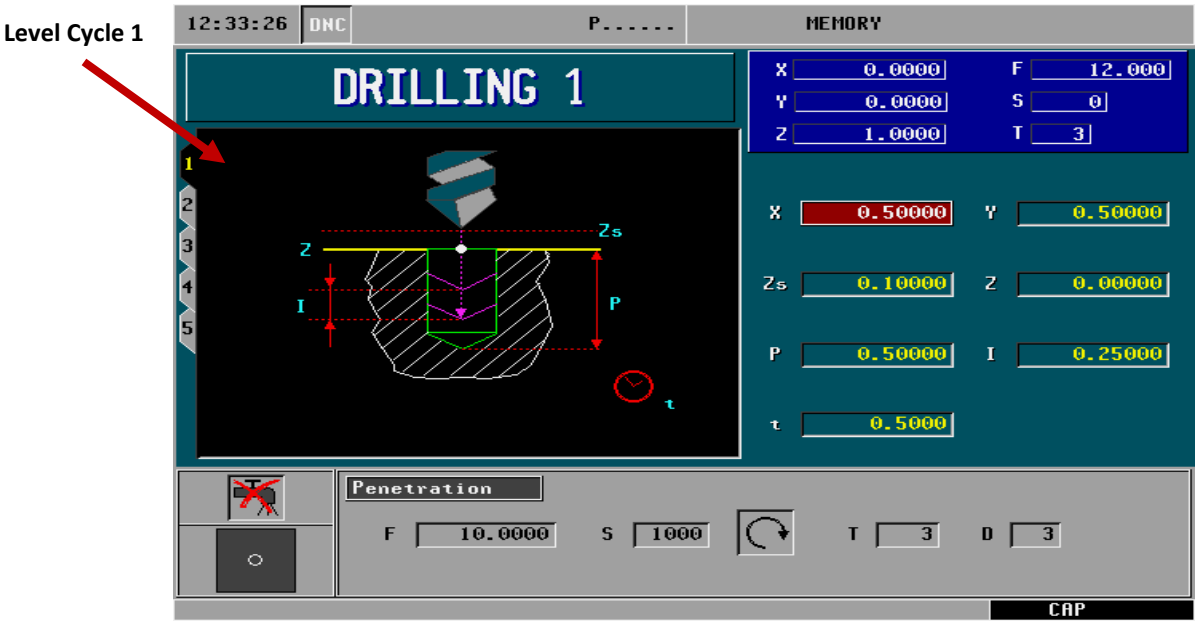
5. Define your Finishing Pass:
    - F- This is your finishing feedrate. In this example we will be using a value of 10 inches per minute. Press ENTER
    - S- This is your spindle speed. In this example we are using a value of 1000 rpm. Press ENTER
    - Select your spindle direction.  To toggle between directions push the  "half-key". In this example we will be using a Clockwise Rotation.
    - T- Enter your tool number. In this example we are using T2.
    - D- Enter in your Tool Offset Number. In this example we are using D2.
    - (β) This is your Sideways Penetration Angle. In this example we will penetrate into the center of our pocket on a 30 Degree Angle. Press ENTER
    -  This is the type of milling (Climb/Conventional Milling).
    - δ This is the Finishing Pass. In this example we will be leaving 0.010" for the finishing pass. Press ENTER
    - (N) This is the Number of Finishing Passes in Z. In this example we will be using only 1 finishing pass. Press ENTER
    - δz This is the Finishing Pass in Z. In this example we will be leaving 0.01 for the finishing in Z.
6. Simulate the program: See Page 21 Step 7 on Simulation



7. Save your Cycle to your part-program: See page 21 step 8 on Saving a Program



DRILLING + RECTANGULAR POSIT.

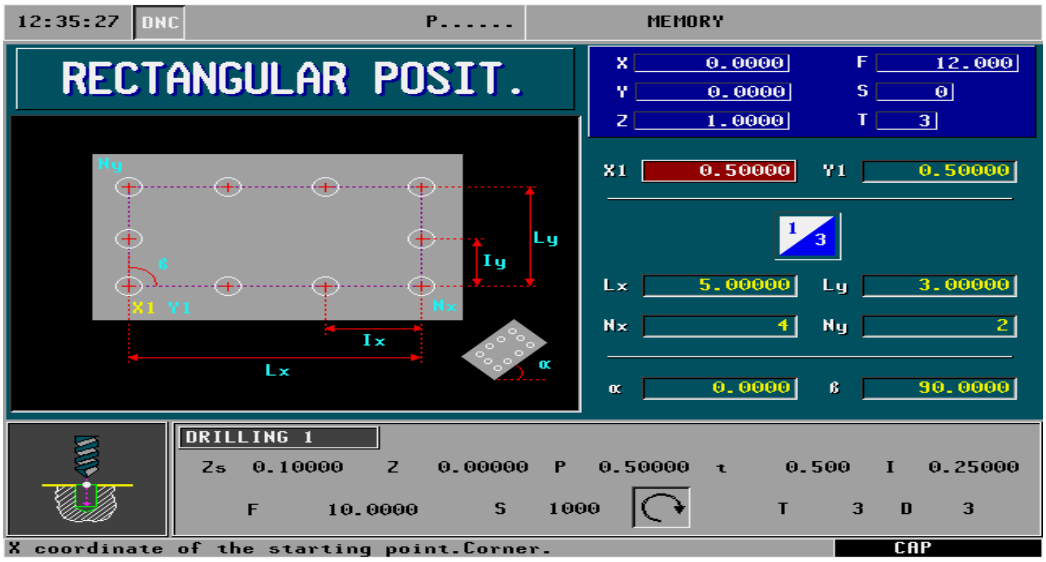


1. Select the Drilling Icon 
2. Be sure to have Drilling 1 (Level Cycle 1) selected.
3. Define the position of your first hole:
    - X- This is the position of your first hole along the X Axis. In this example we will be using a value of 0.500”. Press ENTER
    - Y- This is the position of your first hole along the Y Axis. In this example we will be using a value of -0.500”. Press ENTER
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”. Press ENTER
    - Z- This is the starting point on the Z axis. In this example we will be using a value of 0. Press ENTER
    - P- This is the total depth of the hole. In this example we will be using a value of 0.500”. Press ENTER
    - 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. Press ENTER
- Coolant ICON  can turn coolant OFF or ON by using the  ”half-key”. In this example NO coolant will be used.

5. Define your Penetration:
   - F- This is your feedrate. In this example we will be using a value of 10 inches per minute. Press ENTER
   - S- This is your spindle speed. In this example we are using a value of 1000 rpm. Press ENTER
   - 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  Located on the left of the CNC screen.



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

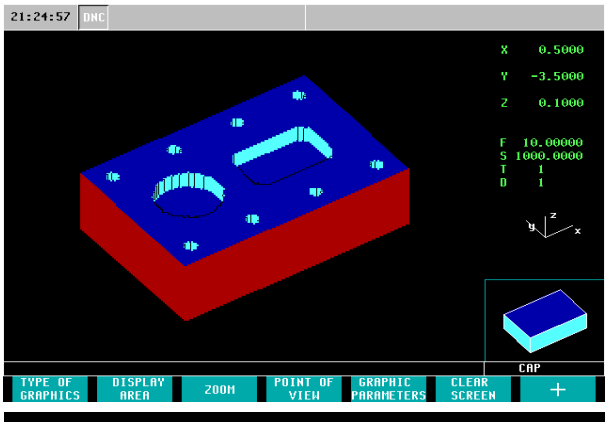
There are 3 different options of entering position locations.

This is example is 1 of 3.  Highlight box and use half key  to toggle.

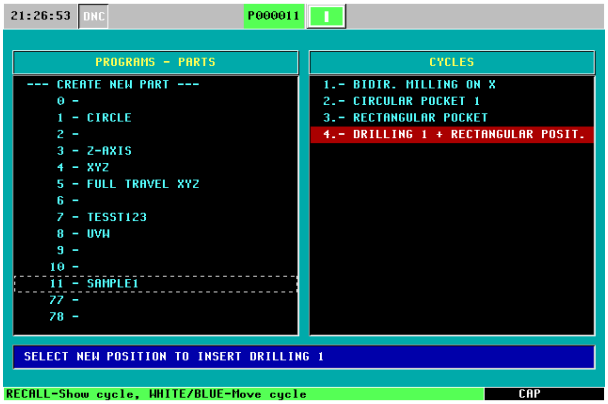
7. Define the length and number of hole along each axis:

- 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. Press ENTER
- 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. Press ENTER
- Nx- This is your total number of holes along the X Axis. In this example we will be using a value of 4. Press ENTER
- Ny- This is your total number of holes along the Y Axis. In this example we will be using a value of 2. Press ENTER
- "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. Press ENTER
- (β)This is the angle between the sides. In this example we will be using a value of 90 degrees Press ENTER

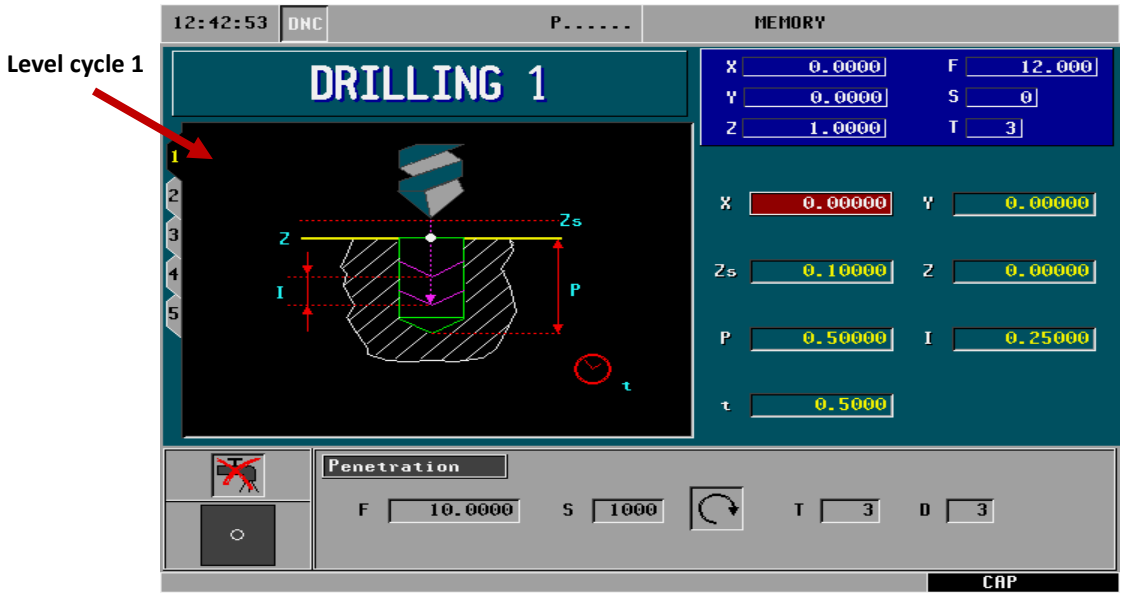
8. Simulate the program: See Page 21 on Simulation



9. Save your Cycle to your part-program: See page 21 on Saving a Program



DRILLING+ARC POSITIONING

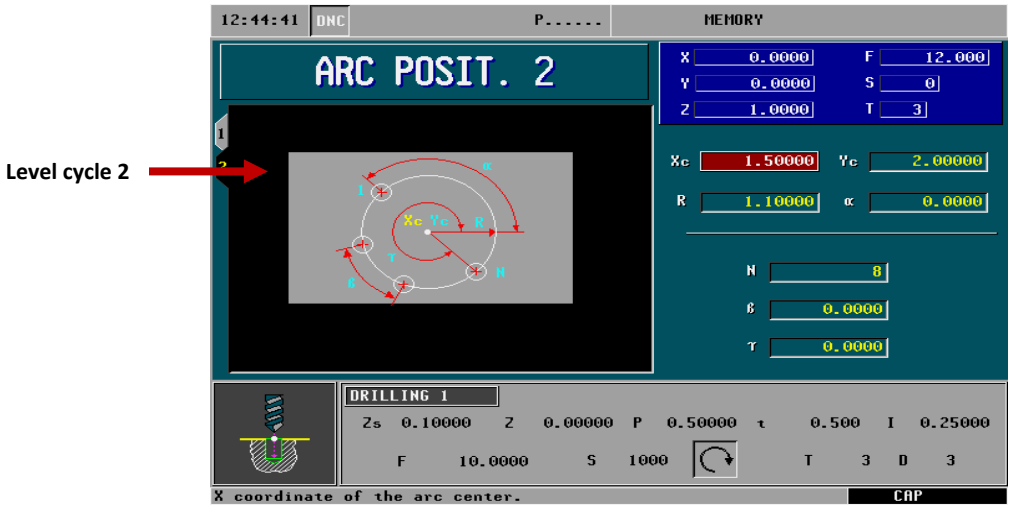


1. Select the Drilling Icon 

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. Press ENTER
    - Y- This is the position of your first hole along the Y Axis. In this example we will be using a value of 0. Press ENTER
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". Press ENTER
    - Z- This is the starting point on the Z axis. In this example we will be using a value of 0 Press ENTER
    - P- This is the total depth of the hole. In this example we will be using a value of 0.500" Press ENTER
    - I- This is the drilling peck. In this example we will be drilling down .25 per pass. Press ENTER

- t- This is the dwell time (seconds). In this example there will be a 0.5 second dwell time per pass. Press ENTER
- Coolant ICON  can turn coolant OFF or ON by using the  ”half-key” In this example NO coolant will be used.

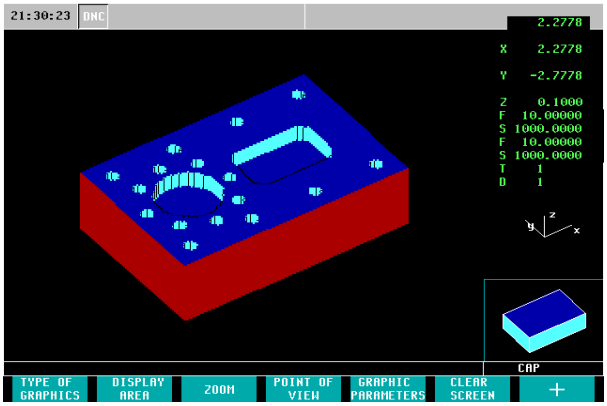
5. Define your Penetration:
   - F- This is your feedrate. In this example we will be using a value of 10 inches per minute. Press ENTER
   - S- This is your spindle speed. In this example we are using a value of 1000 rpm. Press ENTER
   - Select your spindle direction.  To toggle between directions 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. Press ENTER
   - D- Enter in your Tool Offset Number. In this example we are using D3.
6. Select the Arc Positioning Icon  Located on left of CNC screen



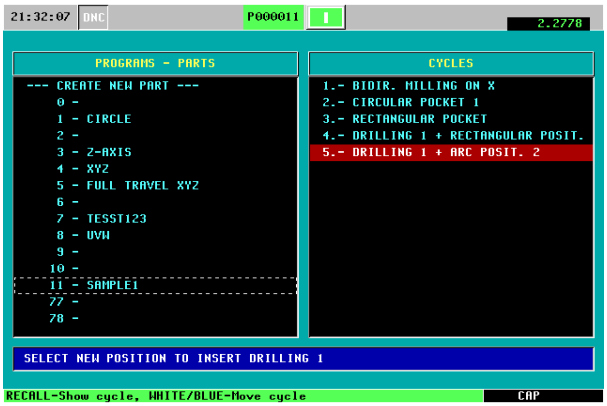
7. Go into Level Cycle 2 (ARC POSIT. 2) 
8. Define the Center Coordinates and Machining Conditions:
   - Xc- This is the center point of your bolt-hole position on the X Axis. In this example we will be using a value of 1.5 Press ENTER
   - Yc- This is the center point of your bolt-hole position on the Y Axis. In this example we will be using the value of 2 Press ENTER
   - r- This is the radius of your bolt-hole pattern. In this example we will be using a value of 1.1 Press ENTER

- "α" - 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. Press ENTER
- N- This is the total number of holes in the arc. In this example we will be using a value of 8. Press ENTER
- (β) This is the angular distance between points. In this example we will be using a value of 0 Press ENTER
- (τ) This is the angle of the end point. In this example we will be using a value of 0 Press ENTER

9. Simulate the program: See Page 21 on Simulation



10. Save your Cycle to your part-program See page 21 on Saving a Program

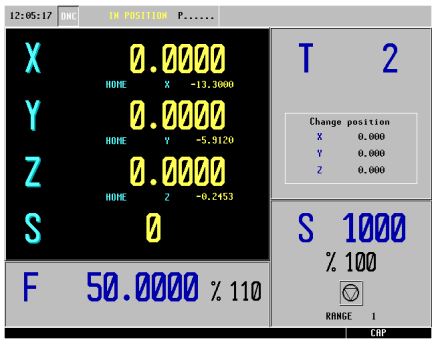


You have now created and simulated your part-program and are ready for execution.

