



Operator Manual

This reference covers the following Vertical Machining Centers:

VMC-15
VMC-15XT
VMC-2016L
VMC-2216
VMC-2216HT
VMC-3016
VMC-3016L
VMC-3016HT
VMC-3020
VMC-4020
VMC-4020HT
VMC-4020A
VMC-4525
VMC-5020A
VMC-6030
VMC-8030



© Copyright 2000 Fadal Machining Centers.
All rights reserved.

Section 1: Power On/Off

Prestart Up Checks

- 1) Check the oil levels. The sight glass on the reservoir should be showing one inch or more of oil (See Figure 1-1). The spindle oiler may have oil in it for as long as six months. The way-lube oiler may run out of oil in one week.

Note: See maintenance manual for specifications on the way-lube and spindle oil.



Figure 1-1 Oil reservoir and sightglass.

- 2) Check the air pressure gauges to verify they are set to at least 80-100 PSI (See Figure 1-2). Air is used to change belt ranges in the spindle, orient the spindle, activate the tool in-out cylinder, and for the air blast during a tool change.

Note: Water is collected and released automatically by the air pressure regulator. If an excessive amount of water continually flows from the bowl, it is advisable to install an additional water trap in the air line going to the VMC.



Figure 1-2 Air pressure gauges.

- 3) Refill the flood coolant level to avoid depletion of coolant when programs are running. The coolant level should be filled to one inch from the top of the tank (See Figure 1-3).



Figure 1-3 Coolant tank fill level.

- 4) Check the spindle cooler reservoir once a month. Keep the fluid level filled to one inch from the top of the reservoir (See Figure 1-4).



Figure 1-4 Spindle cooler reservoir.

- 5) Power on the VMC. On the main power box, lift the knife switch to the upward position. Next, press the green power-on button that is located above the main powerbox (See Figure 1-5). The power button must be depressed for a minimum of two seconds.



Figure 1-5 Main power box and green power button.

Power On for System 97/99

The **Fadal** logo will display on the monitor screen of the pendant when the VMC is powered on (See Figure 1-6).



Figure 1-6 Screen display following power-on.

Automatic Cold Start

The *SYSTEM 97* program is designed to automatically find the origin of the machine (alignment of the x, y, and z-axis). The pendant screen displays the distance it takes for the machine to move to its origin.



IMPORTANT:

If the *SYSTEM 97* cold start initialization has been aborted, it is the responsibility of the operator to manually cold start the VMC. To manually cold start the VMC, use the following *wrong position* start-up procedure.

- 1) From the power-on screen display (See Figure 1-7), press the START button on the pendant.
 - a) The machine will move to the origin position specified on the screen display.
 - b) From the origin position, the machine will perform the cold start procedure automatically.

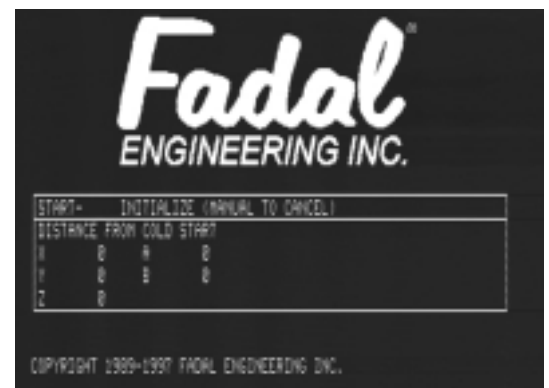


Figure 1-7 Power-on screen display.

2) Cold start is complete when the *last home position* screen is displayed (See Figure 1-8).

a) The *last home position* screen will not be displayed if an auto start-up program is stored in the memory.

b) The auto start-up program is named O9998 (See Auto Start-up program).

c) After the auto start-up program executes, the *last home position* screen is displayed.



Figure 1-8 Last Home Position screen.

3) Press the START button to begin motion to the last home position established by the set commands.

Wrong Power Off Procedure

If the VMC was improperly powered off, the *wrong power off* screen, shown in Figure 1-9, will display.



Figure 1-9 Screen display when VMC is improperly powered off.

WARNING:



Before cold starting the VMC, use *jog* to align the axes to the cold start marks. When using the *jog* mode before the CS command is used, the machine will go into emergency stop if the operator jogs to the physical limits. The physical limits are located .25 inches beyond the software limits.

Note: See also CS command in “Commands” section (Section 12).

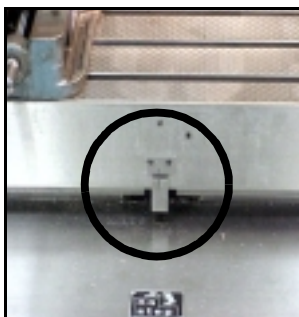
- 1) Press the MANUAL key until the *enter next command* prompt is displayed or, from the quick keys menu, press the *jog* key.
- 2) Press the JOG key and align each axes at the alignment marks within .05 inches of each other (See Figure 1-10).
- 3) Press the MANUAL key until the *enter next command* prompt is displayed. Type CS and press the ENTER key.
- 4) Wait. The machine will find the axis origin.
- 5) This procedure is complete when the move to *last home position* screen is displayed (See Figure 1-11).



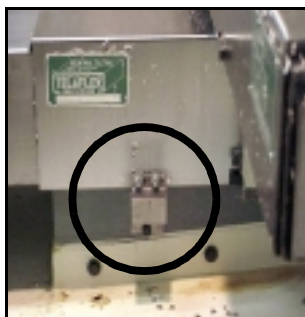
Figure 1-11 Last home position.

- a) The *last home position* will not display if an auto start-up program is in memory.

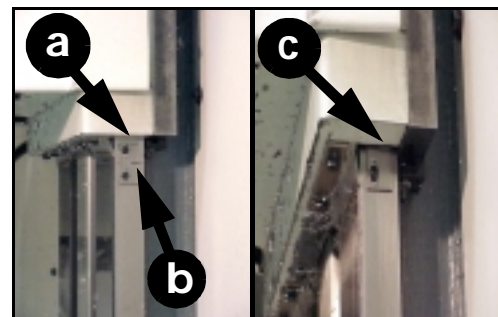
Figure 1-10 X, Y, and Z alignment points.



X-axis alignment.



Y-axis alignment.



a. Bottom of head casting. **b.** Z-axis alignment mark. **c.** Z-axis alignment.

- b) The auto start-up program is named O9998. (See auto start up program)
 - c) After the auto start-up program is finished, the move to *last home position* screen will be displayed.
- 6) Press the START button to begin motion to the last home position established by the set commands.

Auto Startup Program

The *Auto Startup Program* allows a user to enter into memory a program that will be used exclusively for power-on or warm-up. This program must be numbered O9998. After power-on, the program is activated and is ready to run if the operator presses the START button. When the program is finished, the control will automatically switch to the previously active program. In the past, if a warm-up or start-up program was used after power-on, it had to be accessed in the program library and activated. The operator then entered the auto mode to run it and then would switch back again. This action is now automatic and saves time for the operator.

- 1) Write the start-up program and store it in memory by heading the program with O9998. The O9998 number is what makes this an automatic program.
- 2) After the cold start or the initialization, this program will become active. Press the AUTO key to run the program when the *waiting* message is flashing. Press the MANUAL key to exit the program when the waiting message is flashing.
- 3) When the program is finished, with the waiting message flashing, the operator can run the program again by pressing the AUTO key or exit the program by pressing the MANUAL key.

Note: If the startup program ends with an M2, the operator will have an opportunity to run the program as many times as they need. If the program ends with nothing (no M0, M1, M2, or M30) when the program ends, the control will automatically switch to the previously active program.

Power Off Procedure

The Fadal CNC 88HS can be turned off at any time without loss of memory because of the battery backed memory board.

The power-on procedure states that the axis markers must be aligned before the CS command is used. This can be accomplished by jogging the

axes until the markers are in line. However, these markers can be aligned before the power is switched off, making the power-on procedure quicker.

POWER OFF:

- 1) From the command mode, type "SETCS" and press the ENTER key.
- 2) Type HO and press the ENTER key.
- 3) Wait for the light to flash and the word "waiting" to blink on the screen. Press the START button for the machine to move to the cold start position.
- 4) On the main power box, place the knife switch in the down (off) position to power off the VMC.

Note: For *SYSTEM 97*, follow the power-off procedure as outlined above or;

- 1) Press the MANUAL button until the control is in the command mode or the quick keys, edit, or functions menus are displayed on the screen.
- 2) Power off the machine. Place the knife switch, of the main power box, in the downward position.

(This page intentionally left blank.)

Section 2: Pendant

Video ON/OFF Switch

When the machine is idle for long periods of time, the video screen should be turned off. This is necessary when the machine is left on overnight. If the machine is left on overnight, press the EMERGENCY STOP button before leaving the machine.

Optional Stop Switch

Place the OPTIONAL STOP switch in the upward position to activate the optional stop code in the program. The control will display "opt" in the third box bottom line of the position display (See Figure 2-1).

- 1) The program must have an M1 code.
- 2) The program will stop on the line with the M1 code.
- 3) If the program does not have codes to turn the spindle on and coolant on again after the M1 code, the operator must turn these on manually.

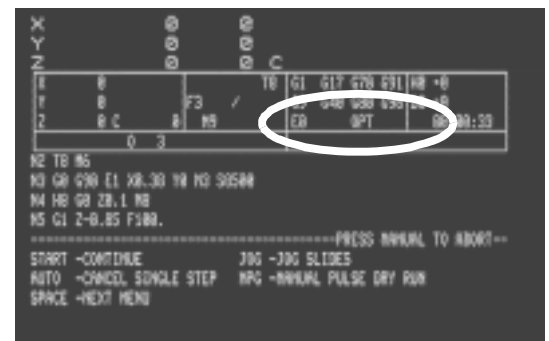


Figure 2-1 "Opt" is displayed when the OPTIONAL STOP switch is on.

Block Skip Switch

Place the BLOCK SKIP switch in the upward position to activate the block skip code in the program. The control will display "blk" in the third box bottom line of the position display (See Figure 2-2).

- 1) The program must have the "/" code before the line number.

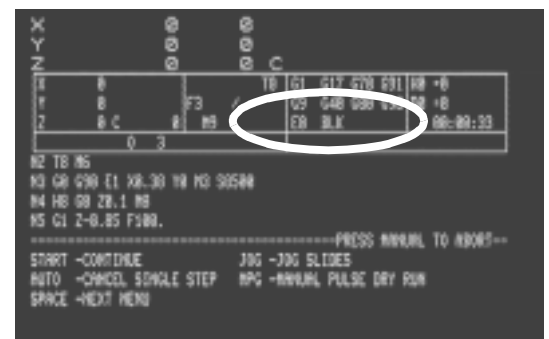


Figure 2-2 "Blk" is displayed when the BLOCK SKIP switch is on.

- 2) The program will skip the line with the "/" code.

Note: A sequence number with a "/" is used for block skip. The block skip feature cannot be used in the middle of the line. It must appear before the sequence number.

Example:

N4 Z.25 F20.

/N5 L101 * With the block skip on, the program will skip line five.