ZERO OFFSETS

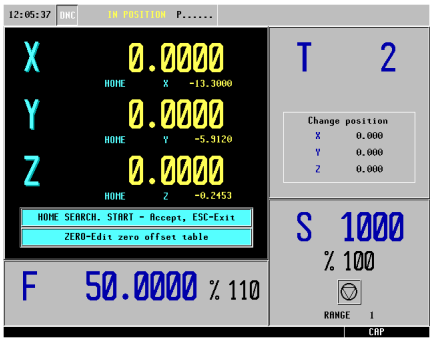
G54 through G57 are our absolute zero offsets. This feature allows us to store a part zero position into a table. With this option you are able to recall your part zero position without having to touch off on your part again. This feature is useful when having parts that are commonly machined. To setup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

1. From the Jog Screen setup your part zero position

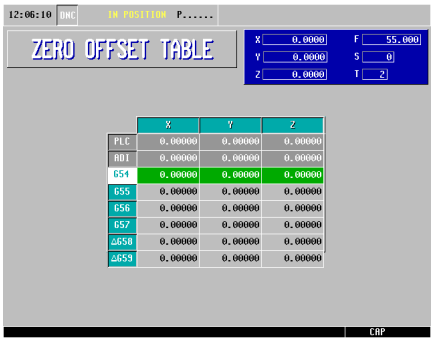


**Note: Refer to page 12 of the manual on how to preset your part zero*

2. Select the Zero Icon 

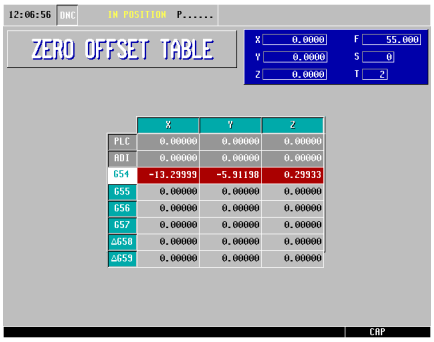


3. Select the Zero Icon  for a second time to enter the Zero Offset Table



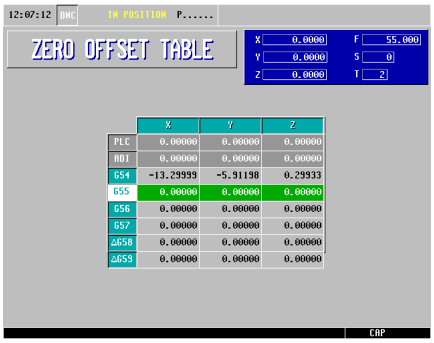
** Note: All of the coordinate values for G54 are currently at zero*

4. Select the Recall Key 



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

5. Press Enter



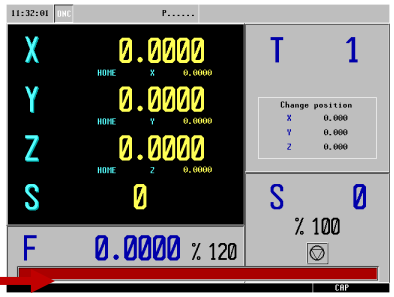
**Note: Your values will not be stored if you do not hit enter*

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:

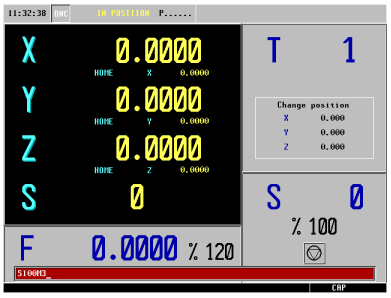
1. From the Jog Screen select the ISO key 

- 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 See below

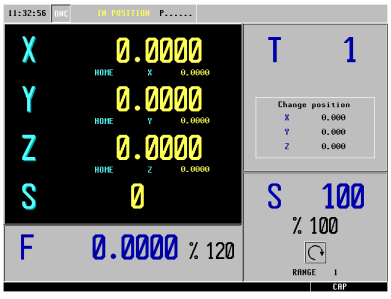


Dialog Box

2. Type in S100 M3



3. Press Cycle Start 



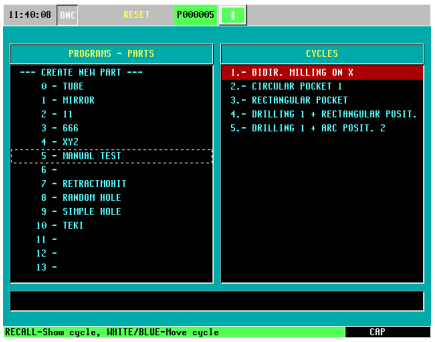
If need be, change to correct gear. Press enter to confirm once gear has been changed.

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

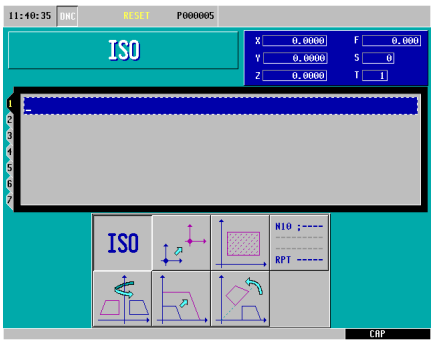
INSERTING CODE IN-BETWEEN CANNED CYCLES

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.

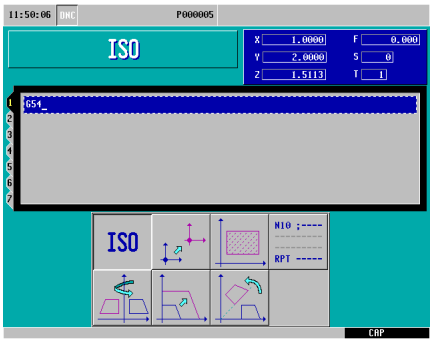
1. From the Jog Screen select the P.PROG key 



2. Select the ISO key 

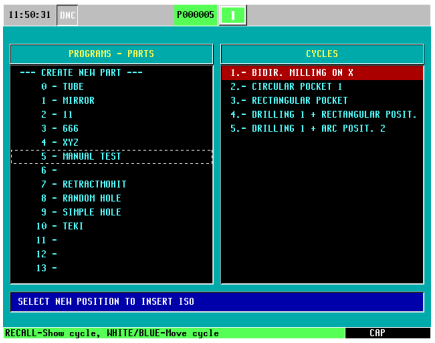


3. Type in G54



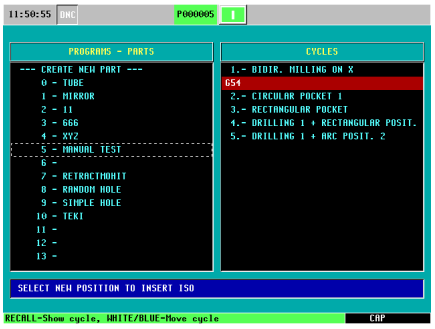
4. Select the P.PROG Key 

Select program where you would like to insert G54.

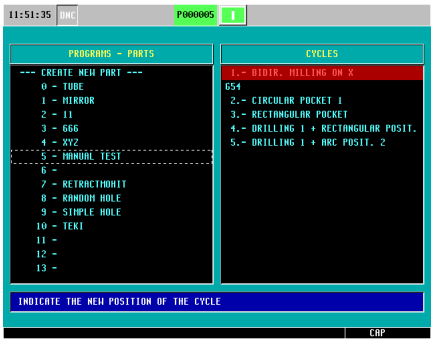


**Note: Make sure our first programmed cycle is highlighted in red before hitting enter.*

5. Press Enter

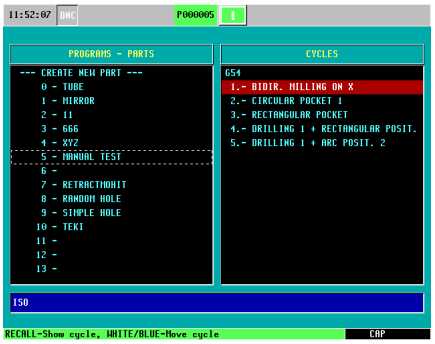


6. Highlight our first programmed cycle and select the Half-Key 

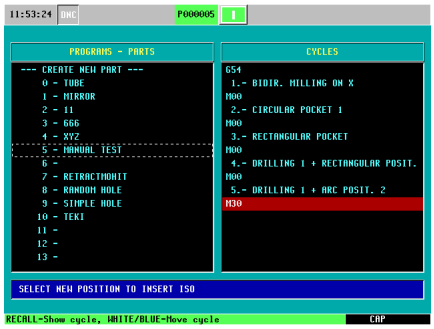


**Note: This allows us to insert our first programmed cycle underneath the G54.*

7. Arrow down to G54 and hit enter.



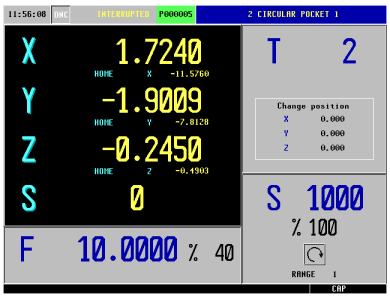
8. Now to insert M00 execute the following commands: (M00 is a conditional stop command)
    - Select the ISO Key
    - Clear out the G54
    - Type in M00
    - Select the P.PROG Key
    - Highlight the first Canned Cycle (BIDR. MILLING ON X)
    - Press Enter
    - Now you can highlight the remaining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



TOOL INSPECTION

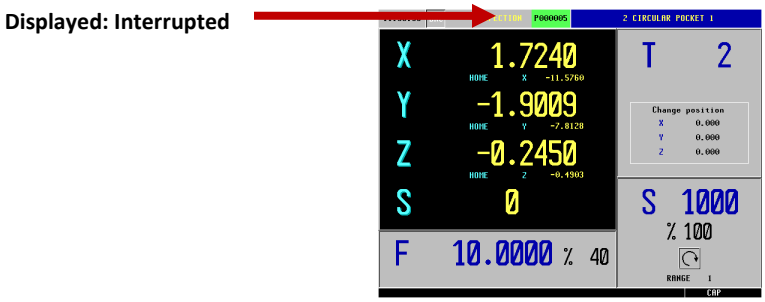
This feature allows us to examine our tool at any point while executing our program. Whether the tool breaks or if you are checking the status, this feature will allow you to stop your program, move your axes around and begin your program from the point you have left off at.

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



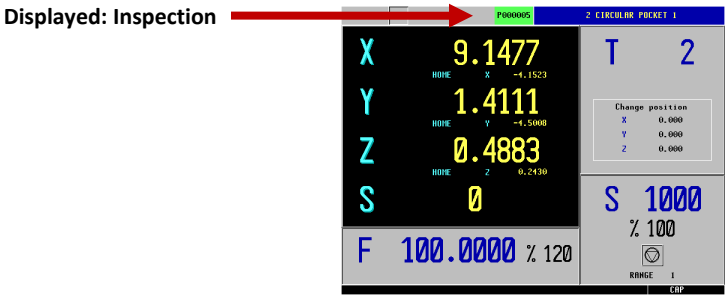
**Note: You will notice on the top of the screen it will read INTERRUPTED*

2. Press T (this is for Tool Inspection)



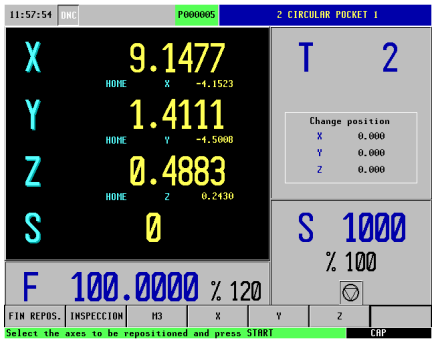
**Note: Notice the top of your screen now reads INSPECTION*

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



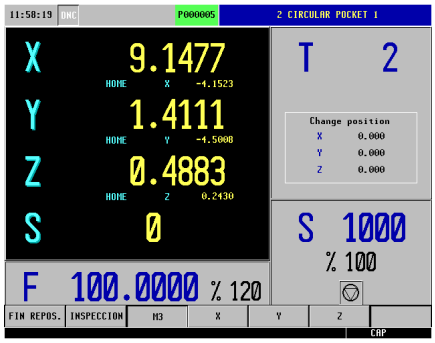
**Note: Notice the coordinate values are different from when we stopped the program*

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

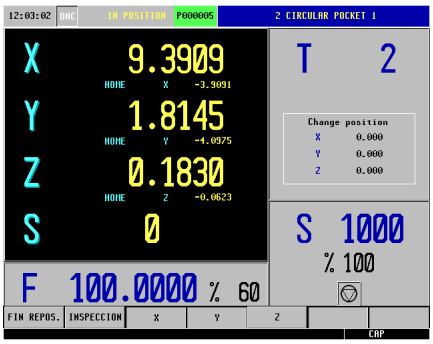


**Note: There are software keys to reposition the Axis*

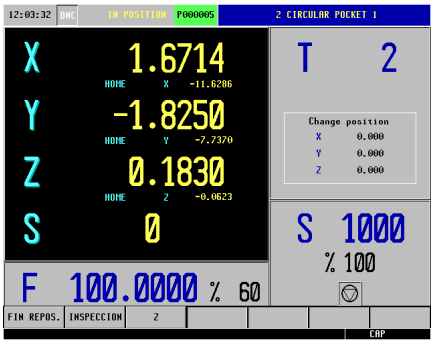
5. Select M3, which is the F3 key, then press Cycle Start to start your spindle



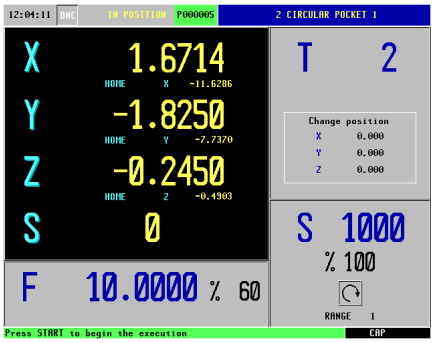
6. Select X and Y and then Cycle Start so these two axes are repositioned first



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



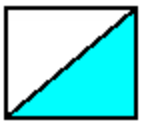
8. Now press Cycle Start to begin the execution of your program



QUICK REFERENCE GUIDE (MC)

F1: TOOL CALIBRATION

F3: PROFILING CYCLE



Used to change square symbols.

F4: SURFACE MILLING

F5: 2-D POCKET

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

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

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

(Used to switch from conversational to g-code)

POSITIONING CYCLE

DRILLING

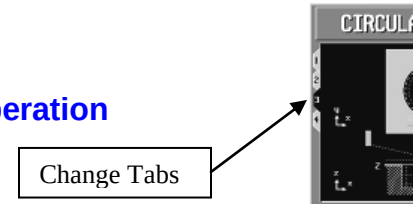
TAPPING

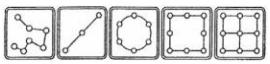
REAMING

BORING

Level cycle to select next page in an operation

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

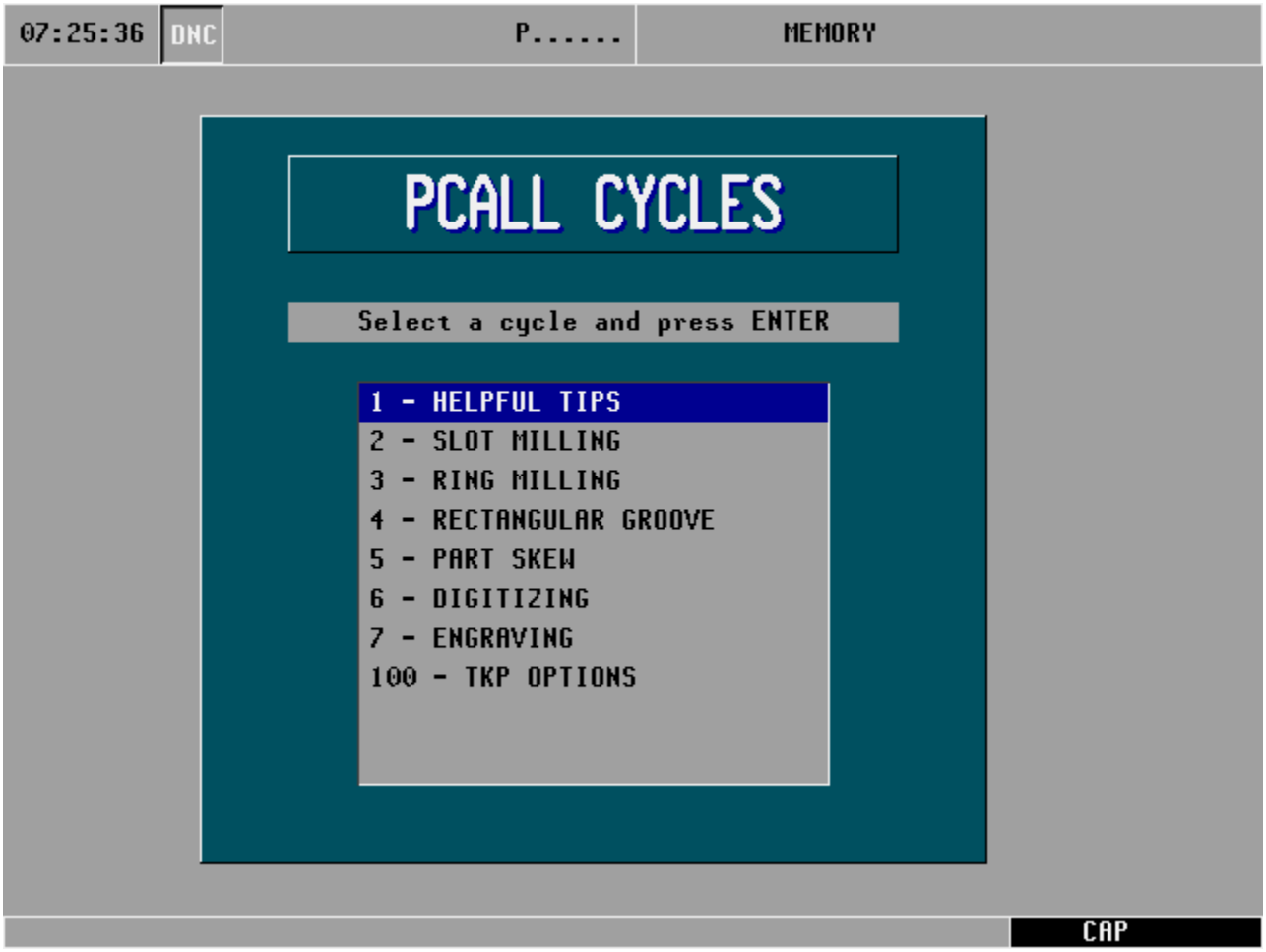
F	FEEDRATE
S	SPINDLE RPM
	SPINDLE DIRECTION (CLOCKWISE / COUNTER CLOCKWISE)
T	TOOL NUMBER
D	OFFSET NUMBER
b	SIDEWAYS PENETRATION ANGLE
	CLIMB / CONVENTIONAL MILLING
D	ROUGHING PASS

FINISHING

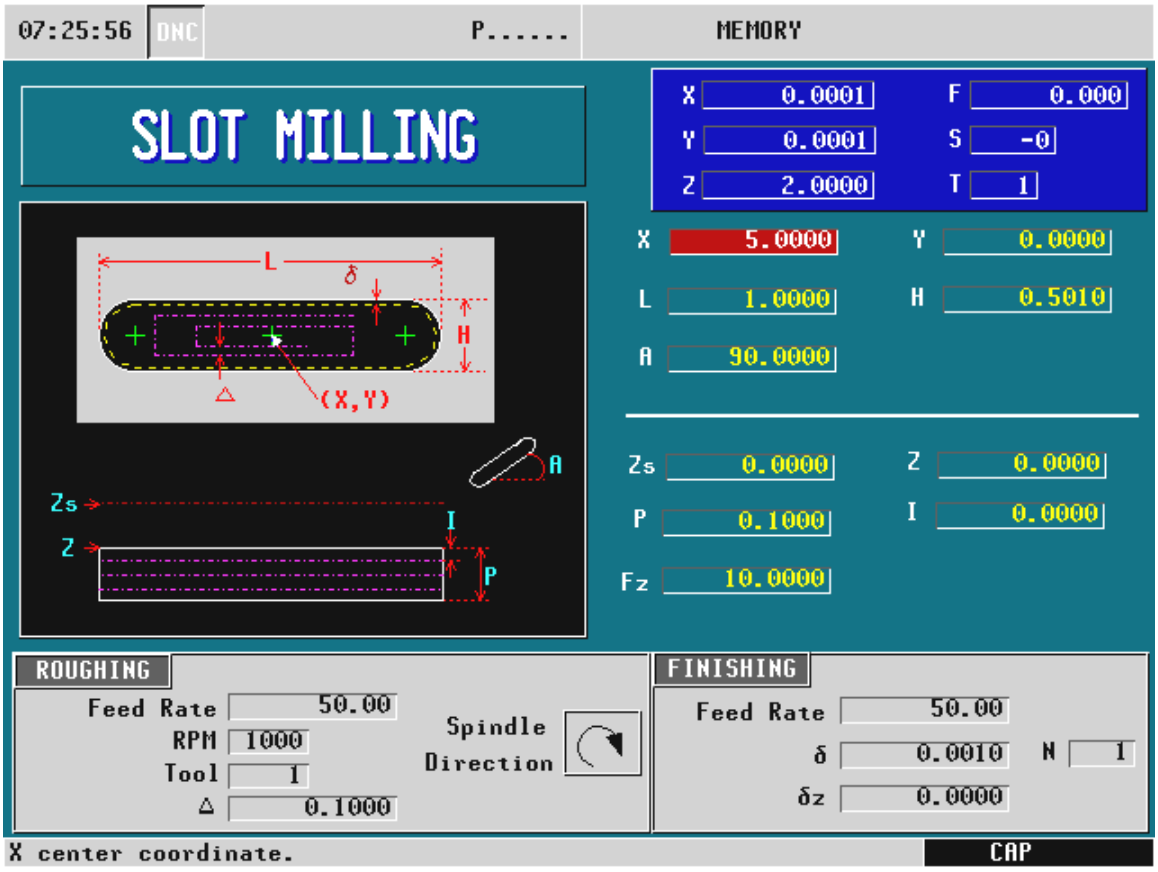
F	FEEDRATE
S	SPINDLE RPM
	SPINDLE DIRECTION (CLOCKWISE / COUNTER CLOCKWISE)
T	TOOL NUMBER
D	OFFSET NUMBER
Q	SIDEWAYS PENETRATION ANGLE
	CLIMB / CONVENTIONAL MILLING
d	FINISHING PASS
N	NUMBER OF FINISHING PASS
dz	FINISHING PASS IN Z

Optional PCALL CYCLES

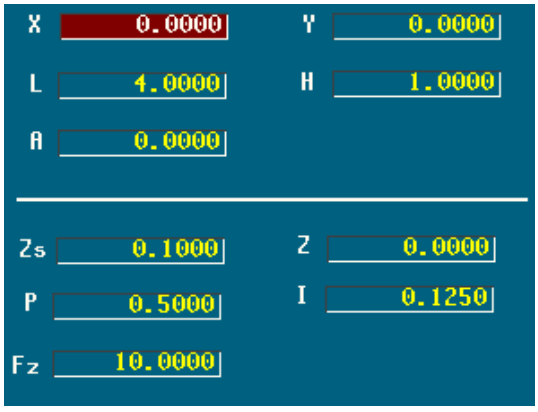
Note: These options only available for the 8055 Turnkey models sold by Fagor Automation USA & Canada



» *Slot Milling*



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.

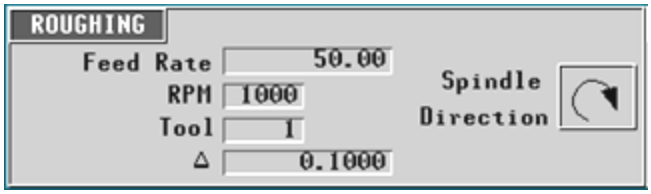
P - Slot depth.

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

Fz - Feed rate for Z axis

`

Roughing setup:



Feed Rate - Inches/minute.

RPM - Spindle speed.

Tool - Tool number.

Δ - Roughing pass. (Step over)

Spindle Direction - Forward or reverse.

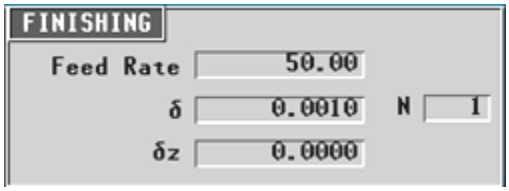
Finishing setup:

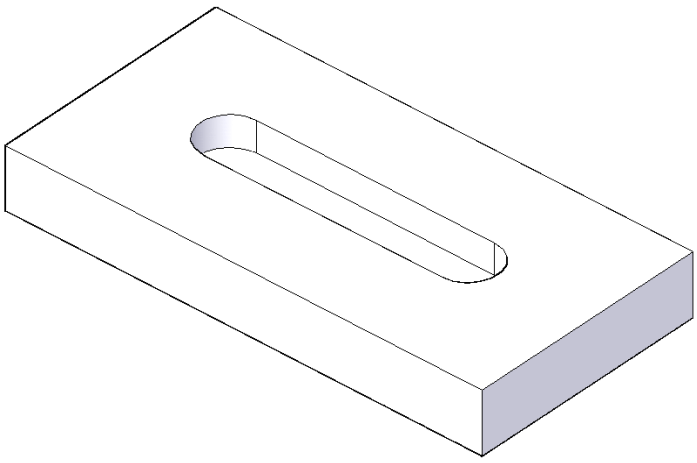
Feed Rate - Inches/minute.

δ - The amount you want to leave on the sides of slot for your finishing pass.

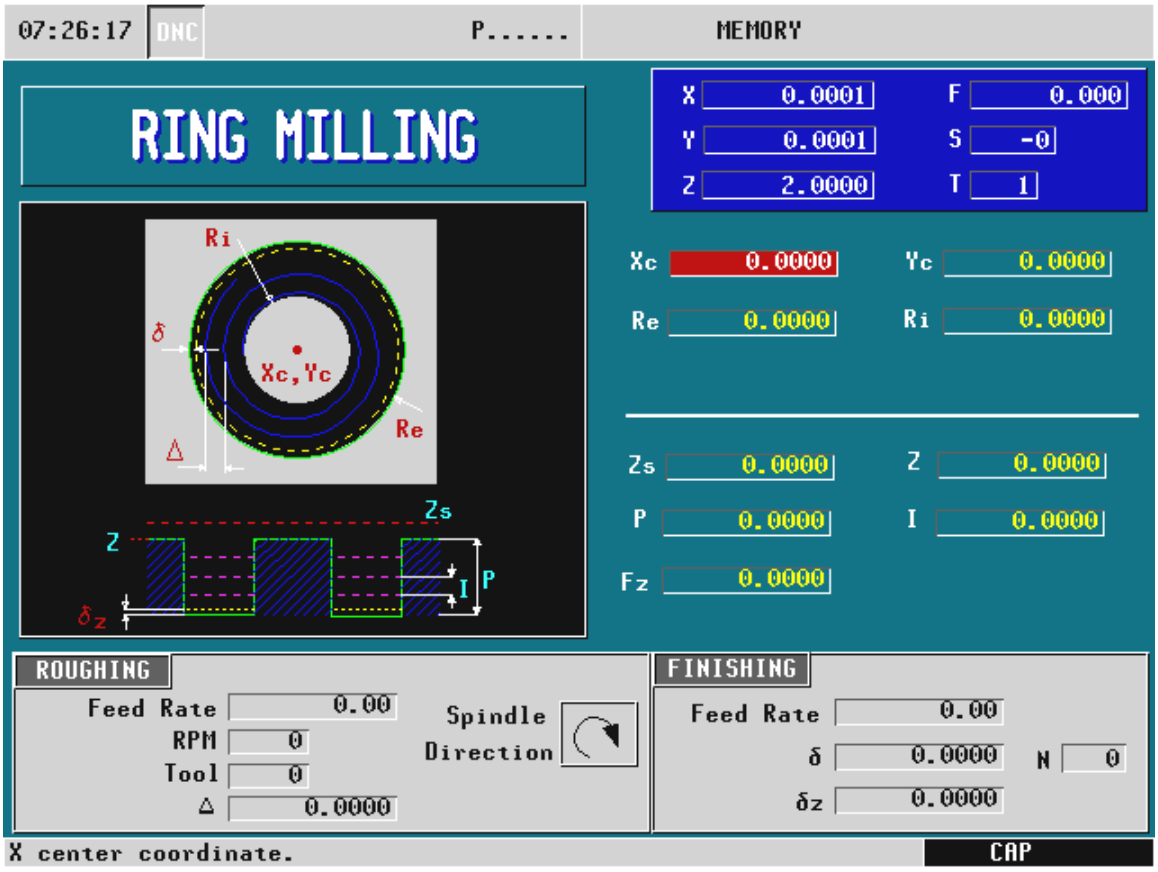
N - Number of finishing passes.

δz - The amount you want to leave on the floor of the slot for your finishing pass.





» *Ring Milling*



Programming a ring:

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

Re - Outside radius.

Ri - Inside radius.

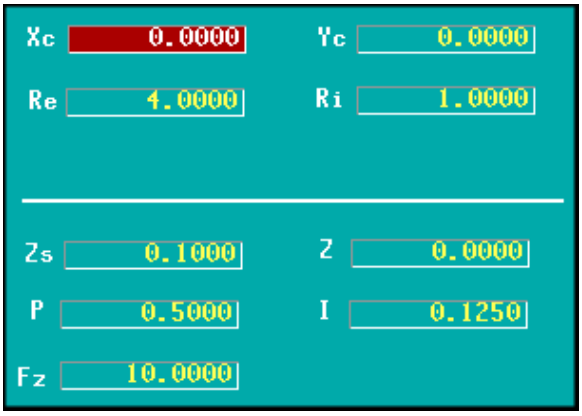
Zs - Safety distance for Z.

Z - Part Z coordinate.

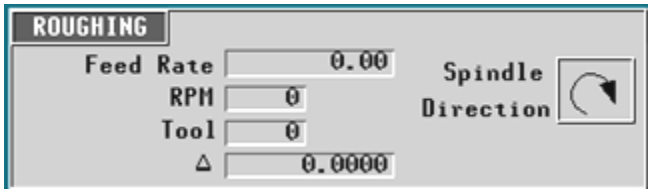
P - Ring depth.

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

Fz - Feed rate for Z axis.



Roughing setup:



Feed Rate - Inches/minute.

RPM - Spindle speed.

Tool - Tool number.

Δ - Roughing pass. (Step over)

Spindle Direction - Forward or reverse.

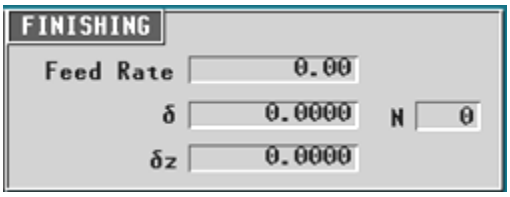
Finishing setup:

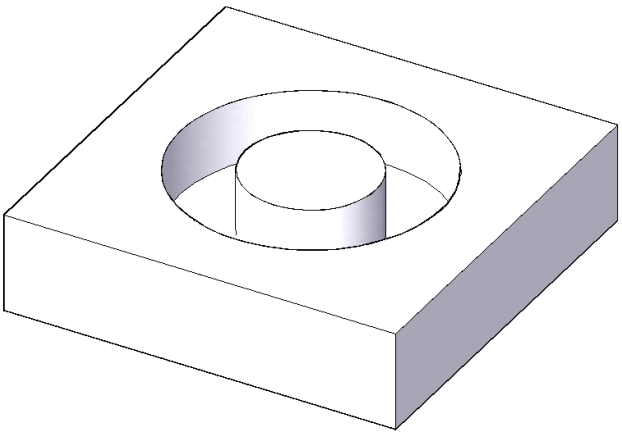
Feed Rate - Inches/minute.

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

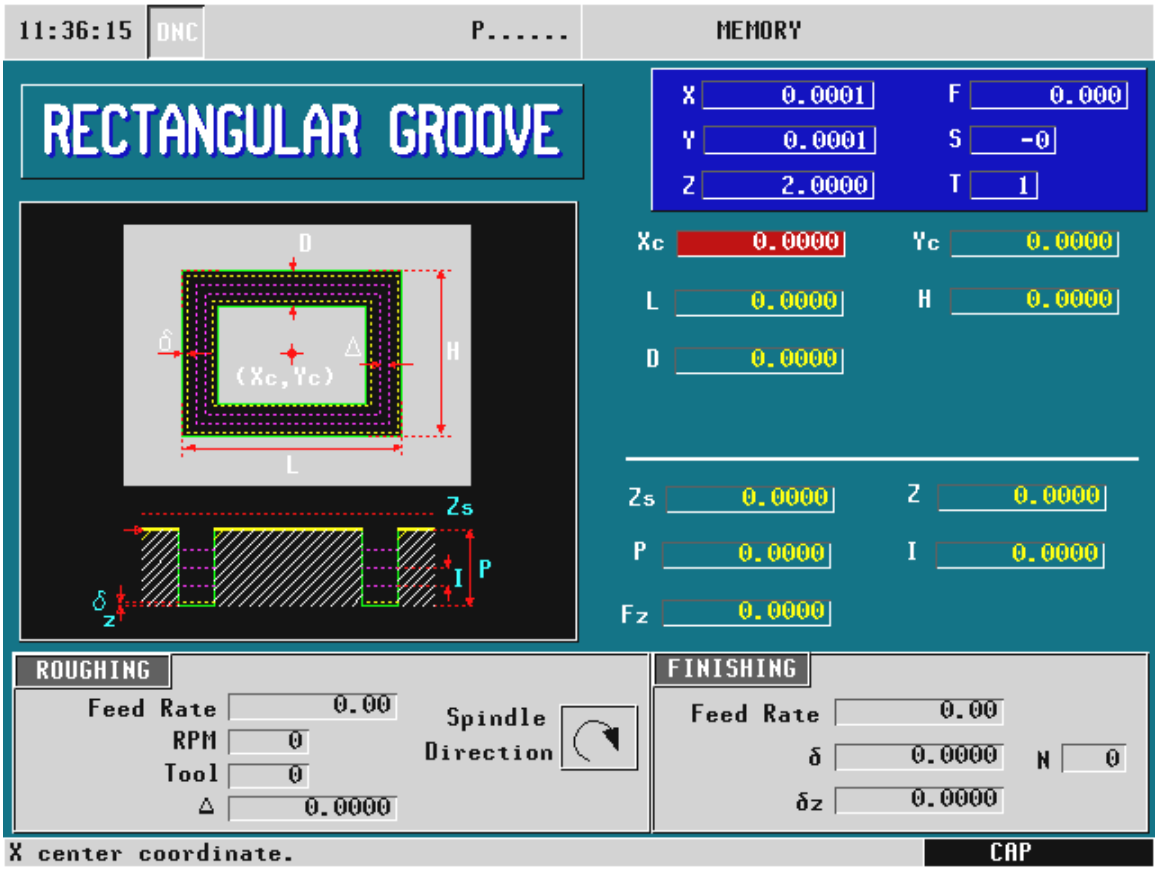
N - Number of finishing passes.