N6 X2.4 *With the block skip on, this line will be machined after line N4

To add a "/" to a sequence number:

- 1) From the page mode (PA) or edit mode, move the cursor to the line you want to edit.
- 2) Press the C key to change the line.
 - a) Press the / key.
 - b) Press the ENTER key two times.

Keylock

The key lock on the pendant is used to protect the program from unauthorized editing. When any of the commands that can alter the program are used, the message "command prohibited by key lock" appears. This feature also prohibits the PU command.

To prohibit program editing:

- 1) Put the key into the key slot on the pendant (See Figure 2-3).
- 2) Turn the key so the slot is in the horizontal position.
- 3) Remove the key and store it in a secured area.

To allow program editing:

- 1) Put the key into the key slot on the pendant.
- 2) Turn the key so the slot is in the vertical position.

Note: See also NOEDIT feature.



Figure 2-3 KEY LOCK used to protect unauthorized editing of programs.

Alternate NOEDIT Feature

The NOEDIT feature is used to prohibit the operator from editing or punching (downloading) the program in memory. The only thing allowed is to delete the program from memory. Any editing should be accomplished in an editor outside the machine and then resend the program to the machine.

1) On the line with the O word type a comment with the word NOEDIT within the first sixteen characters.

Example:

N1 O6 (NOEDIT FACE PLATE

Positioning Read Out Display

Figure 2-4 shows the sections and the purpose of each section used in the positioning read-out display. **a**: Position readout. **b**: Axis load or following error. **c**: Distance to go. **d**: Programmed RPM ("S9000"). **e**: Programmed turret location ("T1"). **f**: Actual RPM. **g**: Programmed feedrate ("F100"). **h**: Active "G" codes. **i**: Active tool offsets. **j**: Last executed "M" code. **k**: Actual feedrate ("95"). **l**: Optional stop on ("OPT"). **m**: Running buffer. **n**: Current program. **o**: Current fixture offsets ("E1"). **p**: Block skip on ("BLK"). **q**: Tool load compensation.

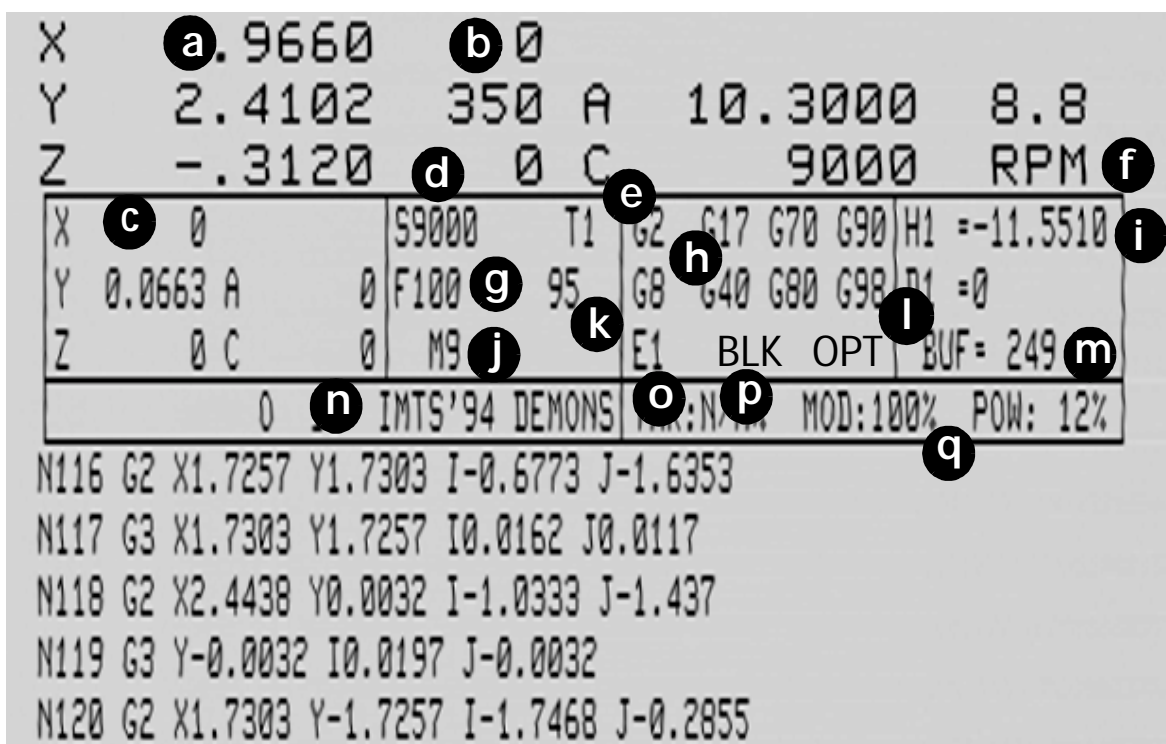


Figure 2-4 The upper portion of the monitor screen showing the *positioning read-out display*.

How to Display the Position Read-out:

- 1) Press the JOG button to see the positioning read-out.
- 2) From the command mode, type MD then press the ENTER button or press the MANUAL button to see the positioning read-out.
- 3) Press the AUTO button to see the positioning read-out.

The position read-out is showing the position of the tool in relation to the position set to zero by the SET (axis), SETH commands (or in relation to the cold start position before the SET (parameter) commands are used).

If a program contains the code G92, all positions are relative to the position preset by the G92 code.

When fixture offsets are used, the read-out is relative to the active fixture offset home position.

Note: When jogging in the metric mode .01 in.=.254 mm, .001 in.=.0254 mm, .0001 in.=.00254 mm. The minimal move in the metric mode is .00254 mm (even though the positioning read-out displays three places after the decimal).

Axis Load Display

The axis load amperage can be displayed for the AC Brushless axis systems. The axis load can be displayed directly to the right of each axis position displayed at the top of the screen. The default setting is axis following error.

- 1) To select load percentage, the operator must type SETP from the command mode.
- 2) Press the space bar two times to get to the last page of the machine parameter pages. Press the ENTER, D, or U key to move the cursor to the axis display selection.
- 3) Select load percentage so that the machine will display axis load. Then press the MANUAL key to exit the parameters page.

Manual Data Input Display

The MDI screen will display the modal codes in effect and the distance to go. This screen is similar to the positioning read-out screen in Auto (See Figure 2-5).



Figure 2-5 Manual Data Input.

Large Format Display

Depressing the = key during Auto will enable the large format screen (See Figure 2-6). To return to the normal screen press the = key again.

Note: On the 32MP controller, hold down SHIFT key and press the + key.

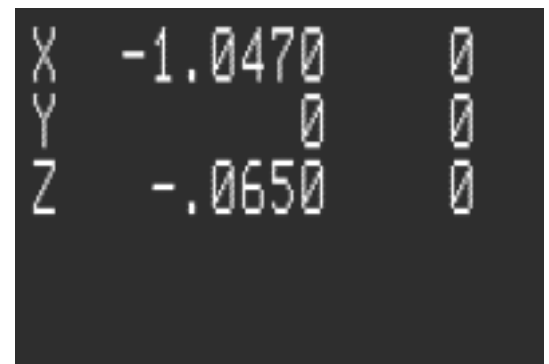


Figure 2-6 Large Format Display.

Modes

- 1) The control works in different modes of operation. Each mode of operation has a specific job to perform.

Example:

The *list* mode is used only for *listing* the program to the screen. The *jog* mode is used only for *jogging* the tool.

- 2) The *command* mode is the upper most mode and it acts like a dispatch center for all the other modes.
 - a) From the *command* mode the operator can go into the *menu* mode to look at the menu.
 - b) To go to another mode of operation, the operator must first go back to the *command* mode, before going to another mode.

- 3) The primary function of the MANUAL key is to get out of any mode of operation and get back to the *command* mode.
- 4) Two letter commands are used to get into other modes, and the MANUAL key is used to get back to the *command* mode.

Note: The MANUAL key is also used to enter the MDI mode.

Quick Keys Menu

There are three separate prompting menus within the machine control. They are the Quick Keys Menu, the Edit Menu, and the Functions Menu. These menus prompt the operator for input to perform machine commands. The Quick Keys menu is used for quick setup functions. Program editing is accomplished with the Edit Menu. The Functions Menu is utilized to perform machine tasks. These menus are activated or deactivated within the machine parameters. Use the SETP command, and the CMD MENU parameter, to turn the menus on or off. When the menus are off, the machine is operated by commands (See the Command Mode explanation on the following page). The Quick Keys Menu is selected by pressing the space bar from ENTER NEXT COMMAND.

Edit Menu

The Edit Menu is selected by pressing the space bar twice ENTER NEXT COMMAND, and allows editing or viewing the currently active program. This menu functions the same as the PA command editor (see PA Command).

Functions Menu

The Functions Menu is selected by pressing the space bar three times from ENTER NEXT COMMAND. The first twelve (12) lines display the currently active program. The bottom four (4) lines display the Functions Menu options.

These menu options are selected by number input. The operator may switch between the Function Menu, the Quick Keys Menu and the Edit Menu by pressing the SPACE bar.

Selection of menu options may display additional menu options. The top line of the subsequent menus displays the previous menu selected. Press the MANUAL key at any time to abort the process and return to the main menu.

The Users Manual has a detailed explanation of each function associated with these menus.

Using the Command Mode

To enter a command, the control must be in the command mode. The command mode is recognized by the words at the bottom of the screen, "enter next command". A list of accepted commands can be found in the Users Manual, chapter 8, or the Command Summary section of this chapter. The menu in the control also has a list of accepted commands. (See "Menu Mode")

The primary use of the MANUAL key is to get into the command mode. Below are examples of how the MANUAL key is used to access the command mode from various modes.

- 1) To go from the *manual data input* (MDI) mode to the *command* mode, at any time press the MANUAL key.
- 2) To go from the *jog* mode to the *command* mode, press the MANUAL key at any time.
- 3) To get into the *command* mode when "waiting-", "single step-", or "slide hold-" are flashing on the screen, press the MANUAL key.
- 4) To go from the *auto* mode to the *command* mode press the SLIDE HOLD key or the SINGLE STEP key, then press the MANUAL key.
- 5) To get into the *command* mode from the *list* mode, the *change* mode, the *sum* mode, the *insert* mode, the *learn* mode, the *tool changer* mode, or the PR procedure press the MANUAL key at any time.

Cursor in the Command Mode

The cursor is the flashing < symbol. When a key is pressed and this symbol is present, the letter, number, or symbol the key represents will be printed on the screen.

The only place on the screen the cursor will be found is in the lower left hand corner.

- 1) When a key is pressed the cursor moves to the right.

Example 1:

ENTER NEXT COMMAND <

ENTER NEXT COMMAND A < Press the letter A and the cursor will move to the right.

ENTER NEXT COMMAND ABC < Press the letter B and C the cursor will move to the right again.

- 2) The BACKSPACE key will move the cursor to the left, over the existing characters in the line.

Example 2:

ENTER NEXT COMMAND AB < From the Example above press the BACK SPACE once.

ENTER NEXT COMMAND A <C Press it again and the cursor will move to the left again. Notice the letter C is showing.

Note: The control will only recognize characters to the left of the cursor, when the ENTER button is pressed. From the Example above, only the letter A would be recognized if the ENTER button is pressed.