δz - The amount you want to leave on the floor of the ring for your finishing pass.



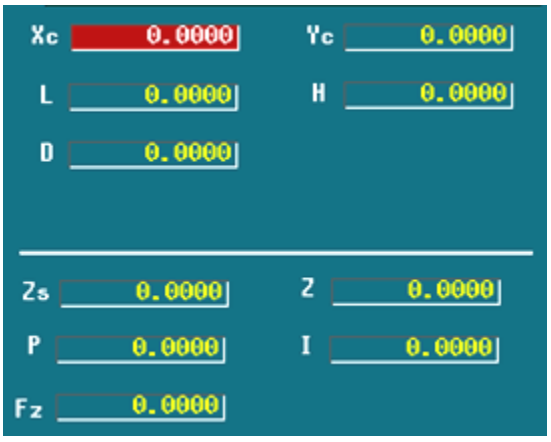


» *Rectangular Groove*



Programming a Rectangular Groove:

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



L – Rectangle length.

H – Rectangle height.

D – Rectangle width.

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

Roughing setup:

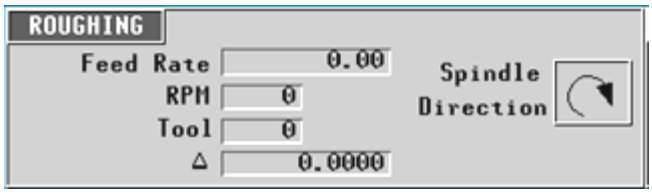
Feed Rate - Inches/minute.

RPM - Spindle speed.

Tool - Tool number.

Δ - Roughing pass. (Step over)

Spindle Direction - Forward or reverse.



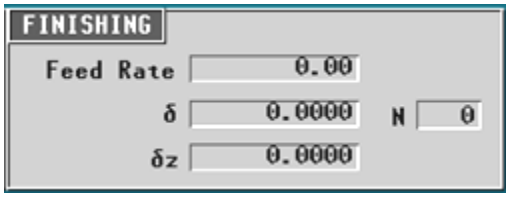
Finishing setup:

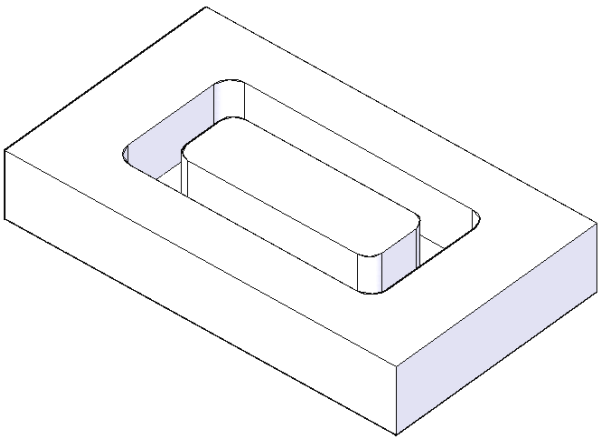
Feed Rate - Inches/minute.

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

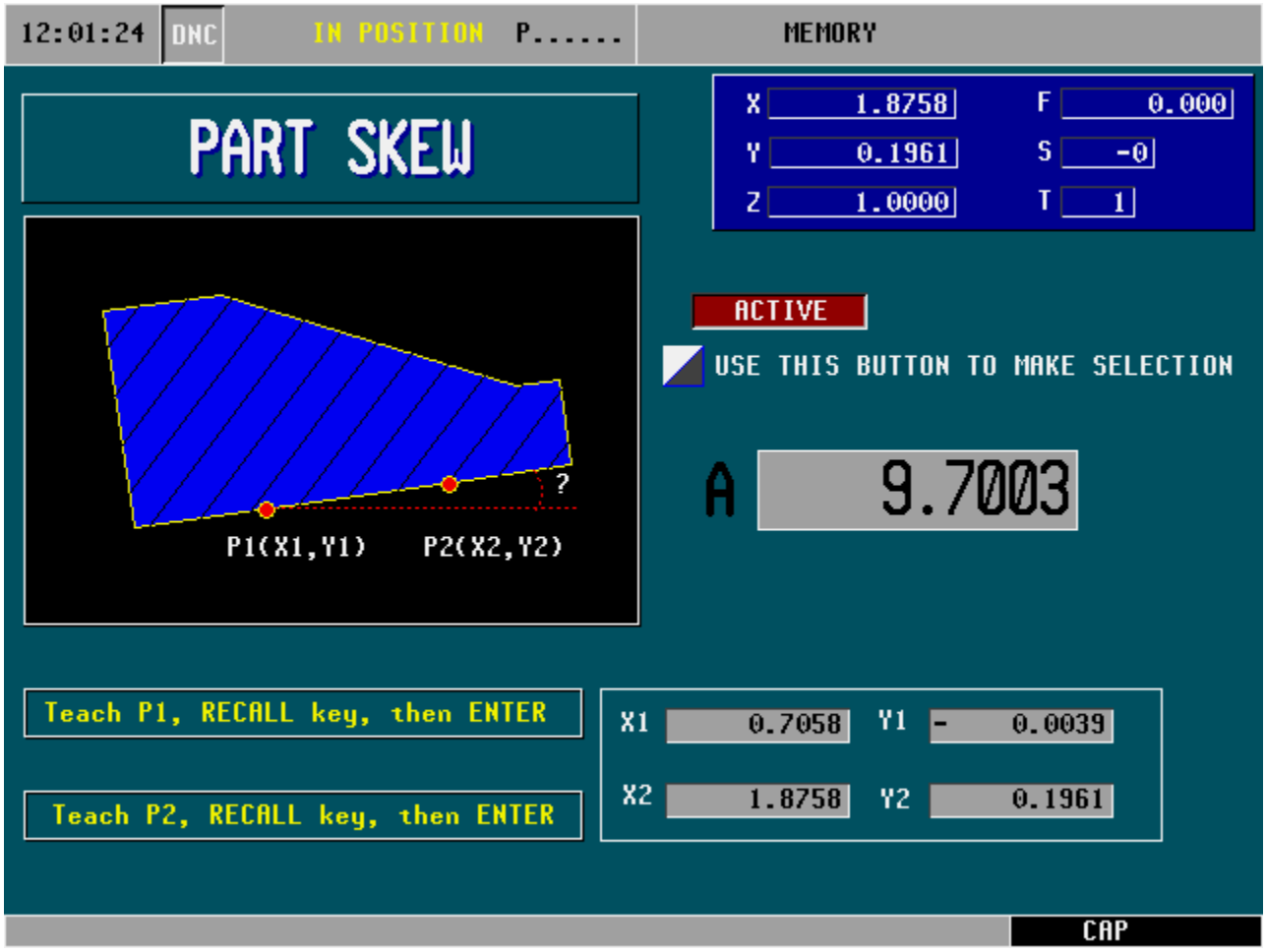
N - Number of finishing passes.

δz - The amount you want to leave on the floor of the rectangle for your finishing pass.





» *Part Skew*



Programming a Part Skew:

ACTIVE or OFF - Icons to turn on/off part skew feature with part program.

Teach P1, RECALL key – Highlight field and press RECALL key to load X & Y Points automatically.

Teach P2, RECALL key – Highlight field and press RECALL key to load X & Y Points and Calculate skew angle automatically.

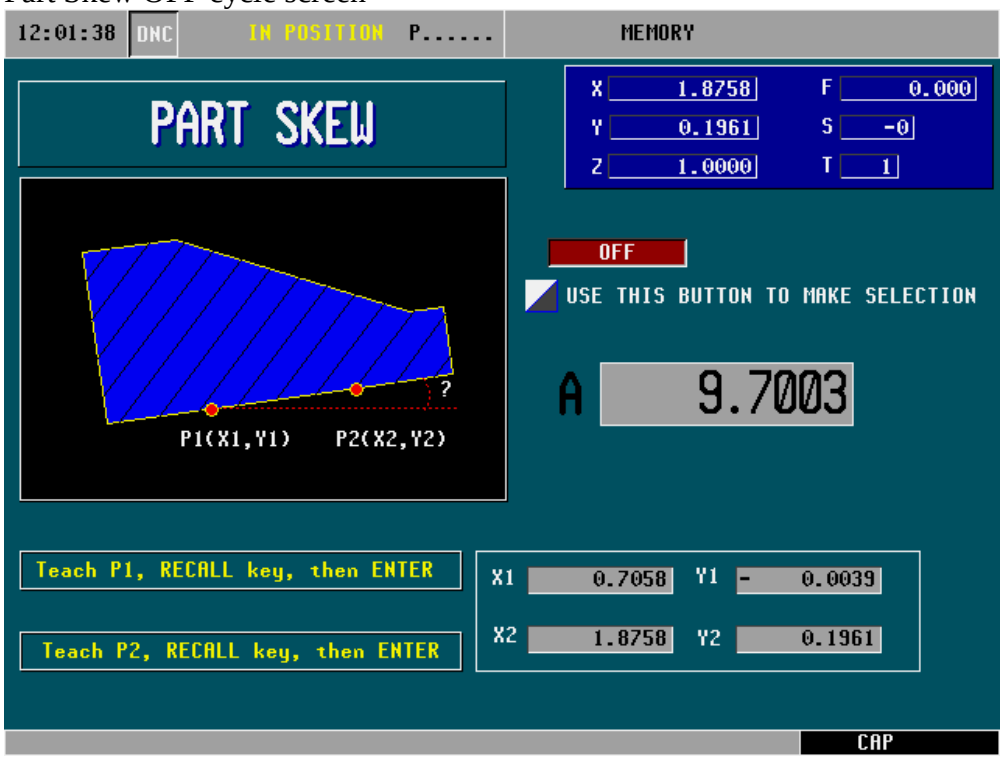
A- Calculated skew angle (read only field).

X1 Y1 First X Y point coordinate (read only field).

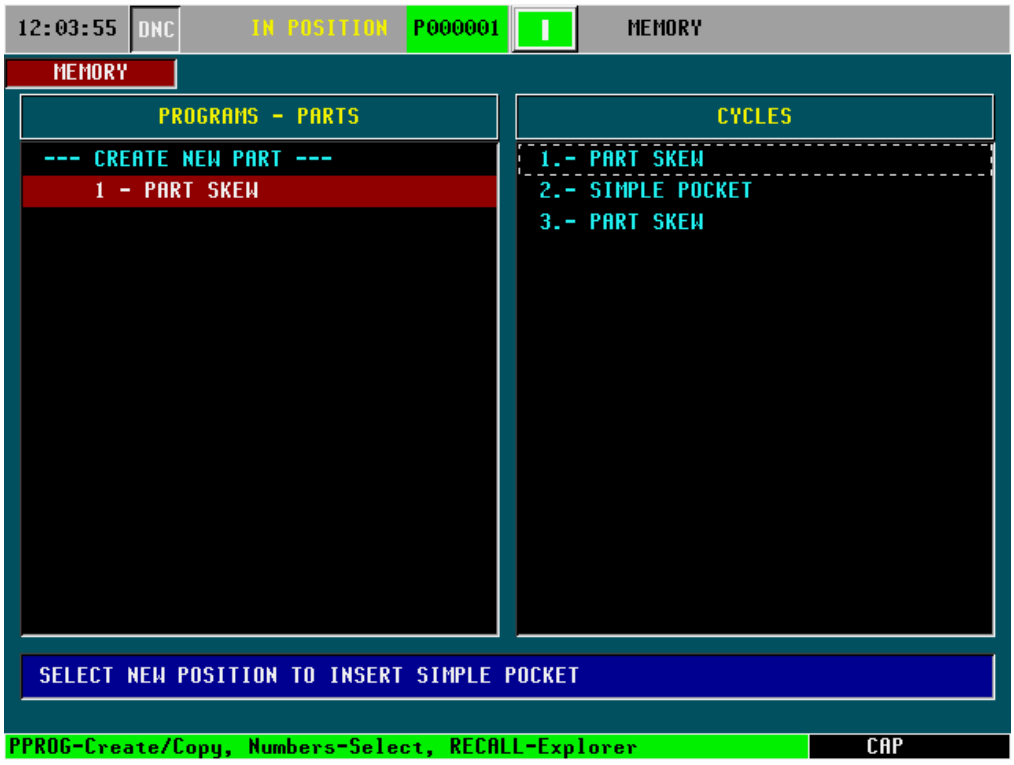
X2 Y2 First X Y point coordinate (read only field).

Note: Add Part Skew ACTIVE event to the beginning of a multi event part program to skew all events that follow, then add Part Skew OFF event to the end of a multi event part program or after any desired event to cancel the skew feature. See example below.

Part Skew OFF cycle screen



Event 1 = Part Skew ACTIVE
Event 2 = Any Conversational Cycle (sample Simple Pocket)
Event 3 = Part Skew OFF



» *Engraving*

First steps for Engraving:

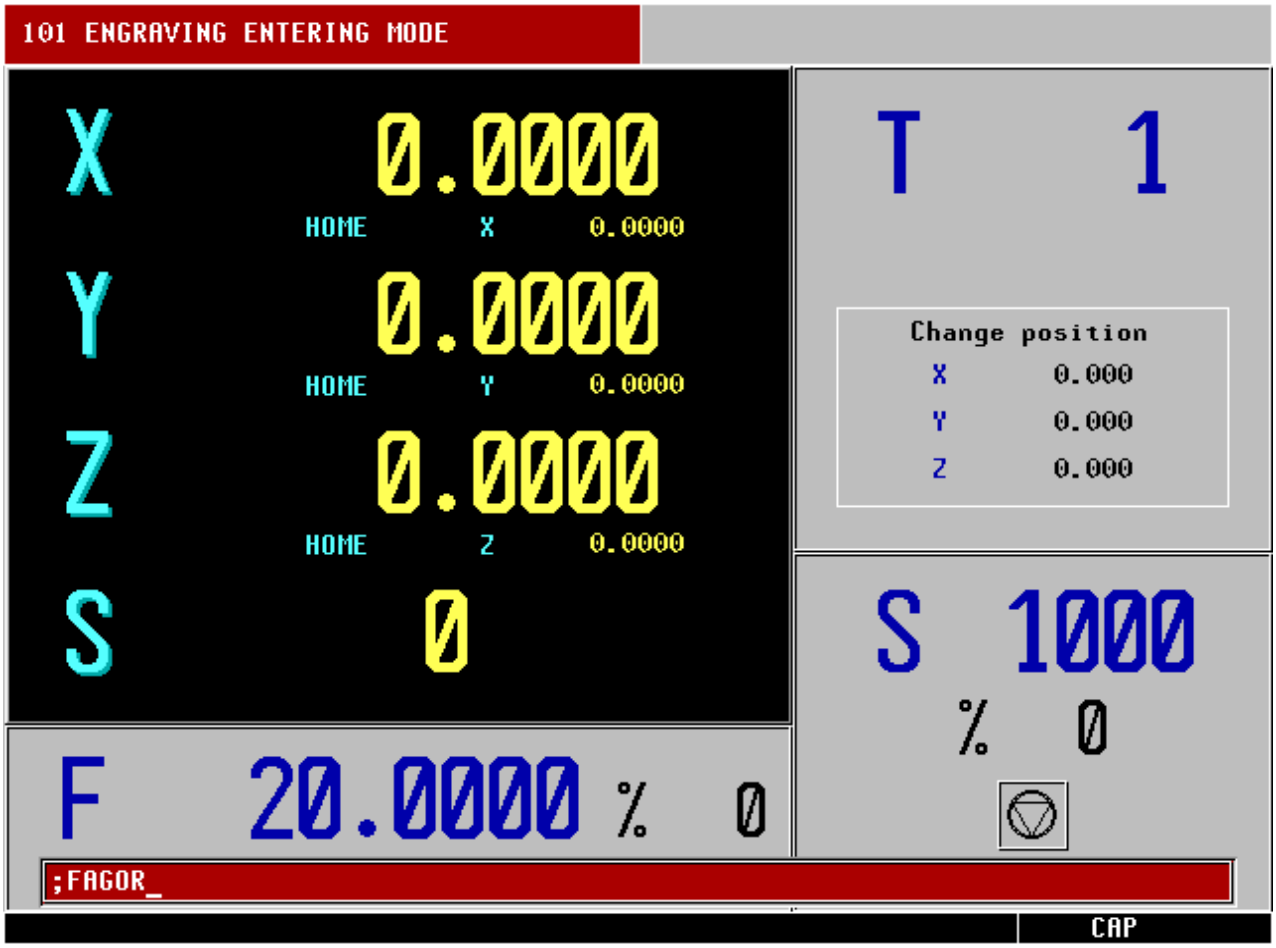
1.) Press the ISO icon key on the front panel below the F2 key. (A red bar will appear on the bottom of the screen)