ENTER NEXT COMMAND AU< Press the letter U, it will be in the place of the letter B and the cursor will now cover the letter C.

ENTER NEXT COMMAND AU,50<,,1 If the ENTER button is pressed now, only AU,50 will be recognized.

Note: The control will not analyze what is typed until the ENTER button is pressed. Always look to see what is on the screen before pressing the ENTER key. (See DELETE KEY for more information on cursor movement.)

Using the Delete Key

The DEL key can be used in the command mode, change mode, insert mode, and manual data mode to delete all characters to the left of the cursor. Pressing the DEL key will not exit the current mode, but will allow the operator to retype the entire line.

Example:

ENTER NEXT COMMAND IN< (press ENTER)

IN

N1 GOG9OS10000M3X1.56Y-3.976E1@

The delete key was pressed here because it was noticed that the letter O was used for the number 0 in G0 and G90.

Notice the @ symbol where the DELETE key was pressed. The cursor has moved to the space below the N1 line. The control is still waiting for input on line N1. The control will only recognize characters to the right of the @ symbol and to the left of the cursor.

N1 GOG9OS1000M3X1.56Y-3.976E1@

G0G90S10000M3X1.56Y-3.976E1< (press ENTER)

N2<

Type the line correctly and press the ENTER button, the control will remain in the current mode.

The insert mode was used as an Example above, however, the DELETE button works the same way in the modes mentioned above, and it can be pressed as often as required.

The DEL key may also be used to delete a line of code from the program while in the PA mode. (See PAGE, USING THE PA COMMAND).

Using the Menu Mode

The *menu* mode is a list of accepted commands and their proper usage format. It is a summary form of the "command" section in the Users Manual. The *menu* mode is only for help when using the commands, it is not necessary to use the *menu* mode to use the commands.

- 1) From the *command* mode, type MU and then press the ENTER key to access the *menu* mode.
- 2) The first page of the menu is the directory (table of contents). This page has a list of all the commands, and the page number containing the explanations.
- 3) To get from page to page in the menu, three methods can be used.

Method 1 Press the ENTER key to page forward one page at a time. The listing will page forward each time the key is pressed.

Method 2 Press the BACKSPACE key to page backwards one page at a time. The listing will page backward each time the key is pressed.

Method 3 Type the page number of the page to jump to and press the ENTER key. The menu will go directly to the page number typed. This can be done at any time, when in the *menu* mode. Type the number one from any page of the menu, press the ENTER key, and the menu will jump to the directory page.

Note: To use a command, press the MANUAL key to exit the *menu* mode. The page being viewed in the *menu* mode will remain on the screen after exiting to the *command* mode.

(This page intentionally left blank.)

Section 3: Jog Mode

Jog Using the Manual Pulse Generator

- 1) At the "Enter Next Command" prompt, or any function menu mode, press the JOG button to get into the jog mode.
- 2) Select the axis to be jogged with the axis selector switch, located directly above and to the left of the manual pulse generator.
- 3) Select the increment to jog by with the increment selector switch, located directly above and to the right of the manual pulse generator.
- 4) To move the tool in a plus direction, turn the manual pulse generator in the clockwise direction.
- 5) To move the tool in a minus direction, turn the manual pulse generator in the counterclockwise direction.
- 6) The feed rate potentiometer has no effect on the rate of motion when using the manual pulse generator.

**WARNING:**

When using the keyboard to jog. The increment and axis selection at the keyboard will override the current settings of the selector switches. When going from the keyboard to the manual pulse generator for jogging, move both selector switches and confirm they are selected by looking in the lower left portion of the screen to see the active Jog mode status.

Note: The jog status is located in the lower left corner of the screen. Before jogging always look in the lower left corner of the screen to see the active jog status.

Note: The manual pulse generator has detentes for each increment on the hand wheel. These may be counted or calculated to make the desired moves.

Jog Using the Keyboard

- 1) At the *Enter Next Command* prompt or any function menu mode, press the JOG key to get into the *jog* mode.
- 2) Select the axis to be jogged by pressing the corresponding letter from the keyboard.
- 3) Select the direction to jog the tool using the + or - keys on the keyboard.
- 4) Select the increment to be jogged by typing H for increments of .010, M for increments of .001, and L for increments of .0001.
- 5) Each time the JOG key is pressed the tool will move in the direction and increment specified. The axis, direction, and increment are listed in the lower left hand corner of the screen.

Example:

If the setting in the lower left corner is X+H each time the jog key is pressed the tool will move in the X+ direction .01.

- 6) For continuous motion, depress the JOG key. The feed rate potentiometer will effect the rate of continuous motion.

Note: If the feed rate potentiometer is set to 0%, no motion will occur when using the JOG key.

WARNING:



When using the keyboard to jog. The increment and axis selection at the keyboard will override the current settings of the selector switches. When going from the keyboard to the manual pulse generator for jogging, move both selector switches and confirm they are selected by looking in the lower left portion of the screen to see the active Jog mode status.

Note: The jog status is located in the lower left corner of the screen. Before jogging always look in the lower left hand corner of the screen to see the active jog status.

Jog Using the Remote Manual Pulse Generator

- 1) At the *Enter Next Command* prompt or any function menu mode, select REMOTE on the pendant jog axis selector switch then press the JOG key to get into the *jog* mode.
- 2) Select the axis to be jogged with the remote axis selector switch, located on the remote manual pulse generator.
- 3) Select the increment to jog by with the increment selector switch, located on the remote manual pulse generator.
- 4) To move the tool in a plus direction, turn the manual pulse generator in the clockwise direction.
- 5) To move the tool in a minus direction, turn the manual pulse generator in the counterclockwise direction.
- 6) The feed rate potentiometer has no effect on the rate of motion when using the manual pulse generator.

**WARNING:**

When using the keyboard to jog. The increment and axis selection at the keyboard will override the current settings of the selector switches. When going from the keyboard to the manual pulse generator for jogging, move both selector switches and confirm they are selected by looking in the lower left portion of the screen to see the active Jog mode status.



Figure 3-1 Optional manual pulse generator remote.

Note: The jog status is located in the lower left corner of the screen. Before jogging, always look in the lower left corner of the screen to see the active jog status.

Note: The manual pulse generator has detentes for each increment on the hand wheel. These may be counted or calculated to make the desired moves.

Jog Using the Vector Jog for 5 Axis Head

This only functions when the 5 axis head parameter is selected in the SETP machine parameters page.

- 1) Select the z-axis either from the keyboard, pendant selector switch, or the remote selector switch.
- 2) Rotation CW or motion positive will move the z-axis in the positive direction.
- 3) Rotation CCW or motion negative will move the z-axis in the negative direction.

Single Step, Jog Away From

The operator can jog away from the current location when the machine is waiting to execute the next block, in the single step mode. The control will automatically return the axes to the original position if desired.

- 1) From the single step mode, when the single step message is flashing (See Figure 3-2), press the JOG key to get into the jog mode.
- 2) Use the keyboard or the manual pulse generator to jog away from the current location.

Note: The spindle can be turned off by pressing the SPINDLE ON/OFF key, and the coolant can be turned off by pressing the appropriate COOLANT key.

- 3) To get the control to return to the position jogged away from, press the MANUAL key to get out of the *jog* mode. Then press the START button.



Option 1 will cause the tool to return to the place where the JOG key was originally pressed. This movement will be at 150 inches per minute, moving the x-axis first, the y, a, and b-axes next, and the z-axis last. The control will then be ready to execute the next line in the program.

Option 2 will cause the tool to remain at this current position, the tool coordinate system will be shifted to this position, and execution of the program will continue.

Option 3 will cause the tool to remain at this current position, the tool coordinate system will remain at its original position, and execution of the program will continue.



Figure 3-3 Screen display after pressing JOG button.

WARNING:

If the spindle and coolant were turned off, they must be turn on again by the operator before the START button is pressed.

WARNING:

Before the START button is pressed, the z-axis must be jogged above the z-axis position jogged away from. Movement back is x and y-axis first, then the z-axis returns.



Slide Hold, Jog Away From

The operator can jog away from the position where the SLIDE HOLD button was pressed. The control will automatically return the axes to the original position if desired.

- 1) From the *slide hold* mode, when the slide hold message is flashing, press the JOG key to get into the *jog* mode.
- 2) Use the keyboard or the manual pulse generator to jog away from the position the machine is waiting in.

Note: The spindle can be turned off by pressing the SPINDLE ON/OFF key, and the coolant can be turned off by pressing the appropriate COOLANT key.

- 3) To get the control to return to the position jogged away from, press the MANUAL key to get out of the *jog* mode. Then press the START button.

The control will prompt the operator with the message shown in figure 3-3.



Figure 3-4 Options for manual slide hold.

Option 1 will cause the tool to return to the place where the Jog button was originally pressed. This movement will be at 150 inches per minute, moving the X axis first, the Y, A, and B axes next, and the Z axis last. The control will then be ready to execute the next line in the program.

Option 2 will cause the tool to remain at this current position, the tool coordinate system will be shifted to this position, and execution of the program will continue.

Option 3 will cause the tool to remain at this current position, the tool coordinate system will remain at its original position, and execution of the program will continue.

WARNING:

If the spindle and coolant were turned off, they must be turn on again by the operator before the START button is pressed.



WARNING:

Before the START button is pressed, the z-axis must be jogged above the z-axis position jogged away from. Movement back is x and y-axis first then the z-axis returns.



Waiting Stage, Jog Away From

The operator can jog away from the current location when the waiting message is flashing. The control will automatically return the axes to the original position if desired.

- 1) From the waiting stage, when the waiting message is flashing, press the JOG key to get into the jog mode.
- 2) Use the keyboard or the manual pulse generator to jog away from the current location.

Note: The spindle can be turned off by pressing the SPINDLE ON/OFF key, and the coolant can be turned off by pressing the COOLANT 1 key.

- 3) To get the tool to return to the position jogged away from, press the MANUAL key to get out of the *jog* mode. Then press the START button.

The control will prompt the operator with the message shown in figure 3-5.



Figure 3-5 "Waiting stage" options menu.

Option 1 will cause the tool to return to the place where the Jog button was originally pressed. This movement will be at 150 inches per minute, moving the x-axis first, the y, a, and b-axes next, and the z-axis last. The control will then be ready to execute the next line in the program.

Option 2 will cause the tool to remain at this current position, the tool coordinate system will be shifted to this position, and execution of the program will continue.

Option 3 will cause the tool to remain at this current position, the tool coordinate system will remain at its original position, and execution of the program will continue.

WARNING:



If the z-axis was jogged more than four inches positive, the z-axis will over travel at a tool change command. Stop and jog down below four inches before a tool change.

WARNING:



If the spindle and coolant were turned off, they must be turned on again by the operator before the START button is pressed.

WARNING:



Before the START button is pressed, the z-axis must be jogged above the z-axis position jogged away from. Return movement is the x-axis first, followed by the y, a, and b-axes, and finally, the z-axis.

(This page intentionally left blank.)

Section 4: Run Mode

Auto, Running a Program

The AU command will only execute the currently active program. To select a program, different than the current program, use the PR command. See "PROGRAM LIBRARY".

- 1) To run the currently active program press the AUTO button.
 - a) The control will prompt the operator to press the AUTO button again, or press MU for the run time menu (see RUN TIME MENU). Press the AUTO button again and wait for the WAITING message to appear on the screen.
 - (1) After pressing the auto button for the second time, the control begins to process the program. Press the START button at any time to interrupt the processing.
- 2) When the WAITING - message appears press the START button to execute the program.
 - a) Inspect the feed rate and spindle potentiometers.

FORMAT 1 The machine will return to the home position; the original SET(axis) position, prior to execution of the program. The control will start with the default codes in effect before reading the currently active program.

FORMAT 2 The machine will begin execution of the program from the current location. The codes and offsets which were modal prior to pressing the AUTO button will remain in effect until canceled by the program.

- 3) If the program ends with the code, M2 or M30, the operator can run the program over again at the completion of the program by pressing the START button when the WAITING - message appears.

Note: See SINGLE STEP and MANUAL PULSE DRY RUN. In the AUTO mode the feed rate potentiometer controls only the programmed feed rate moves.

Waiting

The waiting stage gives the operator an opportunity to continue or abort any automatic motion by the machine before it occurs.

The control is in the waiting stage when the message "WAITING-" is flashing in the upper left portion of the screen, just below the position read-out (See Figure 4-1).



Figure 4-1 The control in waiting mode.

- 1) Press the START button to continue the automatic motion.
- 2) Press the MANUAL button to abort the automatic motion.

Below is a list of places where the waiting message appears before continuing with automatic motion.

- a) After entering a yes (Y) to the move to home question.
- b) After entering the HO command.
- c) After the AUTO button is pressed.
- d) After entering the first line of program coding in the Manual Data Input mode.
- e) While executing the program at a line with a G4 P66000 coded. The G4 P66000 is a dwell coded for one minute and six seconds which immediately puts the control into the waiting stage.
- f) While executing the program at a line with an M0 coded.
- g) While executing the program at a line with an M1 coded, when the OPTIONAL STOP switch is in the upward position.
- h) While executing the program at the line with an M2 coded. The program is ready to run from the beginning again, at this point.

Changing Modes

The control works in inches or metric depending on the mode set at power on. The default mode (condition at power on) is inches. This parameter may be changed using the SETP command. (See DEFAULT PARAMETERS, USING THE SETP COMMAND).

- 1) To establish the metric mode.
 - a) From the command mode type SETME then press the ENTER button.
 - (1) In the first line of the program, use the code G71 to verify that the metric mode has been used. Use of this code is not essential

for running the program in the metric mode. It is only for verification. A message instructing the operator to use the SETME command will appear, if this mode has not been established.

Note: When jogging in the metric mode .01 in.=.254 mm, .001 in.=.0254 mm, .0001 in.=.00254 mm. The minimal move in the metric mode is .00254 mm (even though the positioning read-out displays three places after the decimal). Using .0001 increments the screen will read: .002, then .005, then .007, then .01 and so fourth.

- 2) To establish the *inches* mode.
 - a) From the command mode type SETIN then press the ENTER button.
 - (1) In the first line of the program, use the code G70 to verify that the inches mode has been used. Use of this code is not essential for running the program in the inches mode. It is only for verification. A message instructing the operator to use the SETIN command will appear, if this mode has not been established.

Manual Pulse Dry Run

The Manual Pulse Dry Run Mode is accessible from the Auto Mode or any time the WAITING or SLIDE HOLD messages appear on the screen.

- 1) Press the AUTO button to enter the auto mode.
- 2) Rotate the MPG hand wheel to the right in the positive direction, the operator will be prompted by the message "PRESS START TO ACTIVATE MPD OR MANUAL TO SKIP". Pressing the START button will cause the machine to enter the MPD Dry Run mode.
 - a) The more the hand wheel is turned the further the program is executed.
 - b) At this point in time if the hand wheel is rotated to the left in the negative direction, this will cause the machine to move backward in linear moves only. Forward motion in circles is allowed, but not backward.
 - c) The rate of execution is determined by two factors.
 1. By the rate of hand wheel rotation.
 2. By the feed rate used in the program. The higher the feed rate the faster the machine will move through the moves.
- 3) All moves require rotation of the hand wheel. This includes all the moves for the tool change.

- 4) When the hand wheel is not being rotated, the machine will enter the slide hold mode.
- 5) To exit this mode, press the START button or the AUTO button when the slide hold message is flashing. Doing this will start continuous automatic execution of the program.
- 6) To reenter this mode, rotate the hand wheel in the positive direction when the single step or the slide hold message is flashing, then press the start button.

Dry Run

The Dry Run procedure is used to prove (debug) the program. This procedure is usually performed by the programmer because changes to the program are usually made during the Dry Run procedure. It is referred to as the "Dry Run" because the material is not cut. This is called, "cutting air."

(See also RUN TIME MENU for Dry Run modes.)

- 1) Remove the part from the fixture.
- 2) Place all clamps in their approximate location and tighten to prevent movement.
- 3) While using Dry Run it is recommended that the SINGLE STEP mode be used.(See SINGLE STEP).
- 4) From the command mode type AU,,,3 then press the ENTER button. After pressing the ENTER button press the SINGLE STEP button.

The auto (AU) command AU,,,3 will run the program from the beginning to the end using Dry Run mode option number three. Dry Run option three will run the program and disregard the programmed feed rates. The program will run all feed rate moves at 75 IMP and all rapid moves at 300 IPM. In the single step mode, the feed rate potentiometer will control all feed rate moves and rapid moves. Use the feed rate potentiometer to slow the moves down if necessary. Also use the SLIDE HOLD button when needed.

(See also SLIDE HOLD, SLIDE HOLD, JOG AWAY FROM, and SINGLE STEP, JOG AWAY FROM)

Note: The Dry Run mode can be used in combination with a mid-tape (program) start. (See also MID-TAPE (PROGRAM) START)

Example:

AU,120,,3 This will use dry run option three and mid-tape start option zero.

AU,120,,3,1 This will use dry run option three and mid-tape start option one. (direct block start).

Single Step

The program can be executed one line (or block) at a time, using the SINGLE STEP and the START button.

After the AUTO button or the auto (AU) command is used, to execute the program, the AUTO and the START buttons function similarly. It is only after the SINGLE STEP button is pressed that the START button functions differently than the AUTO button. After the SINGLE STEP button is pressed, every time the START button is pressed the next line in the program is executed.

- 1) Press the AUTO button to get into the Auto mode.
- 2) Press the SINGLE STEP button to get into the single step mode.
- 3) Press the START button to execute the next line in the program.

When the SINGLE STEP button is pressed before the "WAITING -" message appears, the "SINGLE STEP -" message appears. The SINGLE STEP button can be pressed in the waiting stage and the next time the START button is pressed the "SINGLE STEP -" message appears, indicating to the operator the control is in the single step mode.

Note: In the single step mode the feed rate potentiometer has control of the programmed feed rate moves as well as the rapid moves. When in the auto mode the feed rate potentiometer controls only programmed feed rate moves. When the feed rate potentiometer is turned to zero, all machine motion stops, until it is turned up, then motion resumes.

Note: From the single step mode, press the AUTO button to discontinue the SINGLE STEP mode.

Note: From the single step mode, press the MANUAL button to return to the command mode.

Slide Hold

Tool motion can be stopped by using the SLIDE HOLD button. When this button is pressed, the control keeps the tool in the position where the tool is located, until the START button is pressed, at which time the remainder of the programmed move is made.

- 1) During programmed moves (in MDI mode or AUTO mode) press the SLIDE HOLD button.

The slide hold mode can be recognized by the message "SLIDE HOLD-" flashing in the lower left hand portion of the position display.

- 2) Press the START button to get out of the slide hold mode and complete the move.
- 3) When in the slide hold mode, the MANUAL button can be pressed to go into the command mode.

Note: If there is time, use the SLIDE HOLD button before using the EMERGENCY STOP button. The table may coast, when the EMERGENCY STOP button is used, but it will stop immediately when the SLIDE HOLD button is used. To stop the spindle when the SLIDE HOLD button is pressed, press the SPINDLE ON/OFF button.
(See also SLIDE HOLD, JOG AWAY FROM)
(See also SPINDLE, USING THE SPINDLE ON/OFF BUTTON)

Note: When the START button is pressed after the SLIDE HOLD button, the feed rate pot will control the remainder of the current move even if it is in rapid. The next program move will continue at either rapid or feed rate, whichever is programmed.

**Mid-Tape
(Program) Start**

Using the mid-tape start option of the auto command (AU, FROM, TO, DRY RUN OPTION, MID-TAPE START OPTION), the program can be started from any block. The options available are explained below.
(See also PA (page) command)

Option 0: When this option is used, the control will process the program up to the block specified by the first parameter, and start the search for vital programming information from line N0.001. This will result in the machine moving to the previous X, Y, Z A and/or B position specified in the program. The axis will move in the order the program was written. If the program specified the spindle on, coolant on, tool length and/or

fixture offset applied, and cutter radius compensation applied, these will all be in effect when the automatic cycle begins. Likewise if any other vital information has been programmed.

Example 1:

To use **Option 0** type AU,#,,,0 then press the ENTER button (where # is the block number to begin execution).

AU,50 <ENTER> Search starts from the beginning, and execution starts at N50.

Execution of the program will not begin until the message "WAITING -" appears on the screen, and the START or AUTO button is pressed. If the SINGLE STEP button is pressed before the message "WAITING" appears, the message "SINGLE STEP" will appear on the screen and the operator can single step the program by pressing the START button for each step. (See SINGLE STEP)

Note: Use SINGLE STEP for the first few moves so that the feed pot can be used to effect the rapid moves until the single step mode is canceled.

Option 1: When this option is used, the control will jump directly to the block specified by the first parameter, and prepare to begin execution at that block number. This is a Direct Block Start. All CNC coding necessary to run the subsequent blocks must be present at or after the specified first block to begin execution at.

Example 2:

To use **Option 1**, from the command mode type AU,#,,,1 then press the ENTER button (where # is the block number to begin execution).

AU,50, , ,1 <ENTER> Execution starts at N50. Direct Block Start

Execution of the program will not begin until the message "WAITING -" appears on the screen, and the START or AUTO button is pressed. If the SINGLE STEP button is pressed before the message "WAITING" appears, the message "SINGLE STEP" will appear on the screen and the operator can single step the program by pressing the START button for each step. (See SINGLE STEP)

Note: Use SINGLE STEP for the first few moves so that the feed pot can be used to effect the rapid moves until the single step mode is canceled.

Option N: This option is used to define a block in the program where all codes necessary to run the remainder of the program are present. The control will start reading from this line. When used in conjunction with the first parameter, the Start Block Line, the operation can be started deep into the program saving time normally used up by processing from line NO.001.

Example 3:

To use **Option N**, from the command mode type AU,#,,N then press the ENTER button (where # is the block number to begin execution, and N is the block number to begin the search).

AU,1350, , ,1300 <ENTER> - Search begins at N1300, and execution starts at N1350.

Execution of the program will not begin until the message "WAITING -" appears on the screen, and the START or AUTO button is pressed. If the SINGLE STEP button is pressed before the message "WAITING" appears, the message "SINGLE STEP" will appear on the screen and the operator can single step the program by pressing the START button for each step. (See SINGLE STEP)

Note: Use SINGLE STEP for the first few moves so that the feed pot can be used to effect the rapid moves until the single step mode is canceled.