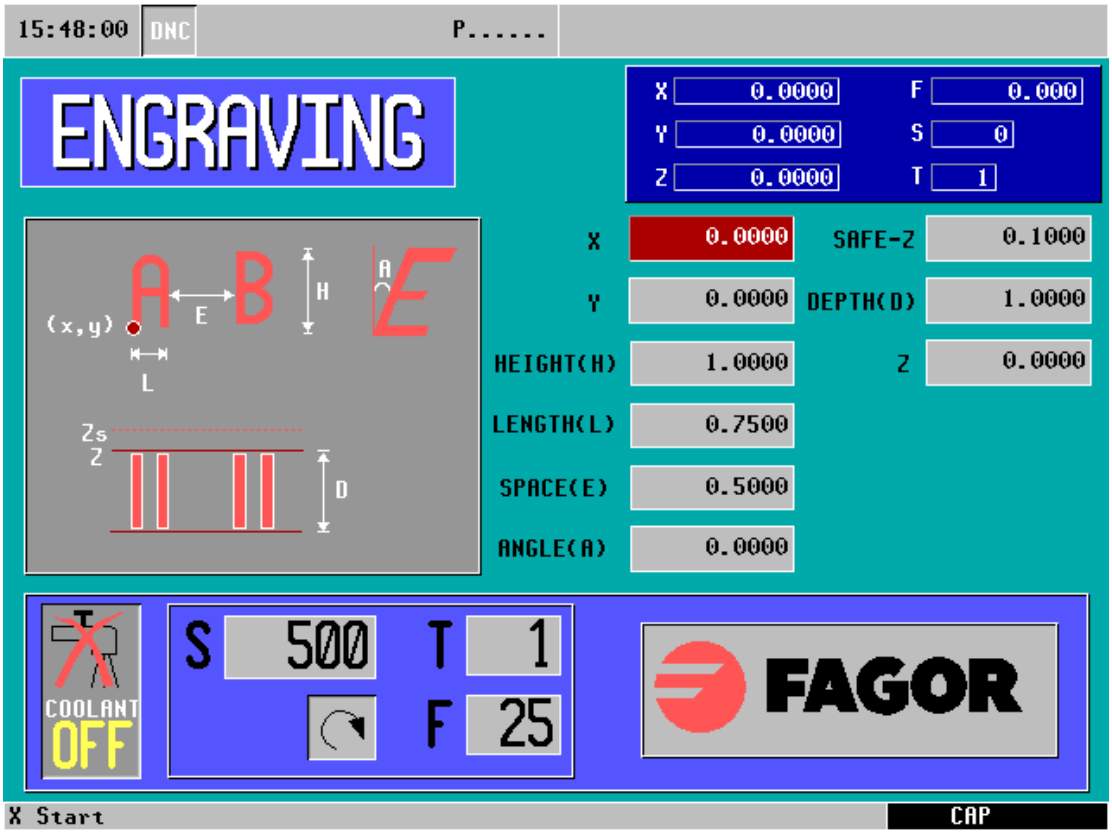
2.) Press SHIFT and then number 0 to add a semicolon to the bar below.

3.) ENGRAVING ENTERING MODE will now appear at the top left screen.

4.) Type in the characters you wish to engrave after the semicolon in the bottom bar.

5.) Press ENTER to save the characters to the temporary memory and launch the engraving cycle screen automatically.





Programming an Engraving :

X,Y – Enter the X and Y corner coordinates.

H – Height of characters.

L – Width of characters.

E – Space between each character.

A – Oblique angle.

SAFE-Z – Safety distance for Z.

D – Engraving depth.

Z – Part Z coordinate.

Coolant– On or off.

S– Spindle speed.

Spindle Direction - Forward or reverse.

T– Tool Number.

F– Feedrate.

Example graphics simulation of Engraving

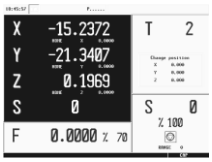
USB Specifications: Use Blank USB stick.
Do not use USB stick with encryption files on it.

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 1st line of the program should have %programcomment,MX,

G-Code Side

Conversational Side



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.

One Button Software Backup

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 + (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*)

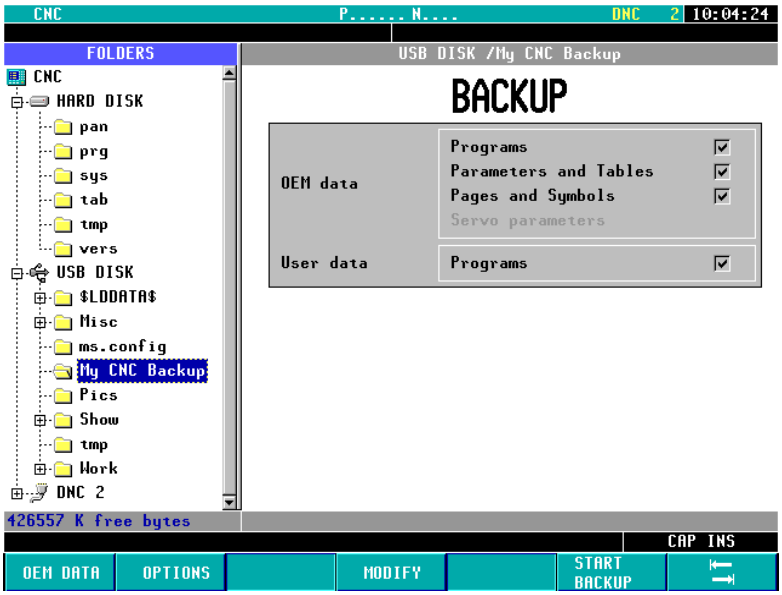
9. Confirm that all parts of the backup are checked off.

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 *

8055 Turnkey Package

Elrod Knee Mill Installation Kit

Fagor Automation Corp. Tel.: 847-981-1500
2250 Estes Avenue Fax.: 847-981-1311
Elk Grove Village, IL 60007

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

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)**

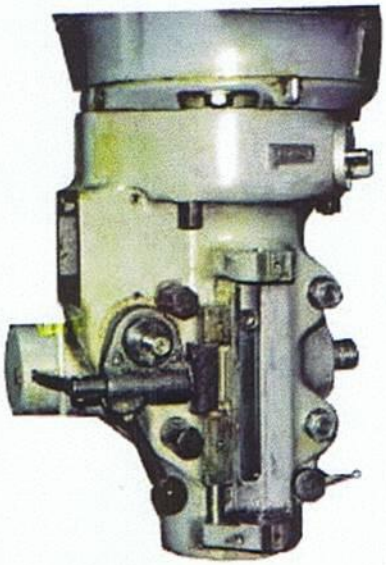


Figure1

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)**

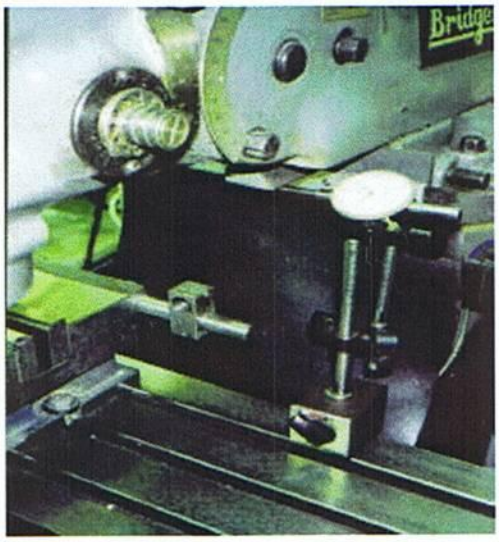


Figure 2

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! <u>There is plenty of stock on the casting in this area do not be concerned</u>!

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)**

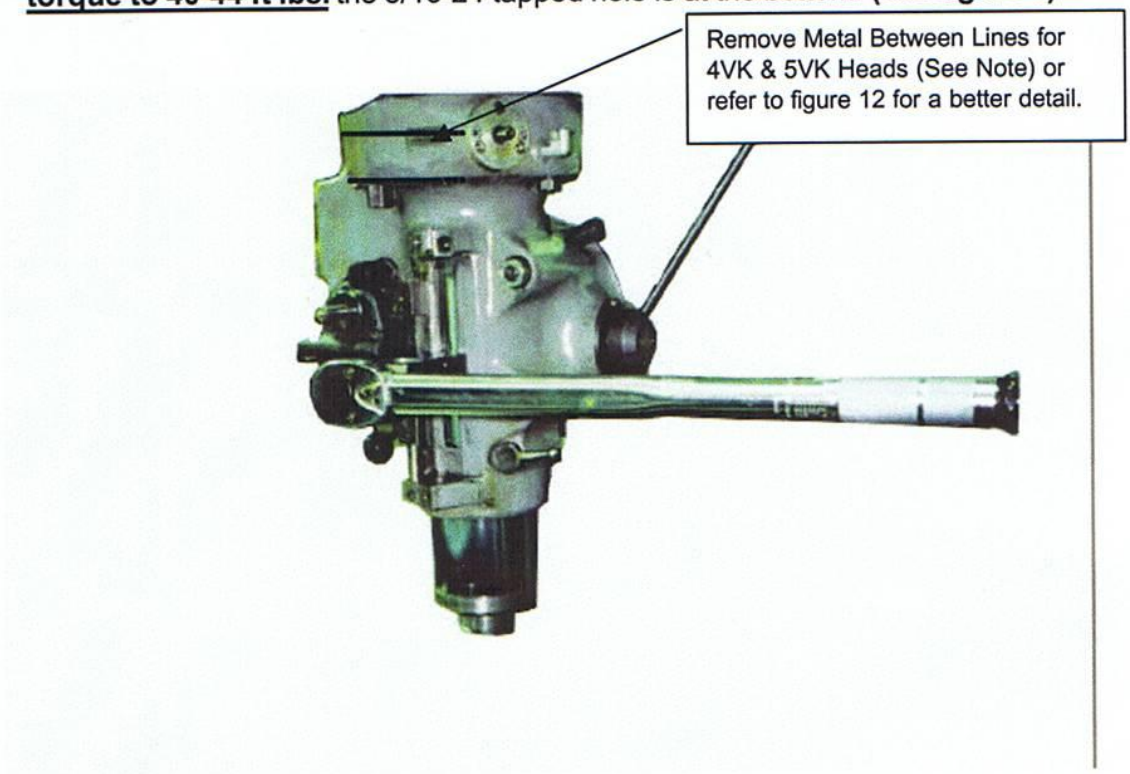


Figure 3

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)**

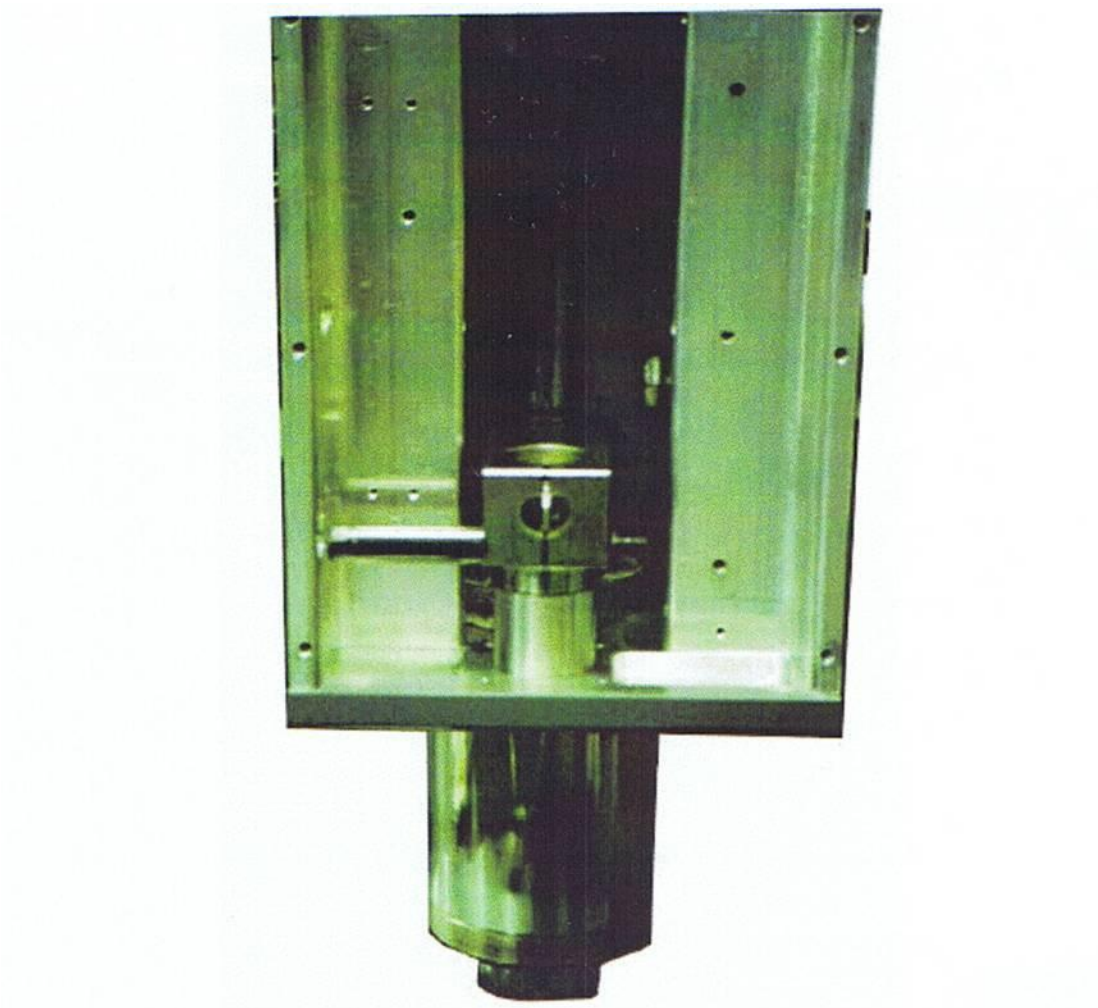


Figure 4

9. Remove the bearing access plate in the bottom of the lower assy.
0. 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.
1. Keep the quill in the lower position using the lock lever.
2. Positon and indicator on the front face of the bottom plate of the lower assy. **(See Figure 5)**

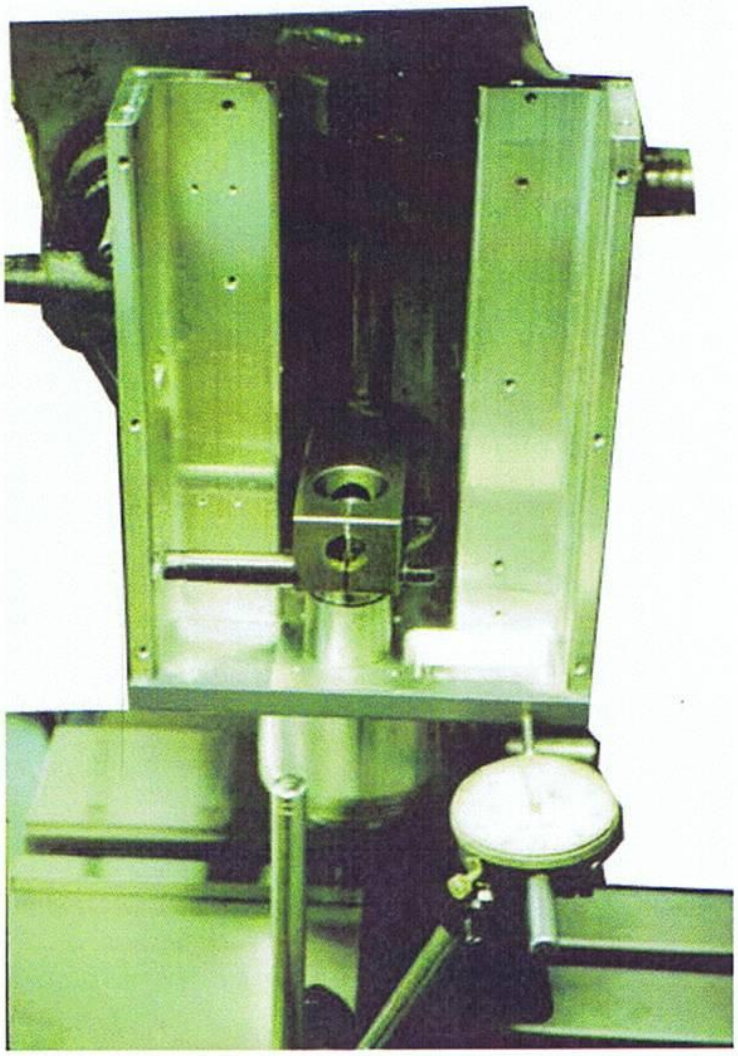


Figure 5

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 scecure 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)**