Section 5: Emergency Stop

Emergency Stop

The emergency stop mode will “cut” the computer control to all axis motors, the spindle drive, and the tool changer. When the control detects a crash or overload situation, it places itself into emergency stop. Also, the operator can press the EMERGENCY STOP button to place the control in the emergency stop mode.

**CAUTION:**

If the machine is in motion when the Emergency Stop is activated, the spindle will free wheel for 30 seconds and the axis will coast to a resting position.

- 1) Press the EMERGENCY STOP button to put the control in the emergency stop mode (See Figure 5-1).

The message “EMERGENCY STOP - TAKE APPROPRIATE ACTION:” will appear on the screen when the control is in the emergency stop mode.

The appropriate action to take is:

- 1) Release the EMERGENCY STOP button if it has been pressed.
- 2) Press the JOG button which enables the amplifiers to reset if the problem has been resolved.

Note: It is not necessary to move the axes to their Cold Start position or establish the part home again. Sending the machine to Part HOME is recommended.



Figure 5-1 Emergency stop button.

```
ENTER NEXT COMMAND
EMERGENCY STOP - TAKE APPROPRIATE ACTION
CHECK EMERGENCY STOP SWITCH AND MOTOR OVERLOAD RELAYS
PRESS THE JOG KEY TO RESET THE EMERGENCY STOP
ENTER NEXT COMMAND
```

Figure 5-2 Screen display in emergency stop mode.

If the message:

"JOG AXES TO HOME POSITIONS, THEN ENTER THE CS COMMAND" appears, continue with the following steps.

To move the axes back to their COLD START home positions:

- 1) Press the manual key until the ENTER NEXT COMMAND prompt is displayed.
- 2) Press the jog key and align each axis at the alignment marks within .05 of each other.
- 3) Press the manual key until the ENTER NEXT COMMAND prompt is displayed, type CS then press the enter key.
- 4) Wait, the machine will now find the axis origin.
- 5) This procedure is complete when the move to last home position screen is displayed.
- 6) Press the start button to begin motion to the last home position established by the set commands.

Section 6: Formats

Formats

From the SETP Command the control has the option to use either:

FORMAT 1 Format 1 is FADAL style programming and machine operation. Format 1 has been uniquely designed to have the CNC control reset before restarting machine operation. When the CNC control is in Format 1 the programmer may program with the minimal of commands. This helps as an aide to the user for programming, setup and operation. Some codes are unique for FORMAT 1 and FORMAT 2. Listed in this chapter is the Examples for each formats programming style.

FORMAT 2 Format 2 allows a 6M/ 10M /11M style programming and machine operation. Format 2 follows the design to allow the programmer to completely command the CNC control. All resets therefore must be done by the programmer in the program and machine operation. Some codes are unique for FORMAT 2 and FORMAT 1. Listed in this chapter are the Examples for each formats programming style.



WARNING:

When the control is in FORMAT 2, it is mandatory to program all preparatory codes!

The SETP command is usually used by the factory and by the setup person who installs the machine. It can also be used by the end user to establish programming, baud rate, and output preferences.

Note: The factory set parameters can be found on the inside of the pendant door.

Changing Parameters

- 1) From the command mode type SETP then press the ENTER button.
- 2) The menu prompt at the bottom of the screen defines the use of the keyboard to move the cursor to the parameter to change. When the parameter to change has the cursor to it's left, the parameter choices will de displayed at the bottom of the screen.

- 3) Select the parameter choice by pressing the number corresponding to the choice and press the ENTER button. The parameter will display the change and the cursor will move to the right.
- 4) Press the MANUAL button to exit and save the parameter setting mode.

Saving Parameters

The PU,4 command is used to output the machine parameter list through the RS-232 port.

To output parameter information:

- 1) Get a computer ready to receive information through the RS-232 port.
- 2) Type the PU,4 command then press the ENTER button.

To receive parameter information:

- 1) Type the TA,1 command and press the ENTER button.
- 2) Send the file the parameter information was saved under to the machine.

Note: The parameters are saved in a file that uses the SP command for parameter data.

**FORMAT 1
Parameter Settings**

The following series of screens represent those displayed upon entering the SETP command. The parameters FORMAT, DEFAULT: G0, DEFAULT: G90, IMM. FIXED CYCLE: and CRC MODE: should be set as shown. All other parameters should be set as they are from the factory (See Figure 6-1A, 6-1B, and 6-1C).

```

                                     PAGE 1 OF 3
*FORMAT: 1      BAUD RATE: 2400      SPINDLE AFTER MG: NO
AXES: X,Y,Z     TRAVEL: X40 Y20 Z20  PENDANT: MOUNTED
DEFAULT: G0
DEFAULT: G90
DEFAULT: G17    TOOL CHANGER CAP: 21  IMM. FIXED CYCLE: NO
RPM FACTOR: 5   SPINDLE TYPE: 10,000  ORIENTATION FACT: 10

THERE ARE TWO PROGRAM OPERATION FORMATS AVAILABLE
DO YOU WANT MAXIMUM GMB/10M/11M COMPATIBILITY?
* 1) NO, ORIGINAL FADAL STYLE - FORMAT 1
  2) YES, FORMAT 2

P-PAGE ENTER-RT BACKSP-LT U-UP D-DN MAN-EXIT OR SELECT NUM. (
```

Figure 6-1A

```

                                     PAGE 2 OF 3
*DEFAULT: INCH  MG=FLOOD  MT=MIST    3 PHASE > 5% LOW: NO
PU FORMAT: FILE
CRC MODE: M96   BINARY BUFFERS: 255  MG TORQUE/RIGID: NO
SPINDLE OFF: NO TURRET FACTOR: 1      CND MENU: SPACE
PALLET: NO      GAIN: 100              RAMP: 100
ASPECT: 65      TIMERS: OFF            OVERLOAD: 2

ENTER THE DEFAULT VALUE
* 1) INCH
  2) METRIC

P-PAGE ENTER-RT BACKSP-LT U-UP D-DN MAN-EXIT OR SELECT NUM. (
```

Figure 6-1B

```

                                     PAGE 3 OF 3
*SCREW: #2      IPM: 400      XYZ RAMP: 160
Z TAP GAIN: HI  VECTOR: NO    AXIS DISPLAY: LOAD
AUTO BRAKE: YES A-PALLET: A-AXIS  B-PALLET: B-AXIS
5 AXIS HEAD: NO GB DETAIL: 21  FEEDBACK: YES

SELECT THE SCREEN PITCH TYPE?
1) INCH PITCH
* 2) 8/10 MILLIMETER PITCH

P-PAGE ENTER-RT BACKSP-LT U-UP D-DN MAN-EXIT OR SELECT NUM. (
```

Figure 6-1C

FORMAT 2 Parameter Settings

The following series of screens, shown in Figures 6-2A, 6-2B, and 6-2C, represent those displayed upon entering the SETP command. The parameters FORMAT:, DEFAULT: G1, DEFAULT: G91 N-WORDS ORDERED:, TOOL TABLE: and CRC MODE: should be set as shown. All other parameters should be set as they are from the factory.

```

                                     PAGE 1 OF 3
FORMAT: 2      *BAUD RATE: 2400    SPINDLE AFTER M6: NO
AXES: X,Y,Z    TRAVEL: 1140 120 220  PENDANT: MOUNTED
DEFAULT: G1
DEFAULT: G91
DEFAULT: G17    TOOL CHANGER CAP: 21
RPM FACTOR: 5   SPINDLE TYPE: 10,000  ORIENTATION FACTR: 10

ENTER DEFAULT BAUD RATE (THE RATE AFTER POWER-ON)
1) 110      5) 1200      9) 19200
2) 150      * 6) 2400    10) 38400
3) 300      7) 4800
4) 600      8) 9600

P-PAGE ENTER-RT BACKSP-LT U-UP D-DN MAN-EXIT OR SELECT NUM. (
```

Figure 6-2A

```

PAGE 2 OF 3
#DEFAULT: INCH  M6=FLOOD  M7=MIST  3 PHASE 1 5% LOW: NO
PU FORMAT: FILE  M-WORDS ORDERED: YES  TOOL TABLE: DIAMETER
CRC MODE: M96  BINARY BUFFERS: 255  HI TORQUE/RIGID: NO
SPINDLE OFF: NO  TURRET FACTOR: 1  CMD MENU: SPACE
PALLET: NO  GAIN: 100  RAMP: 100
ASPECT: 65  TIMERS: OFF  OVERLOAD: 2

ENTER THE DEFAULT VALUE
* 1) INCH
  2) METRIC

P-PAGE ENTER-RT BACKSP-LT U-UP D-DN MAN-EXIT OR SELECT NUM. <

```

Figure 6-2B

```

PAGE 3 OF 3
#SCREW: #2  IPM: 400  XYZ RAMP: 160
2 TAP GAIN: HI  VECTOR: NO  AXIS DISPLAY: LOAD
AUTO BRAKE: YES  A-PALLET: A-AXIS  B-PALLET: B-AXIS
5 AXIS HEAD: NO  G0 DETAIL: 21  FEEDBACK: YES

SELECT THE SCREW PITCH TYPE?
1) INCH PITCH
* 2) 8/10 MILLIMETER PITCH

P-PAGE ENTER-RT BACKSP-LT U-UP D-DN MAN-EXIT OR SELECT NUM. <

```

Figure 6-2C

FORMAT 1 Program Example #1

```

O1
M6 T1(.....TOOL #1 ID)
G0 G90 S##### M3 E## X####.#### Y####.####
H# M8 Z####.####
G4 P1000
[ Math work ]
G91 X####.#### L9 — the L9 repeats this line nine times
M5 M9
G0 G90 G80
Z0 G53

```

```

M6 T2(.....TOOL #2 ID)
G90 G0 S##### M3 E## X###.#### Y###.####
H# M8 Z###.####
G4 P1000
  [ Math work ]
M5 M9
G0 G90 G80
Z0 G53
E0 X0 Y0
M2

```

FORMAT 1 Program Example #2

```

O1
(SUBROUTINE SECTION)
L100 (SUB #1 FOR _____) — each sub begins with an
  L word, a sub number (1-99), and a two digit zero repetition (00).
M17
L200 (SUB #2 FOR _____)
  [ Math work ]
M17
L300 (SUB #3 FOR _____)
  [ Math work ]
M17 — this M17 marks the end of the last sub
M30 — the M30 marks the end of the program's sub section
( MAIN PROGRAM )
M6 T1(.....TOOL #1 ID)
G0 G90 S##### M3 E## X###.#### Y###.####
H# M8 Z###.####
G4 P1000
  [Math work ]
L101 — this line calls sub #1 one time
M98 P5 L1 — this line calls subprogram 5 one time
L215 — this line calls sub #2 fifteen times
G91 X##.#### L9 — the L9 repeats this line nine times
M5 M9
G0 G90 G80
Z0 G53
M6 T2(.....TOOL #2 ID)
G90 G0 S##### M3 E## X###.#### Y###.####
H# M8 Z###.####
G4 P1000
  [ Math work ]
M5 M9
G0 G90 G80

```

Z0 G53
E0 X0 Y0
M2

O5 (SUBPROGRAM #5 FOR _____)
[Math work]

M99 — ending this program with an M99 makes it a subprogram.

**Notes on FORMAT
1 style
programming:**

- 1) Each line requires a sequence number and will be generated by the CNC control. Sequence numbers use an N####.### word. Decimal points can be used to input lines between existing line numbers. The sequence numbers can be added when the program is typed at the computer, but the numbers are added automatically when the program is transferred to the control with the TA (tape input) command. So it is possible to type the program without line numbers and have the control add the line numbers automatically. The NU command is used to renumber the lines.
- 2) Decimals are required for all dimensions, feed rates, and angles.

Example:

X Y Z ###.#### (or ##.##### for higher precision)
F ####.##
A ####.###
B ####.###

The decimal point does not appear with dimensions, feed rates, or angles, the control will add them. For Example if an X1 is entered it is accepted as X.0001 (See also DECIMALS and SIGNS).

- 3) The O word on the first line is not required. The O word is used only for identification of the program in the program library for multiple program storage. The first sixteen characters of the comment with the O word will appear when the program library directory is displayed. This comment can be longer than sixteen characters but the remainder of the comment will only appear when the program is listed or printed. (See also the section on program library, the PR command.)
- 4) If the word NOEDIT is used within the first sixteen characters of the comment on the line with the O word, the program cannot be edit. The is the same function as using the KEY lock on the pendant.

FORMAT 2
Program Example

O# (COMMENT)
 (COMMENT)

G0 G17 G40 G70 G80 G90 H0 Z0

E0 X0 Y0 — this moves to the SETH position or cold start.

G92 X##.#### Y##.#### (or G54 X# Y# or E1 X# Y#)

M6 T1

(.....TOOL #1 ID)

G0 G90 S##### M3 X###.#### Y###.####

H# M8 Z###.#### D#

G4 P1000

[Math work]

M98 P4 L4 — this calls subprogram #4 four times

M98 P3 L27 — this calls subprogram #3 twenty seven times

G91 X##.#### G90 L9 — the L9 repeats this line nine times

M5 M9

G0 G90 H0 Z0

M6 T2

(.....TOOL #2 ID)

G0 G90 S##### M3 X###.#### Y###.#### E## (or G54-G59)

H# M8 Z###.#### D#

G4 P1000

[Math work]

M5 M9

G0 G90 H0 Z0

E0 X0 Y0 — use this line to cancel all fixture offsets

M30

O3 (SUBPROGRAM #3 FOR _____)

[Math work]

M99... ending this program with an M99 makes it a subprogram.

It is called with an M98 P3 L#. The L word is used for number of repeats.

O4 (SUBPROGRAM #4 FOR _____)

[Math work]

M98 P3 L1 — sub #4 is calling sub #3 one time (nesting)

M99 — ending this program with an M99 makes it a subprogram.

It is called with an M98 P4 L#. The L word is used for number of repeats.

**Notes on FORMAT
2 style
programming:**

- 1) The safe block is mandatory at the beginning of the program. This is also known as a preparatory code line.

```
O# ( COMMENT  
( COMMENT  
Z0 G53  
G0 G17 G40 G70 G80 G90 H0 E0 Z0  
G28 X0 Y0 Z0 (A0 B0 if needed
```

- 2) The G28, with the axis designated to move, will bring the machine to the zero position established by the cold start command (CS) or to the zero position established by the SETX, SETY, SETZ, or SETH commands and/or Fixture offsets, whichever was used last. The SET commands can be used to establish a zero position at any desired location. The program does not need to start from the machine home position. See also the section on establishing home position.
- 3) Fixed subroutines such as bolt hole pattern subroutines or pocket cutting routines, from other machines cannot be used on this control. See the users manual for information on using fixed subroutines from the CNC 88 control.
- 4) The SETP command allows certain changes to default codes, diameter or radius input, and M7 M8 preferences.
- 5) When typing any variable (R0,R1 - R9) always type the positive or negative symbol before the value. R1+.5 R2+2.34 R3+5.7124 R4-6.765. See VMC User's Manual Subroutines section for proper usage.
- 6) The H word will only pick up the tool length offset. The D word must be used to pick up the diameter or radius from the tool table.
- 7) Tool changes can be made from any XY position. The M6 code change will move the head to the tool position and make the exchange. The T word and the M6 can appear together, M6T# or T# M6, and usually do with this type of tool changer; however, they may appear on separate lines.
- 8) Each line requires a sequence number. Sequence numbers use an N####.### word. Decimal points can be used to input lines between existing line numbers. The sequence numbers can be added when the program is typed at the computer, but the numbers are added automatically when the program is transferred to the control with the TA (tape input) command. So it is possible to type the program without line numbers and have the control add the line numbers automatically. The NU command is used to renumber the lines.

- 9) Decimals are required for all dimensions, feed rates, and angles.

Example:

```
X Y Z ###.#### (or ##.##### for higher precision)
F ####.##
A ####.###
B ####.###
```

If decimals do not appear with dimensions, feed rates, or angles, the control will add them. For Example if an X1 is entered it is accepted as X.0001

- 10) The O word on the first line is not required. The O word is used only for identification of the program in the program library for multiple program storage. The first sixteen characters of the comment with the O word will appear when the program library directory is displayed. This comment can be longer than sixteen characters but the remainder of the comment will only appear when the program is listed or printed. (See also the section on program library, the PR command)
- 11) If the word NOEDIT is used within the first sixteen characters of the comment on the line with the O word, the program cannot be edited. This is the same function as using the key lock on the pendant.

**Viewing the
Software ID
Number**

Each software version has an ID number. Each time the software is updated write the new ID number in a convenient location. This number will be requested for some service or programming calls. The ID number is located on the last page of the MENU DIRECTORY.

- 1) From the command mode type MU (for the menu mode) then press the ENTER button.
- 2) Look at the first page of the menu. The page numbers of each command will be displayed on the screen.
- 3) Find the last page number displayed in the MENU DIRECTORY

Example 1:

ERROR NUMBERS 40-42
MESSAGES 43

- 4) Type the page number then press the ENTER button.

Example 2:

43 from the Example above

- 5) The *ID* number will now be displayed in the upper left corner of the screen.

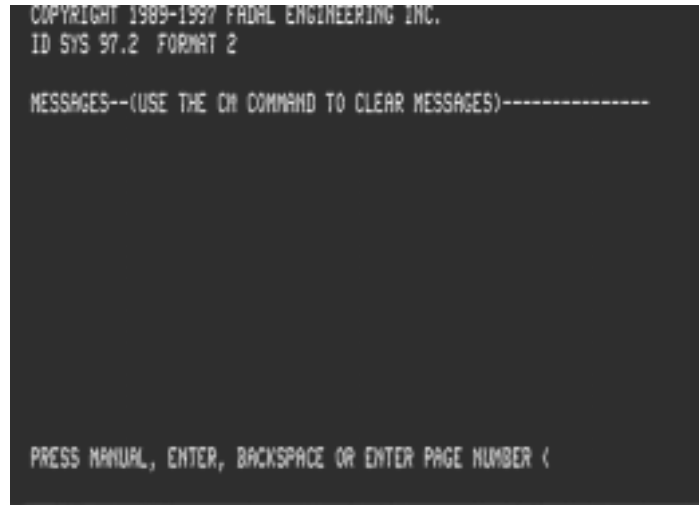


Figure 6-3 The software ID number is displayed in the upper lefthand corner of the display.

(This page intentionally left blank.)

Section 7: Spindle

Manually Loading and Unloading a Holder

Manual Operations

A tool can be manually loaded or unloaded into the spindle by using the TOOL IN/OUT button. The control must be in the COMMAND mode (ENTER NEXT COMMAND), AUTO mode or the MDI (manual data input) mode to use this button. The button will not activate the tool in-out piston if the spindle is rotating.

- 1) The tool holder must be held in the left hand with the thumb and the first finger grasping the holder below the "V" flange. No other fingers should have contact with the holder or the tool in the holder. The area below the "V" flange is called the safe zone (See Figure 7-1).
- 2) When unloading a tool from the spindle, grasp the tool in the safe zone and depress the TOOL IN/OUT button (See Figure 7-2). Keep the TOOL IN/OUT button depressed until the tool is completely out of the spindle.
- 3) When loading a tool into the spindle, grasp the tool in the safe zone and depress the TOOL IN/OUT button. Place the holder into the spindle after depressing the TOOL IN/OUT button, not before. The keys on the nose of the spindle must fit into the key ways on the tool holder flange. Release the TOOL IN/OUT button to lock the tool into the spindle.



Figure 7-1 Spindle safe zone area.



Figure 7-2 Grasp spindle in safe zone area.

Note: When loading a holder into the spindle, inspect the taper for chips and dents. Remove any chips or dents from the taper with a flat stone.

Note: Confirm the retention knob is securely tightened before placing the tool in the spindle.

Spindle, RPM Established from MDI

The spindle RPM can be established from the MDI (manual data input) mode. When the RPM is established the spindle can be turned on using the SPINDLE ON/OFF button and the SHIFT button.

- 1) Press the MANUAL button while in the command mode to enter the MDI mode.
- 2) Type the code S# (where # is the desired RPM. EX: S1400) then press the ENTER button.
- 3) The waiting message will now be flashing. At this point press the START button which will only execute a belt range change, if required, and register the RPM to memory.
- 4) Press the MANUAL button to exit the MDI mode.

VMC 2216, 4020, 4020HT, and 6030 RPM ranges are as follows:

Low range: 45 - 2500
High range: 2501 - 10000

VMC 20 RPM range is as follows:
Low range: 75 - 1250
High range: 1251 - 5000

VMC 2216, 4020, and 6030 with the 15000 RPM spindle has a single RPM range from 300 - 15000 RPM.

VMC 15 may have AUTO HI/LOW or MANUAL belt change. The RPM selected will automatically select the low or high range.

Note: Use the spindle override potentiometer to vary the RPM manually;
M3 = CW SPINDLE ON, M4 = CCW SPINDLE ON, M5 = SPINDLE OFF.

The last RPM setting used by the control is active when the spindle is turned on manually unless it is changed in the MDI mode.

Using the Spindle ON/OFF Button

Spindle On

After establishing the RPM from MDI, or at any time in any mode of operation, the spindle can be turned on manually by using the SPINDLE ON/OFF button and the SHIFT button.

- 1) Depress one of the SHIFT buttons, and while it is depressed press the SPINDLE ON/OFF button. The spindle will turn on in the forward direction.

Note: If the spindle does not turn on it may be that the SPINDLE ON/OFF button was pressed before the SHIFT button, or one of the buttons is not pressed properly. The buttons must be pressed directly in the center of each square.



CAUTION:

The spindle will turn on at the last programmed RPM.

Spindle Off

When the spindle is on, from any mode of operation, press the SPINDLE ON/OFF button alone, to turn the spindle off.
See also "SPINDLE REVERSED WHILE USING JOG".

Spindle, Loading a Holder from the ATC

When tool holders are in the ATC (automatic tool changer), they can be loaded into the spindle by using the MDI (manual data input) mode.