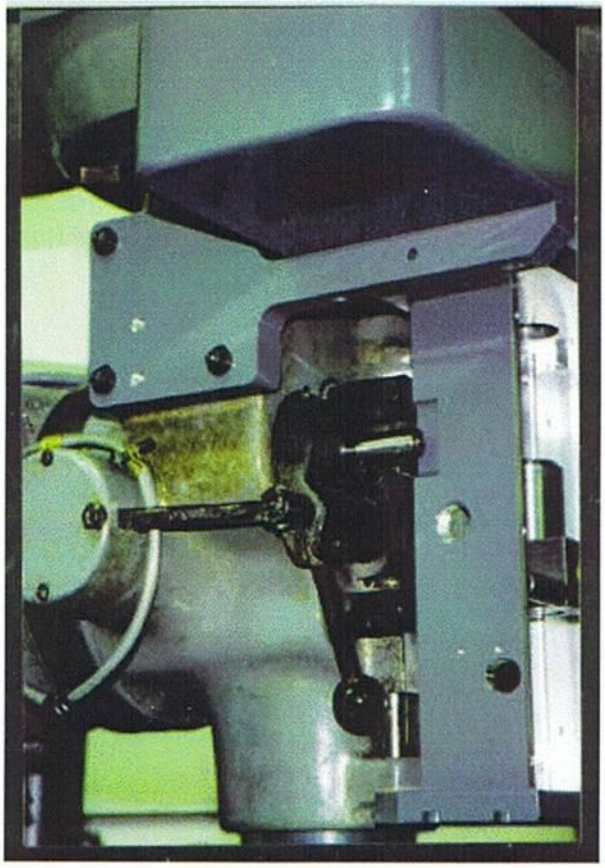


Figure 7

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 ¼" 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)**

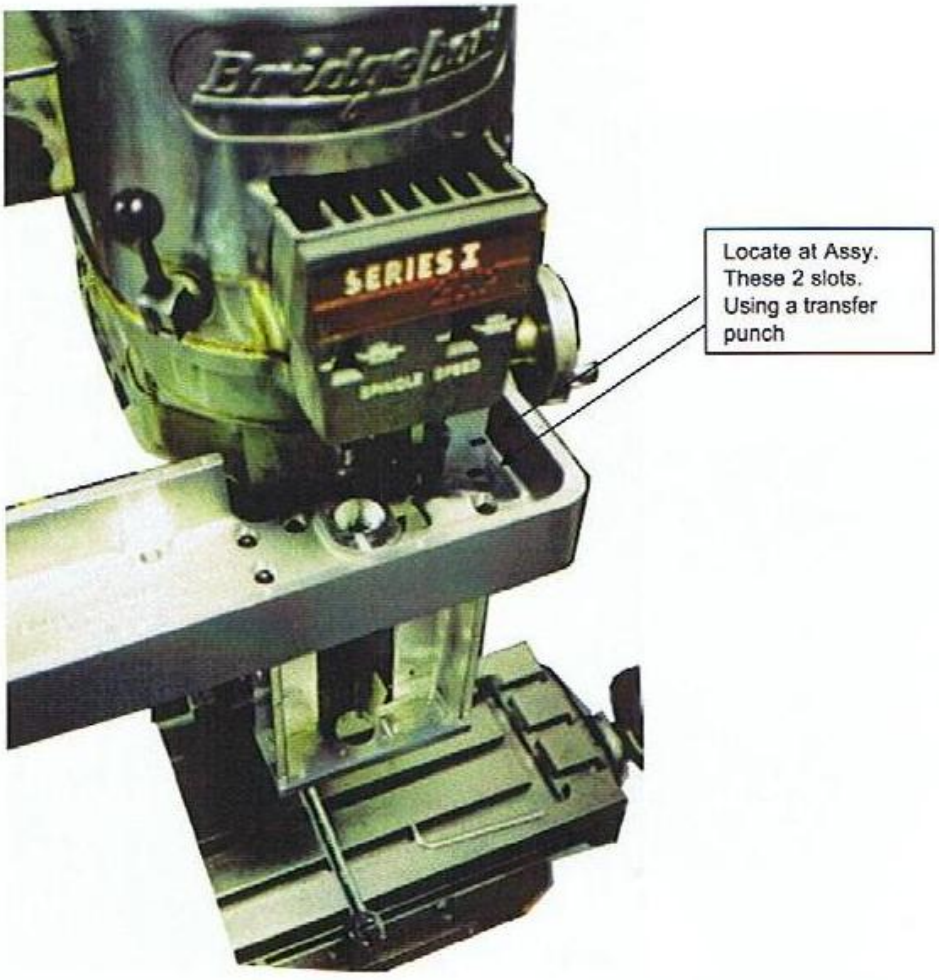


Figure 8

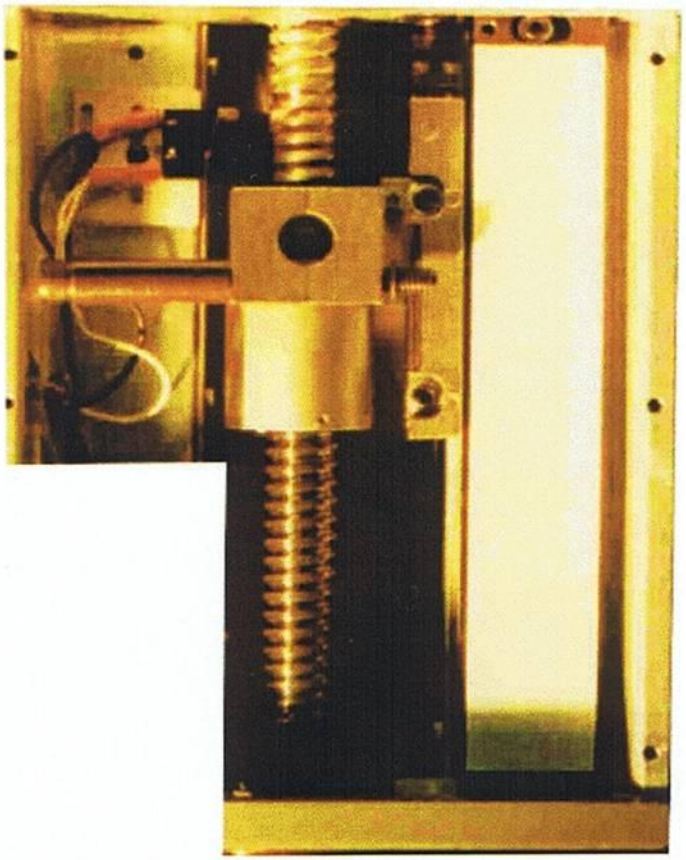


Figure 9

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)**

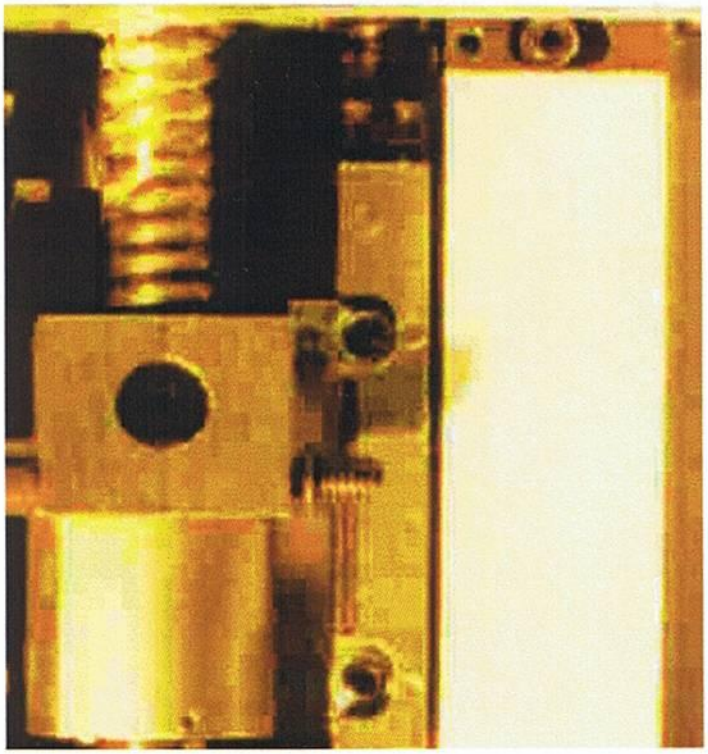


Figure10

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)**

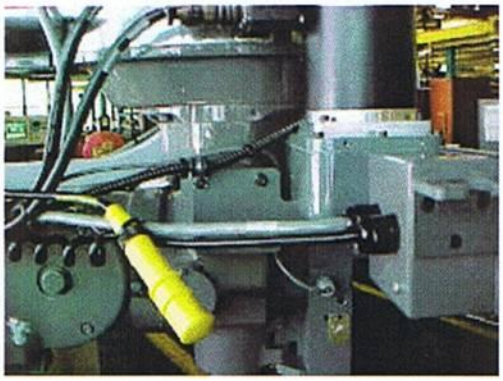
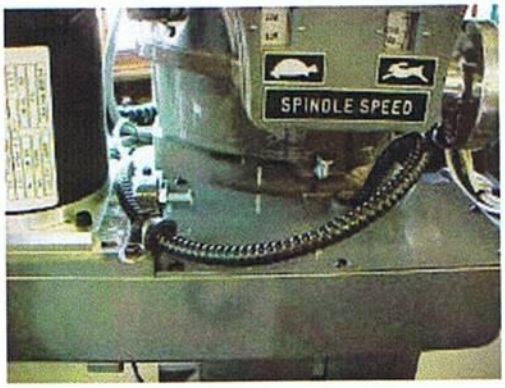
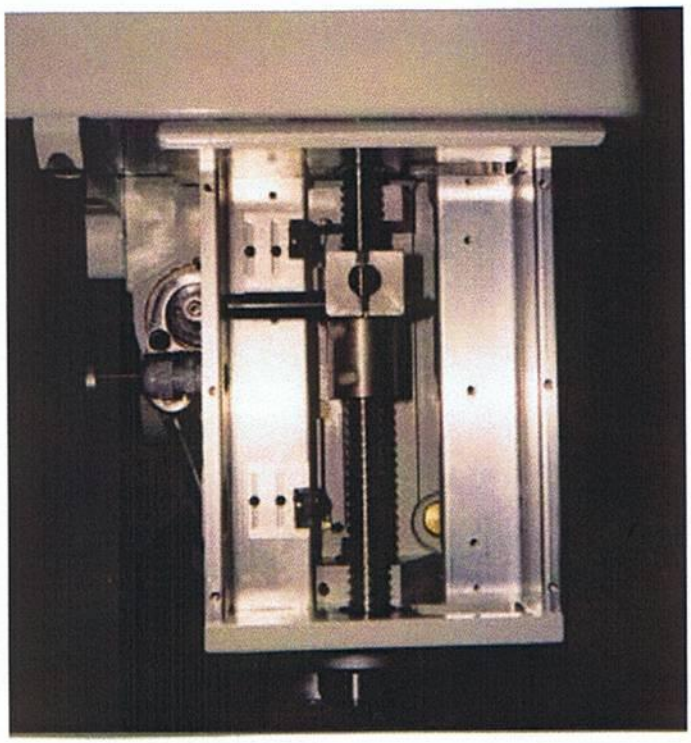


Figure 11 A,B,C

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.
i. 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.
i. 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)**

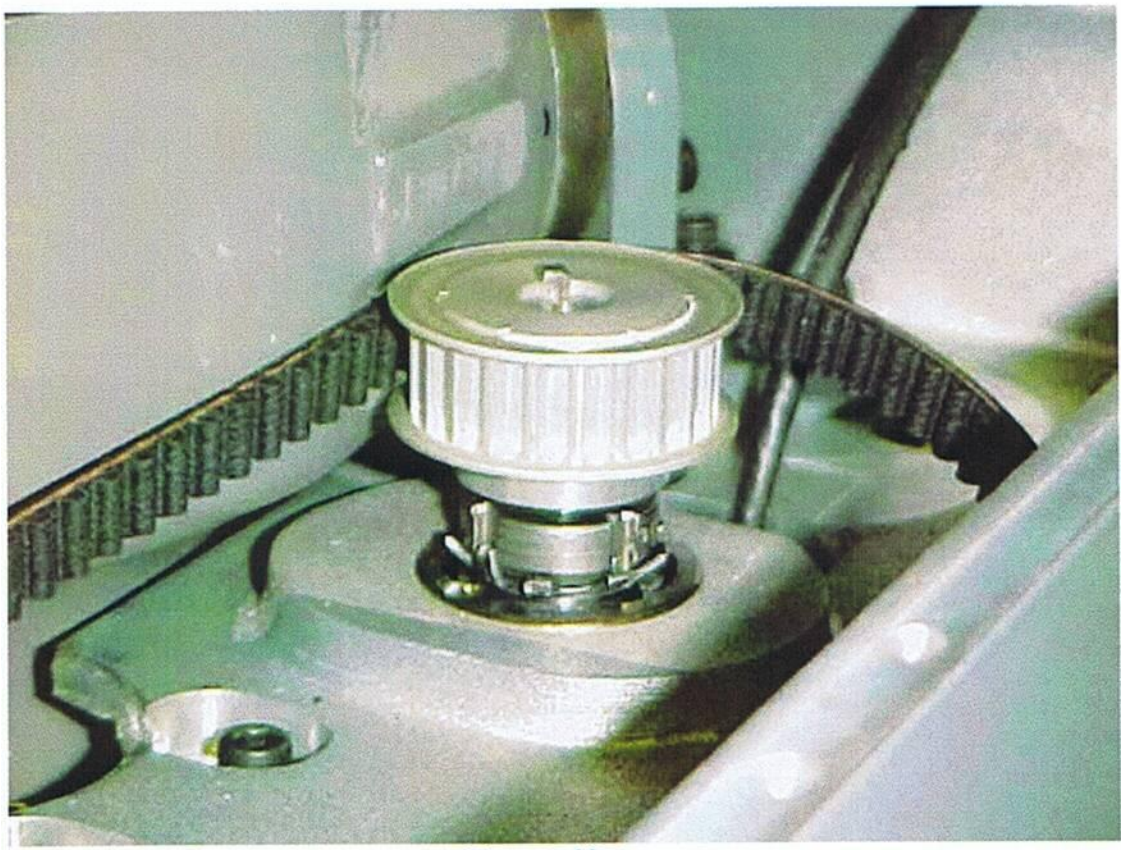


Figure 13

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) ¼-20 x 1 ½" 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 achived from the face of the motor plate to the first flange. Secure the pulley.**(See figure 14)**

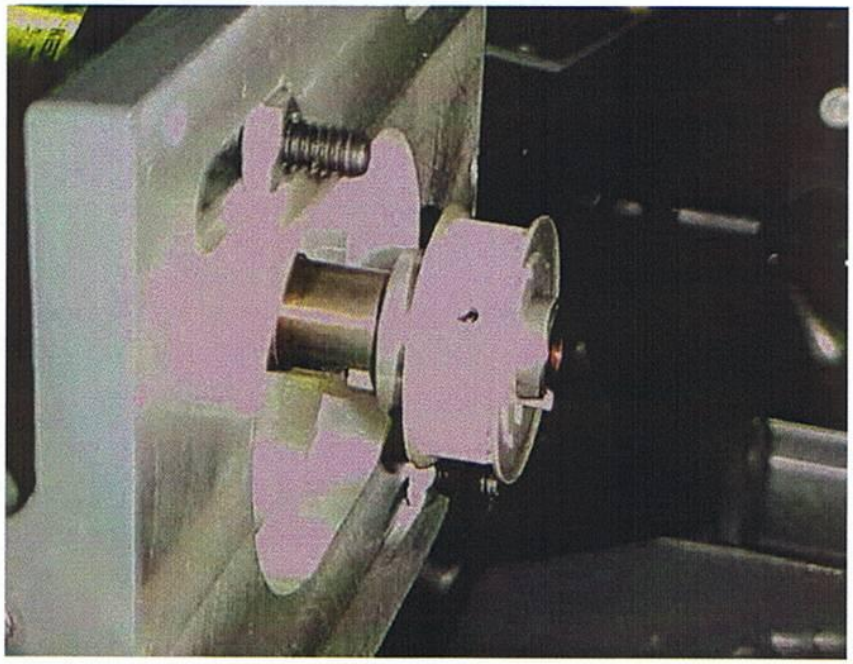


Figure 14

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 ¾" SHCS supplied.
70. Screw the ¼-20 pan head screw against the nema flange until the 194 belt is preloaded with tension. Tighten the ¼-20 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 in the normally closed position or normally open position **(See you 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 ¼ .
3. Use a 1/4 inch allen wrench and loosen the 5/16-24 s.h.c.s. in the ballnu
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 i 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.

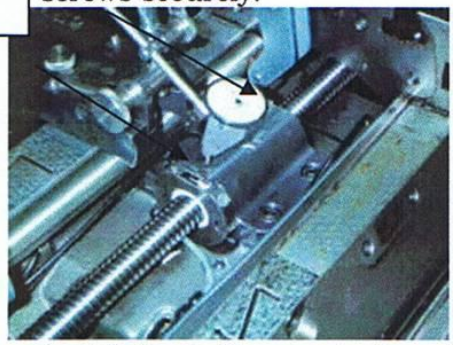
ved to
#4 & 5
ually
ith
& #40

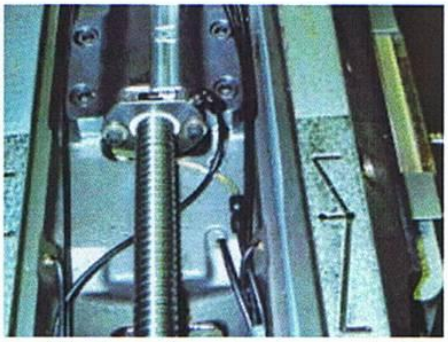
Elrod X&Y Bracket
Model DOXLFT95
Taper Lock Pulleys
REV 20404

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 and indicator with a surface gage to help with the alignment. Place ¾" 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.





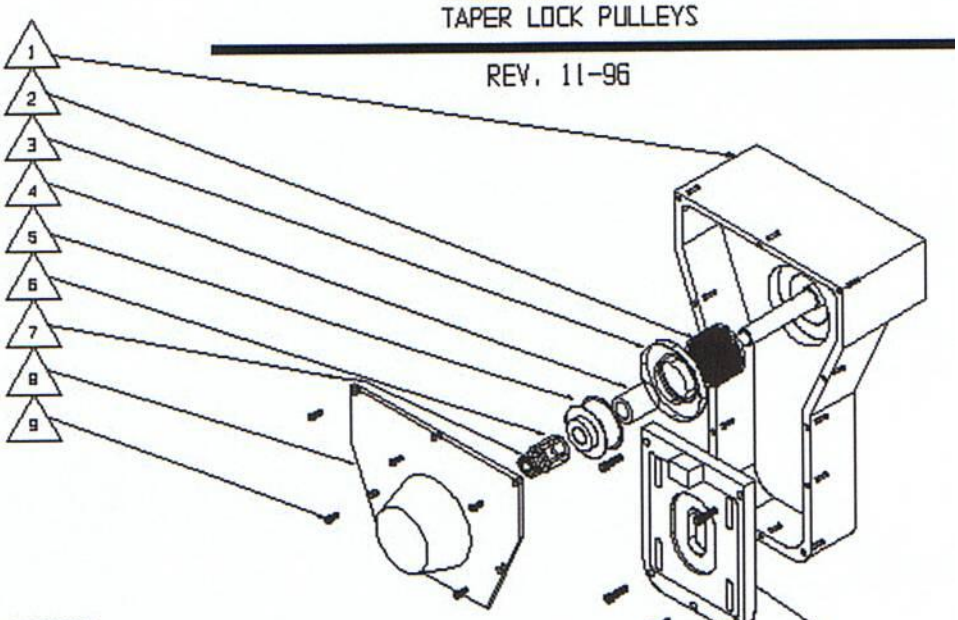
6. Install the ball screw in the yoke, the long bearing journal is on the left side with most screws.
7. Locate the lube access hole on the nut flange and install the angled fitting and insert lube line in it.
8. Grease the duplex angular contact bearings and install them in the housing with the word 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 DDXLFT95
TAPER LOCK PULLEYS

REV. 11-96

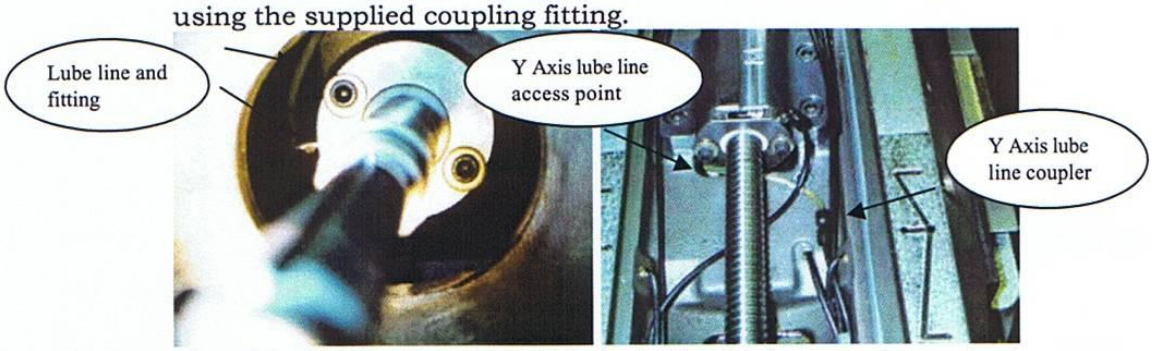


1. X CASTING
2. 7204 YDU BEARINGS WITH .001" SPACER
3. BEARING RETAINER (USE EXISTING PART AND QTY.3 S.H.C.S. FROM YOUR MILL)
4. 1.875" LONG SPACER FOR 1:1 OR 1.3125" LONG SPACER FOR 2:1
5. 22 OR 32 TOOTH PULLEY (1:1 RATIO 2 22T) (2:1 RATIO 1 32T 1 16T)
6. 7/16" LONG SPACER 1:1 5/8" LONG SPACER FOR 2:1
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 X35 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.

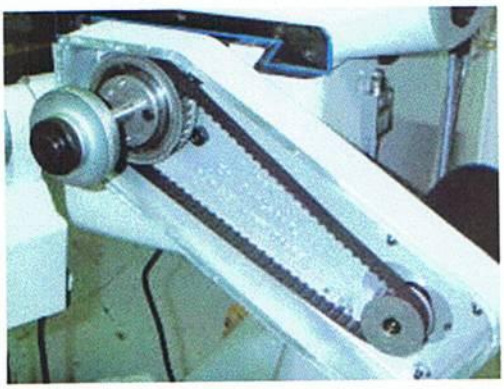


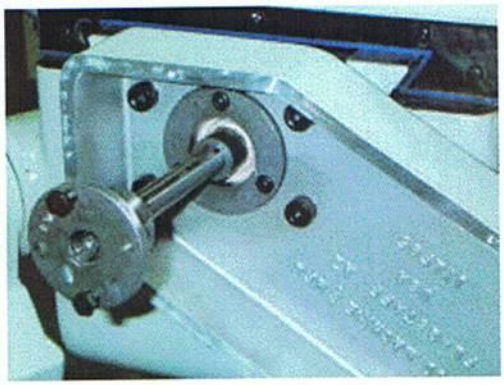
7. Use white Lithium grease to lubricate the duplex angular contact bearings and install them with the word thrust facing each other on each bearing in the Y housing.
8. Use the original bearing retainer ring to hold them in place.
9. Tighten the screws in the retainer equally to prevent any tilting.
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 proceding!**
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 ¼-20 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





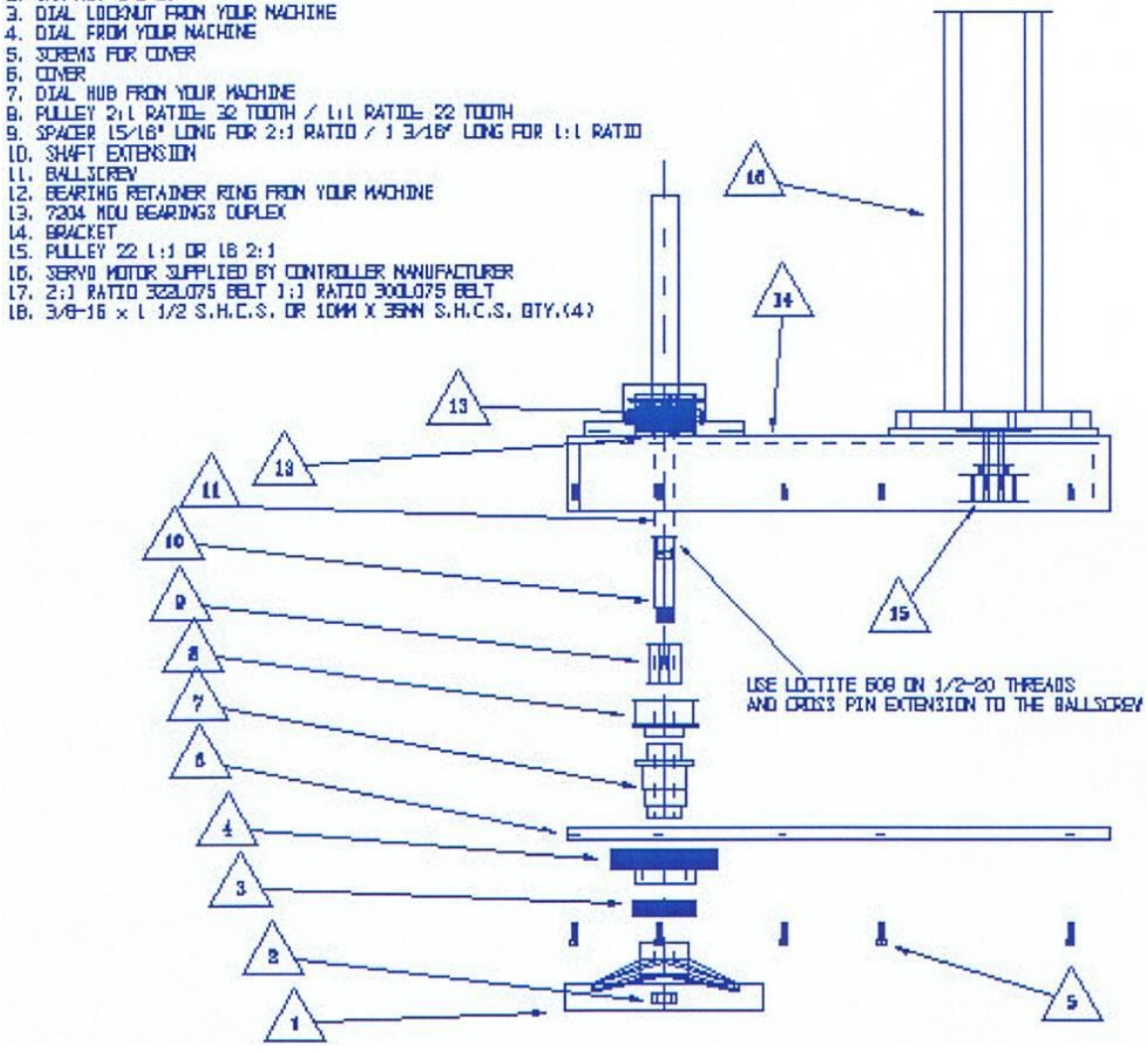
Elrod X&Y Bracket
Model DOXLFT95
Taper Lock Pulleys
REV 20404

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

EXPLODED ASSEMBLY

PARTS LIST

1. HANDWHEEL
2. JAM 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 MOU BEARINGS DUPLEX
14. BRACKET
15. PULLEY 22 1:1 OR 16 2:1
16. SERVO MOTOR SUPPLIED BY CONTROLLER MANUFACTURER
17. 2:1 RATIO 322L075 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)

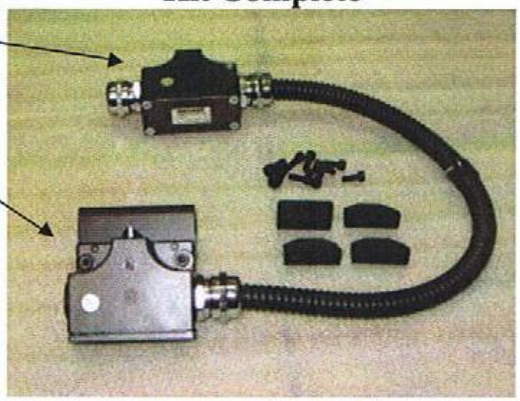


Elrod Limit Switch Kit

Y Axis Switch and Bracket

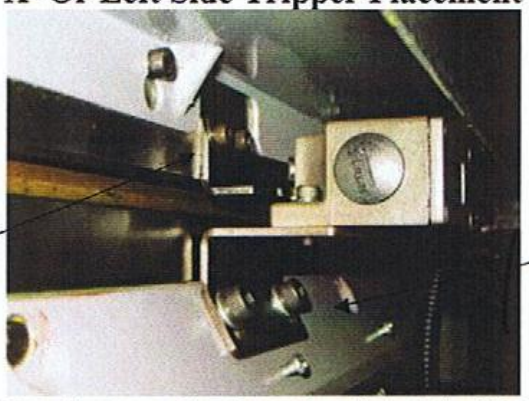
X Axis Switch and Bracket

Kit Complete



X- Or Left Side Tripper Placement

Long Leg Tripper Bracket

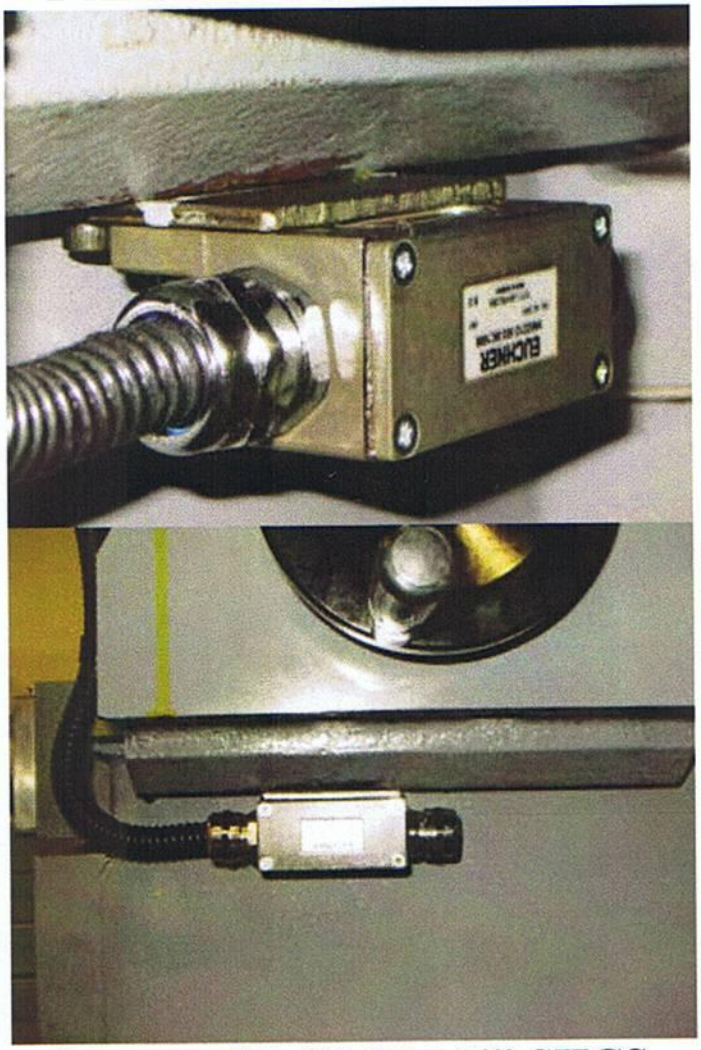


Mount X Axis Switch using original holes in the front face of the saddle your crosslide

Use the long leg bracket for this side!

<u>Tripper positioned to trip the bottom plunger</u>

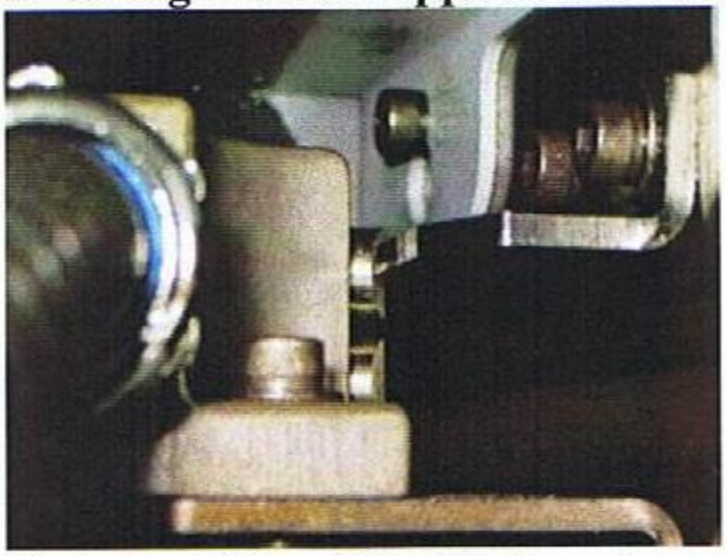
Y Axis Limit Switch Placement



Mount with 10-32 x ½" SHCS

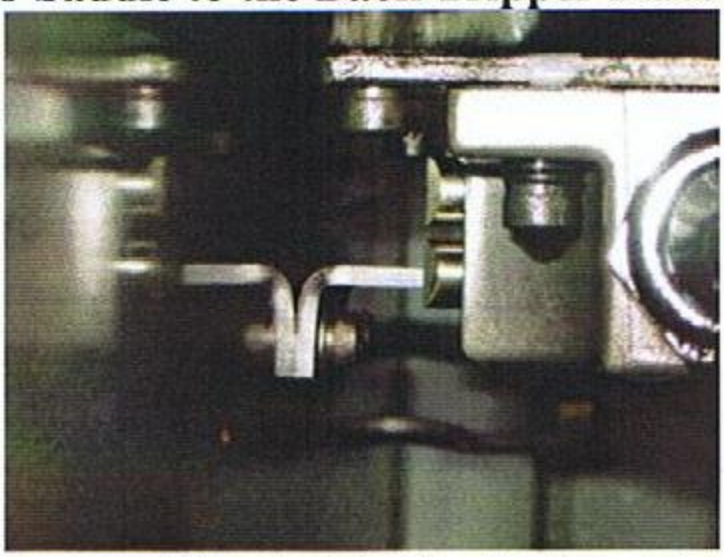
Note: Position the switch plungers close enough to be actuated with trippers when mounted Approx. .750" to .812" from the side of the knee.