- 1) Type the command MD then press the ENTER button, or press the MANUAL button at the command mode, to get into the manual data input mode.
- 2) Type M6T# (where # is the turret location of the tool to be loaded into the spindle) then press the ENTER button.

Example:

M6 T1

- 3) The waiting message will flash. At this point press the START button to make the exchange.
- 4) Press the MANUAL button to exit the MDI mode.

**Spindle, Unorient
to Turn by Hand**

To turn the spindle by hand, for setup purposes, the spindle must be unoriented. This is not necessary on the VMC 15.

- 1) From the command mode type MD and then press the ENTER button.
- 2) Type S.2 and then press the ENTER button. This places the machine in High belt.
- 3) After the waiting message appears on the screen press the START button.
- 4) The spindle will turn on, then off, when the START button is pressed.
- 5) Press the MANUAL button.
- 6) The spindle will now be free to turn by hand. Steps one through five above, put the machine in the high belt range and the M3 turns the spindle on which releases the orientation.

Note: Release of the spindle orientation, without regard to the belt range, may also be accomplished by pressing the SPINDLE ON/OFF button.

Spindle, Orient

The spindle will orient to the same position each time it is instructed to orient. This position is the same position the spindle must locate to load tools from the turret. Orientation is accomplished by the following methods.

Method 1

- 1) Type the command MD then press the ENTER button, or press the MANUAL button at the command mode, to get into the manual data input mode.
- 2) Type the code M19 then press the ENTER button.
- 3) The waiting message will flash. At this point press the START button to execute the orientation.
- 4) Press the MANUAL button to exit the MDI mode.

Method 2

- 1) Type TC,1, at the command mode, then press the ENTER button. This is the command to open the ATC which will first orient the spindle, then open up the ATC.
- 2) Press the MANUAL button to return the turret to the parked position.

- Method 3**
- 1) Type the command MD then press the ENTER button, or press the MANUAL button at the command mode, to get into the manual data input mode.
 - 2) Type the code M6 then press the ENTER button.
 - 3) The waiting message will flash. At this point press the START button which will execute a tool change and orient the spindle.
 - 4) Press the MANUAL button to exit the MDI mode.

**Spindle Reversed
While Using Jog**

- 1) From the command mode type MD then press the enter button.
- 2) Type S# M4 (where # is the desired RPM). The waiting message will flash, then press the START button.
- 3) Press the START button for the spindle to start.
- 4) Press the SLIDE HOLD button.
- 5) Press the JOG button to get into the JOG mode. (See SLIDE HOLE, JOG AWAY FROM)

Note: Press the MANUAL button to get out of the jog mode, and press the MANUAL button again, in the slide hold mode, to exit the MDI mode.

(This page intentionally left blank.)

Section 8: Tool Changer

Turret (ATC), Open for Loading; Using the TC,1 Command

- 1) From the command mode type TC,1 then press the ENTER button. The ATC will open and remain open until the MANUAL button is pressed. If the Z axis is not at the cold start position the control will enter the waiting state. Press the START button to return to the cold start position then ATC will open.
- 2) Use the TURRET CW and the TURRET CCW buttons to rotate the turret.
- 3) Press the MANUAL button to close the ATC. This will bring the head down. If a tool is under the spindle, the tool will be loaded into the spindle and the ATC will move back to its home position.

Note: When loading tools into the turret to run a program, place tool number one in the turret and use the TURRET CW button to put the next tool in the turret until all the tools are placed in the turret. Rotate the turret until the first tool is in the number one position before setting the tool order. (See TURRET (ATC), ESTABLISH TOOL ORDER; USING THE SETTO COMMAND to establish tool order.)

Tool Changer Crash

- 1) Press the JOG button to get the turret into home position.
- 2) Move to the cold start position using either JOG or the power off procedure: Type SETCS and press ENTER. Type HO and press Enter, then press START.
- 3) Type CS and press the ENTER button. See POWER ON.
- 4) Use the TURRET CCW or CW button to rotate the turret until there is an open turret location directly opposing the spindle.
- 5) From the command mode type SETTO. This will establish this location as tool number one.
- 6) Type TC,1 to open the tool changer.

- 7) Rotate the turret with the TURRET CCW or CW buttons until tool number one is directly under the spindle, then press the MANUAL button to pick up tool number one.
- 8) Type SETTO to establish this location as tool number one.

**Turret Loading,
Putting Tools in
the Tool Changer**

When placing tools in the turret, the location of tool number one can be any convenient turret location. (See TURRET (ATC), ESTABLISH TOOL ORDER; USING THE SETTO COMMAND)

- 1) After the turret is open, move the turret around until there is an open spot directly in front of the open door.
 - a) Or take the tool out of the location directly in front of the open door. (See the TC,1 command, to open the turret door.)
- 2) Place tool #1 in the turret lid by aligning the clip with the V flange of the tool holder.
 - a) The tongue in the back of the clip must go into the key slot on the tool holder, for the tool to be seated properly.
- 3) Push the TURRET CW button, and then place the next tool in the turret lid.
 - a) The remaining tools are all placed into the turret lid, in sequential order. Press the TURRET CW button, after each tool is placed into the lid.
- 4) Use either the TURRET CW or TURRET CCW button to rotate the turret lid around so that tool number one, from the program, is directly under the spindle.
- 5) Press the MANUAL button to bring the head down over tool number one.
- 6) From the command mode type, SETTO then press the ENTER button. This will establish this location as tool number one. (See TURRET (ATC), ESTABLISH TOOL ORDER; USING THE SETTO COMMAND)

**Turret (ATC),
Establish Tool
Order; Using the
SETTO Command**

The turret is not numbered, so it is up to the operator to establish turret location number one. Then the control will sequentially number the remaining locations (CW viewed from the top). This allows for multiple tool groups from various jobs to be placed in the turret at one time. At the beginning of each job, the operator will rotate the turret and establish a new turret location number one for each new job.

- 1) Open the ATC and use either the TURRET CW or TURRET CCW button to rotate the turret to the number one location. (see TURRET (ATC), OPEN FOR LOADING; USING THE TC,1 COMMAND)
- 2) Press the MANUAL button to close the ATC. This will bring the head down. If a tool is in location number one, the tool will be loaded into the spindle and the ATC will move back to its home position.
- 3) From the command mode type SETTO then press the ENTER button. The SETTO command will establish this turret location as turret location number one.

Note: The RI command will also establish tool order when the tool table is cleared.

(This page intentionally left blank.)

Section 9: Coolant

Using the Coolant 1 and Coolant 2 Buttons

The coolant buttons can be used to turn the coolant on and off.

- 1) Press the COOLANT 1 button, when the flood coolant is off, to turn the flood coolant on.
 - a) Press the COOLANT 1 button, when the flood coolant is on, to turn the coolant off.
- 2) Press the COOLANT 2 button, when the mist coolant is off, to turn the mist coolant on.
 - a) Press the COOLANT 2 button, when the mist coolant is on, to turn the coolant off.

Note: M7 or M8 for coolant ON. M9 for coolant OFF.

Coolant-Through Spindle

If the VMC is equipped with the *coolant-through-the-spindle* option the power switch to the system needs to be in the *ON* position (See Figure 9-1).

- 1) Press the COOLANT 2 button, when the coolant-through is OFF, to turn the coolant ON.
- 2) Press the COOLANT 2 button, when the coolant is on, to turn the coolant off.



Figure 9-1 Power switch for machines with *coolant-through-the-spindle* option.

(This page intentionally left blank.)

Section 10: Commands

Table 1: Command Summary

COMMAND	COMMAND CODE	LINE COMMANDS DESCRIPTION
Auto	AU	AU,start from, to,dry run opt., direct block start
Backlash	BL	BL, axis #, amount at center, at - end, at + end
Change Device	CD	CD,baud rate, line feed opt., command echo
Change Program Blocks	CH	CH, from, through
Cold Start	CS	CS
Clear Messages	CM	CM
Copy Program Blocks	CO	CO, from, through, to just after
Delete Program Blocks	DE	DE, from, through
Diagnostic Mode	DI	DI (for maintenance personnel use)
DNC	DNC (DNCX)	DNC, video option, error option, dry run, start block number
Draw	DR	DR
Display Fixture Offsets	DF	DF
Display Tool Table	DT	DT, page number
Display Tool Time Table	DTT	DTT
Display Macro Variable Table	DV	DV
Fixture Offsets	FO	FO, number, (X amount), (Y amount)
Home All Axes	HO	HO
Insert Program Blocks	IN	IN, from, sequence increment

Table 1: Command Summary

COMMAND	COMMAND CODE	LINE COMMANDS DESCRIPTION
Learn Mode	LE	LE, first block number, increment, tool number
List Program Blocks	LI	LI, from, through
Macro Mode	MA	MA
Manual Data Input	MD	MD
Memory Free	ME	ME
Menu	MU	MU
New Program	NE	NE
Number Program Lines	NU	NU, increment for each number
Page Editor	PA	PA
Program Maintenance	PR	PR
Punch Program	PU	PU, data option, code option, TTY option
Reinitialize	RI	RI
Reset Cold Start Position	SETCS	SETCS
Reset Tool Changer	SETTO	SETTO
System Parameter Table	SETP	SETP
Set Length Offset	SL	SL, tool number, optional modification
Saved Parameters	SP	SP, parameter #, option #
Sum Program	SU	SU, display from, through, CRC option, list option
Survey	SV	SV
Tape (program) Input	TA	TA, device option, error option, add at end
Tool Changer Open	TC,1	TC,1
Tool Data Input	TO	TO, tool number, diameter, length offset
Utilities	UT	UT
Verification of Punched Tape	VT	VT, device option

**Command Use
Summary**

A list of the commands and their proper usage format is found in Commands section of the Users Manual. The commands and their usage format can also be found in the menu mode of the control (see Menu Mode). If an error is made when entering a command, the words "ERROR - ENTER COMMAND MU TO SEE THE MENU" appear directly above the words "ENTER NEXT COMMAND." It is not necessary to go to the menu mode after this message appears, this is only a suggestion.

If the message "PARAMETER ERROR" or "TOO MANY PARAMETERS" appears, the command was not entered properly and must be entered again. Always look to see if a message appears after entering a command, to see if the command entered, is accepted. To save time and confusion, it is suggested to look at the screen to see what was typed, before pressing the enter button.

AU, from, to, Dry Run Option, Mid-tape Start Option

Above is the AUTO command. This will be used as an Example to show how commands are used to instruct the machine to perform various functions.

- 1) The first two letters represent the command. Commands are letters only and are usually an abbreviation for the mode or job they represent.
- 2) The first comma separates the command from the parameter FROM. And the subsequent commas separate each parameter from the other. The commas are important and an error message will appear if the commas are not placed properly.
- 3) The parameters modify or control the command. For Example, the command above is for execution of the program, and the parameters control how the program is to be executed. Parameters are represented by numbers only. After each comma only a number can appear, or the message "PARAMETER ERROR" will appear.

Example 1:

AU,50 <ENTER>This is correct

AU,N50 <ENTER>This is incorrect

- 4) The first parameter comes after the first comma, the second parameter comes after the second comma and so on. If the third parameter is the only parameter used, three commas must appear, etc.