X+ or Right Side Tripper Placement



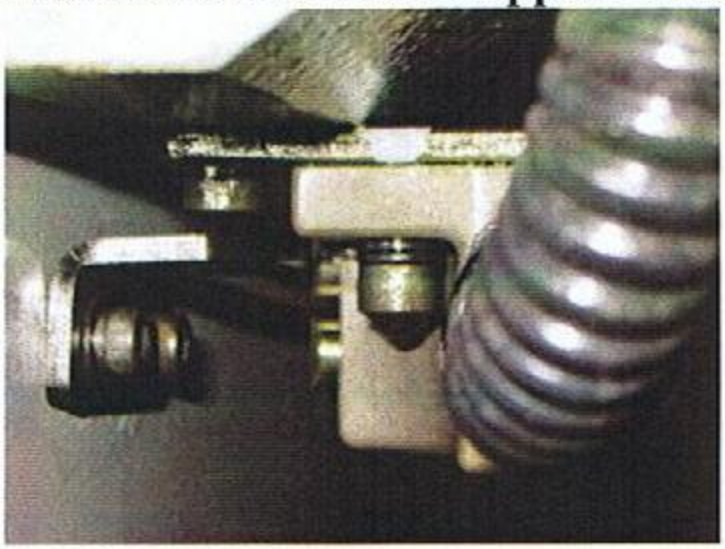
Tripper positioned to trip top plunger

Y - or Saddle to the Back Tripper Placement



Tripper position to trip the bottom plunger

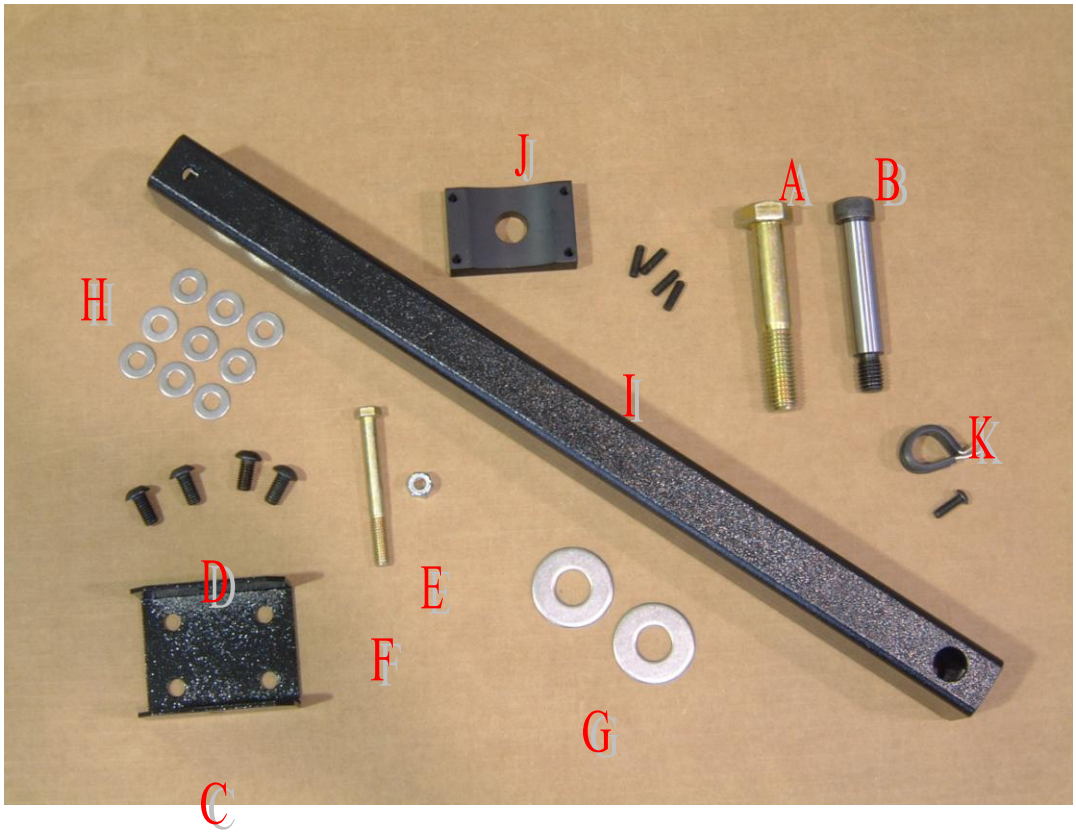
Y+ or Saddle to the Front Tripper Placement



Tripper positioned to trip the top plunger

Completed Installation

TURNKEY MOUNTING ARM KIT ASSEMBLY



1 - Bolt hex head ¾-10 x 4 ½ (A) **(Note: used on oriental machines)**

1 - Bolt SHCS shoulder bolt 5/8-11 x 3 ¼ (B) **(Note: used on Bridgeport machines)**

1 – C shape bracket (C)

4 - Bolts BHCS 3/8-16 x ¾ (D)

1 - Nut nylock 3/8-16 (E)

1 - Bolt hex head 3/8-16 x 3 ½ (F)

2 - Washer 3/4 ID (G)

9 - Washer 3/8 ID (H)

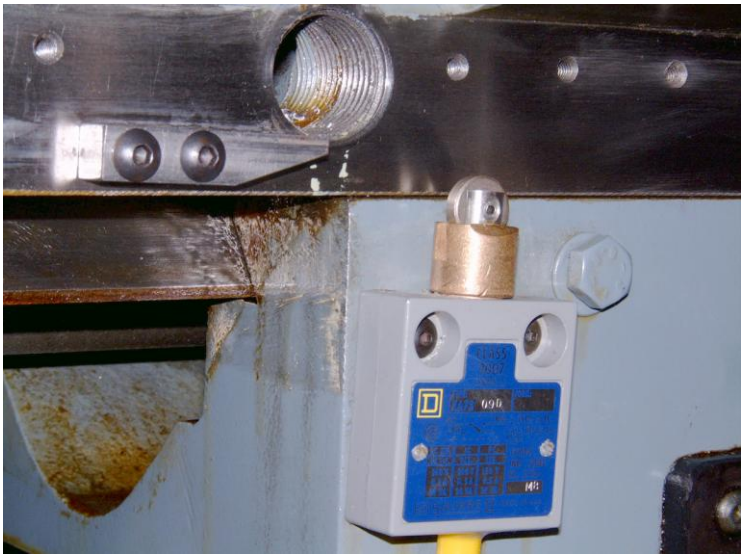
1 – 22” Arm (I)

1 – Saddel Braket with 4 jacking screws M6 x 20mm (J)

1 – Motor cable clamp and BHCS ¼ - 20 x 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

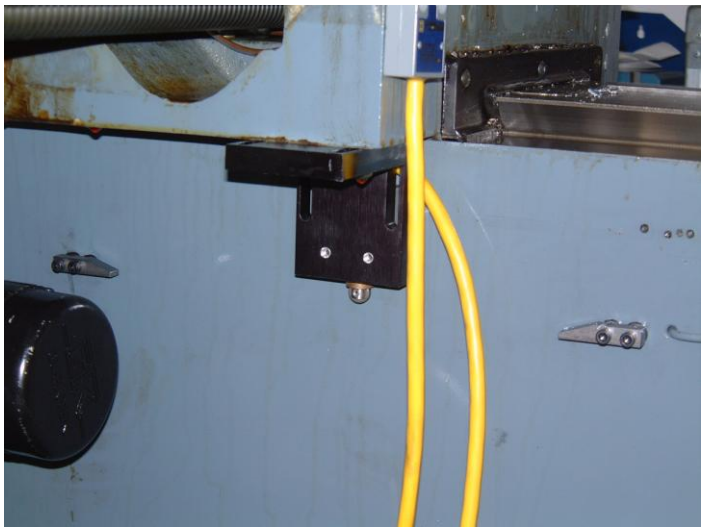
Note: The X axis dog may be different as shown above.

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.

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 form 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.

Fagor TKP Scale Setup

Once the Fagor Scale has been installed on the machine, the feedback must be changed from motor to scale feedback.

To install the Scale cable which is typically an EC cable, open the main electrical cabinet and slide the reader head side cable through the ¾" conduit clamp.

Next, pull the cable to appropriate length needed and screw the clamp nut ring:

Now take out the covers for the long wire duct. Notice you will have a cable with two connectors on in. Usually it will be marked X10 for X axis, X11 for Y axis, X12 for Z axis, etc.

Disconnect the bottom connector with a flat head screw driver and connect the EC cable and tighten.

Now it will run on Scale feedback. The motor feedback will not be connected. It will remain disconnected. Put the covers back on the wire duct.

On the outside of the main cabinet, pull the appropriate scale cable length needed. Now with a Phillips screw driver tighten the two screws on the conduit clamp outside the main cabinet. Do not over tighten the screws because it may break the cable.

Repeat these steps if more than one scale is installed.

Now the CNC must be set for Scale feedback.

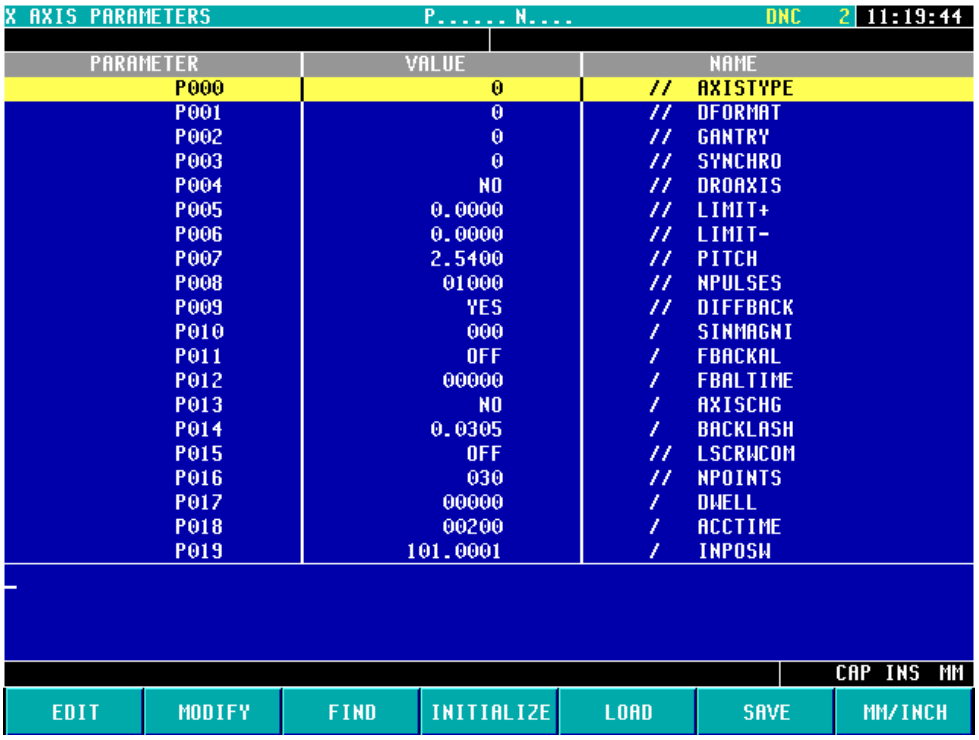
Fagor TKP Scale Parameter Set Up

Once the Fagor Linear Scale(s) have been installed, the Parameters must be set.

1) Go to the G-code side: [Shift], [Esc]

2) Go to the Machine Parameters: [+], [F4] for Machine Parameters, [F2] Axis Parameter. Choose the appropriate axis: [F1,F2,F3,etc].

3) Once the axis has been chosen, make sure the parameters are in METRIC… This can be done by pressing [F7]. (Below lower right corner of the screen).



To set up Parameters, please look at the appropriate Fagor Scale Group below.

SCALE GROUP	FAGOR SCALE MODELS
1	SX, SVX, GX, MX, MVX, MKX, CX, CVX
2	SP, SVP, GP, MP, MVP, CP, CVP
3	SOX, SOVX, GOX, MOX, MOVX, COX, COVX
4	SOP, SOVP, GOP, MOP, MOVP, COP, COVP

Once the appropriate Scale Group has been chosen, change the CNC Axis Parameter accordingly to the Scale Group. Please see Fagor Installation Manual for more details.

CNC AXIS PARAMETRS	SCALE GROUP 1	SCALE GROUP 2	SCALE GROUP 3	SCALE GROUP 4
(P5) LIMIT +	LENGTH OF SCALE	LENGTH OF SCALE	LENGTH OF SCALE	LENGTH OF SCALE
(P6) LIMIT -	LENGTH OF SCALE	LENGTH OF SCALE	LENGTH OF SCALE	LENGTH OF SCALE
(P7) PITCH	.004 MM	.02 MM	.004 MM	.02 MM
(P8) NPULSES	0	0	0	0
(P9) DIFFBACK	YES	YES	YES	YES
(P10) SINMAGNI	0	20	0	20
(P11) FBACKAL	ON	ON	ON	ON
(P14) BACKLASH	0	0	0	0
(P31) DECINPUT	YES/NO HOME SW.	YES/NO HOME SW.	YES/NO HOME SW.	YES/NO HOME SW.
(P32) REFPULSE	+	-	+	-
(P36) REFVALUE	USER DEFINE	USER DEFINE	USER DEFINE	USER DEFINE
(P52) IOTYPE	0	0	1 OR 2	1 OR 2
(P53) ABSOFF	0	0	0	0
(P57) EXTMULT	5	1	5	1
(P68) IOCODI1	0	0	1000	1000
(P69) IOCODI2	0	0	1001	1001

After the parameters have been change, Press [Shift], then [RESET]. This will reboot the CNC and the parameters will take effect.

If the Axis is jogging in the wrong direction, change Axis Parameter (P13) AXISCHG : YES/NO OR (P26) LOOPCHG: YES/NO

TURNKEY INPUTS

I 1	Emergency (pre-wired)	(Normally Open)	X2 pin 10
I 2	Spindle Overload/Fault	(Normally Open)	X2 pin 29
I 3	Z axis Limit (-)	(Normally Closed)	X2 pin 11
I 4	X axis Home/Limit (+)	(Normally Closed)	X2 pin 30
I 5	Y axis Home/Limit (+)	(Normally Closed)	X2 pin 12
I 6	Z axis Home/Limit (+)	(Normally Closed)	X2 pin 31
I 7	X axis Limit (-)	(Normally Closed)	X2 pin 13
I 8	Lube Level	(Normally Closed)	X2 pin 32
I 9	Spindle High Gear	(Normally Open)	X2 pin 14
I 10	Spinlde Brake	(Normally Open)	X2 pin 33
I 11	Y axis Limit (-)	(Normally Closed)	X2 pin 15
I 12	A axis Home	(Normally Open)	X2 pin 34
I 13	A axis unclamped	(Normally Open)	X2 pin 16
I 14	A axis clamped	(Normally Open)	X2 pin 35
I 15	Coolant Pump Overload	(Normally Closed)	X2 pin 17
I 16	Remote Cycle Start/Stop	(Normally Open)	X2 pin 36
I 97	Koyo X Select	(Normally Open)	X7 pin 14
I 98	Koyo Y Select	(Normally Open)	X7 pin 33
I 99	Koyo Z Select	(Normally Open)	X7 pin 15
I 100	Koyo 4th Select	(Normally Open)	X7 pin 34
I 101	x1 Resolution	(Normally Open)	X7 pin 16
I 102	x 10 Resolution	(Normally Open)	X7 pin 35
I 103	x 100 Resolution	(Normally Open)	X7 pin 17

TURNKEY OUTPUTS

O 1	CM Master Relay	X2 pin 2
O 2	X and Y Drive Enable	X2 pin 21
O 3	M3	X2 pin 3
O 4	M4	X2 pin 22
O 5	M8	X2 pin 4
O 6	Z Drive enable	X2 pin 23
O 7	A axis Drive enable / Spindle coars stop	X2 pin 5
O 8	Lube Pump / A axis unclamp / Spindle fast stop	X2 pin 24
	O7 and O8 depends on rotary axis option or Rigid Tapping. Cannot have both options at the same time.	