Example 2:

AU,,,3 <ENTER>

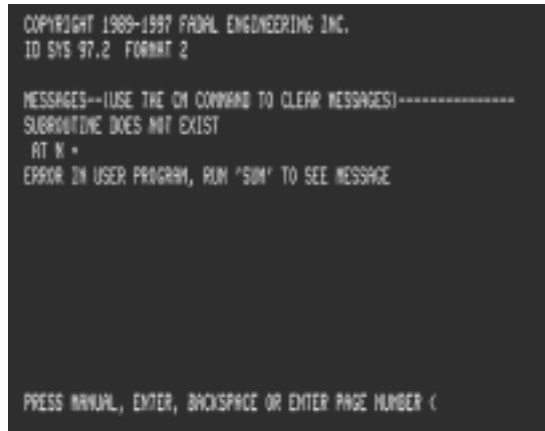
This example uses only the third parameter. The control will run the program from the beginning, to the end, under dry run option three.

Viewing Error Messages

Error messages displayed on the screen are retained on the new message page found in the help menu.

To access the message page in the help menu:

- 1) From the command mode type MU and press the ENTER key to enter into the help menu.
- 2) Type the page number for the message page and press the ENTER key. At this time messages are found on page 43.



```
COPYRIGHT 1989-1997 FADAL ENGINEERING INC.  
ID SYS 97.2  FORMAT 2  
  
MESSAGES--(USE THE CM COMMAND TO CLEAR MESSAGES)-----  
SUBROUTINE DOES NOT EXIST  
AT N =  
ERROR IN USER PROGRAM, RUN 'SDM' TO SEE MESSAGE  
  
PRESS MANUAL, ENTER, BACKSPACE OR ENTER PAGE NUMBER <
```

Figure 10-1 Message page.

To clear the messages from the message page:

- 1) From the command mode type CM and press the ENTER key.

AU COMMAND

Auto, Using the AU Command

The AUTO command is used instead of the AUTO key when a mid program start or a dry run is desired.

AU,From,To,Dry Run, Direct Block Start

- 1) The *From* parameter specifies the first block to be executed. If it is zero, the first program block of the main program is assumed. For mid program starts, all machine axes are automatically positioned to the location they would have been prior to the block specified, and all modal function codes specified before the starting block are automatically in effect (Spindle ON, Coolant ON, Absolute mode, etc.).
- 2) The *To* parameter specifies the block to end the program execution. If it is zero, the program is executed until an M2 or M30 (Format 2) end of program.
- 3) The *Dry Run* parameter specifies the dry run option. By entering a 1, 2 or 3, the program will be executed in the dry run mode. In this mode, all rapid moves are under the control of the feed rate override pot.

Option 1: Interpolation moves are made at programmed feed rate, point to point moves are made at 150 IPM.

Option 2: Interpolation moves are made at 150 IPM, point to point moves are made at 150 IPM.

Option 3: Interpolation moves are made at 75 IPM, point to point moves are made at 300 IPM.

- 4) The *Direct Block Start* parameter selection of 1 specifies that the execution begins directly and the control will not search for modal function codes specified before the block number in the first parameter, caution must be taken. Any number greater than 1 starts the modal code search at the block number specified by the fourth parameter.

BL COMMAND

Backlash, Using the BL Command

Backlash occurs when the tool changes direction. It is compensating for mechanical play in the ball screw, giving the screw an extra amount of turn getting the balls to touch the opposite lead of the thread before the table starts to move.

- 1) To see the backlash setting in the control, type "BL", in command mode, then press the ENTER button. (The factory setting is on the inside of the pendant door.)
- 2) To enter a backlash setting, go to command mode, type "BL", axis #, amount of backlash in center of table, amount at negative (-) side of tabel, amount at positive (+) side of table, press ENTER button.

Example:

BL,1,3,5,1 This will enter for the X axis a backlash setting of .0003 for the center of the table, .0005 for the left side and .0001 for the right side.

Note: Depending on the tool location on the table, one of the three backlash setting will be used.

The procedure below is an Example of determining the backlash compensation required for an axis. The X axis will be used for this Example. After the CS, Cold Start, command, do not return to the last home position.

- 1) From the command mode type "BL,1,0,0,0". This will zero the backlash compensation for the entire length of the X axis.
- 2) Place a surface indicator in a tool holder and jog the needle to Z-.1 and X.1 from the X positive side of a block in the center of the table. From the command mode type SETH then press the ENTER button. The face of the indicator should be toward the right side of the machine.
- 3) Insert the following program into memory:

```
N1 G91 G1 X-.1 F30.  
N2 G4 P66000  
N3 Z.25 G5  
N4 X-.1  
N5 X.1  
N6 Z-.25  
N7 G4 P66000  
N8 X.1 G5  
N9 M2
```

CD COMMAND

Setting Baud Rate Using the CD (Change Device) Command

When two devices communicate they must do so at the same rate. This rate is called the "BAUD RATE." Most communication software for computers will allow the operator to vary the BAUD RATE, depending upon the BAUD RATE of the device the computer is trying to communicate with. Some devices have a fixed BAUD RATE, so the control must be set to communicate at that fixed BAUD RATE.

To change the BAUD RATE at the control:

- 1) From the command mode type CD,# (where # is one of the BAUD RATE options)

Below is a list of BAUD RATE options.

- 1=110 baud
- 2=150 baud
- 3=300 baud
- 4=600 baud
- 5=1200 baud
- 6=2400 baud
- 7=4800 baud
- 8=9600 baud
- 9=19200 baud
- 10=38000 baud
- 11=57600 baud
- 12=115200 baud

Note: To establish the default BAUD RATE (the rate that the control will use without using the CD command) see DEFAULT PARAMETERS, USING THE SETP COMMAND.

Note: The command mode must be used to set the 57600 baud and 115200 baud communications rates.

CH COMMAND

Change Program Lines, Using the CH Command

Program coding can be changed in a line of the program by using the CH command. Program coding can also be added to a line in the program by using the CH command.

Example1:

To change a code in a line:

- 1) From the command mode type CH,# (where # is the number of the line to be changed) then press the ENTER button.
 - a) Type CH,30 then press the ENTER button.
N30 G0 G90 S700 M3 E1 X-3.743 Y- 2. The line as it is, in the program, will appear in the lower left hand portion of the screen, with the cursor on the line below.
N30 G0 G90 S700 M3 E1 X-3.743 Y-2.
 - b) Type the code or codes to be changed, then press the ENTER button.
S850 X3.743 Only the S word and the X word will be changed in the line.

Note: List the program to confirm the change was made correctly. (see LIST PROGRAM, USING THE LI COMMAND)

Note: If more than one G word or M word exists in the program, to change the similar words after the first word in the line, the first word(s) must be retyped.

Example 1a:

N30 G0 G90 S850 M3 E1 X3.743 Y-2.
G0 G91 To change the G90 to a G91 the G0 must be retyped so it will remain the same in the line.

Example 2:

To add a code to a line in the program:

- 1) From the command mode type CH,# (where # is the number of the line to be changed) then press the ENTER button.
 - a) Type CH,32 then press the ENTER button.

N32 G82 G99 R0.05 Z-.25 F10. The line as it is, in the program, will appear in the lower left hand portion of the screen, with the cursor on the line below.
N32 G82 G99 R0.05 Z-.25 F10.

- b) Type the code or codes to be added, then press the ENTER button.

P258 The P word will be added to the line when the ENTER button is pressed.

The line will look like this:

N32 G82 G99 R0.05 Z-.25 F10. P258

The control remains in the change mode until the MANUAL button is pressed or the ENTER button is pressed and no change has been made to the line.

This mode allows the replacement of letters or numbers (referred to as strings) with any new letters or numbers. This way each character in the line can be changed individually. Comments in the line can have a misspelled word and the operator can change only the misspelled word and not retype the entire comment. This ability is called FIND-AND-REPLACE.

Example 3:

Using FIND-AND-REPLACE:

- 1) From the command mode type CH,# (where # is the number of the line to be changed) then press the ENTER button.

- a) Type CH,78 then press the ENTER button.

N78 X5.05 Y-3.125* HOLLE D2 The line as it is, in the program, will appear in the lower left hand portion of the screen, with the cursor on the line below.

N78 X5.05 Y-3.125* HOLLE D2

<

HOLLE;HOLE Retype the characters to be changed (old string), type a semicolon and then type the characters as they should be (new string) then press the ENTER button.

N78 X5.05 Y-3.125* HOLE D2 The line as it is now in memory will appear on the 125;152 screen. Type the next characters to replace and press the enter button. In this case the numbers 125 will be changed to 152.

N78 X5.05 Y-3.152* HOLE D2 The new line will be displayed like this.

Example 4:

To remove (delete) characters from the line.

When it is desired to remove any character from a line, type the characters to be removed, a semicolon, then press the ENTER button.

N103 X-0.19 Y-0.38 J-0.19 G3

Type X; ENTER to remove X-0.19 from the line.

The CH command can be utilized to change the program in line number groups.

Example 5:

To change a group of lines.

1) From the command mode type CH,1st#,2nd# (where 1st# is the first line to edit, and 2nd# is the last line to edit) then press the ENTER button.

a) Type CH,5,15 then press the ENTER button.

N5 X5.05 Y.235 The first line will appear.

N5 X5.05 Y.235

X5.05;X5.025 Enter the change.

N5 X5.025 Y.235

N6 X6. The next line will appear.

When the ENTER button is pressed without making a change the next line will appear. If the last line to edit is on the screen, pressing the ENTER button will exit the CH mode. To exit the CH mode at any time press the MANUAL button.

CO COMMAND

Copy Program Blocks, Using the CO Command

Existing blocks (lines) in the program can be duplicated from one portion of the program to another using the CO command. This command will create the sequence numbers required to copy the blocks into other portions of the program.

- 1) From the command mode type CO, 1st #, 2nd #, 3rd # then press the ENTER button.
 - a) The 1st # represents the first block to copy.
 - b) The 2nd # represents the last block to copy.
 - c) The 3rd # represents the block number to copy the blocks just after. If a sequence number exists after this block, decimal numbers will be used for the duplicated blocks.

Example:

CO,12,26,56 This will copy lines 12 through 26 and place them just after line 56.

CO,50,78,.5 This will copy lines 50 through 78 and place them just after line .5.

CO,39,50,42 This will copy lines 39 through 50 and place them just after 42.

CS COMMAND

Cold Start

The CS command will establish the machine zero position. For the Z axis, the CS command establishes the tool change position. The Z axis cold start position is the point where the tool turret is directly aligned with the V flange of the tool in the spindle. The CS command will also establish the axes limits. After the CS command is used the control will establish "software" limits. The operator can jog up to the software limits. The control will not allow further motion. Before using the CS command the machine will not stop at the axes limits. When using JOG before the CS command is used the machine will go into emergency stop if the operator jogs to the physical limits. The physical limits are located .25 inches beyond the software limits. (See also POWER ON/OFF)

DE COMMAND

Delete Program Lines, Using the DE Command

Program lines can be deleted by using the DE command.

Example 1:

To delete one line:

- 1) From the command mode type DE,# (where # is the line to be deleted) then press the ENTER button.
 - a) Type DE,62.01 then press the ENTER button. This will delete line number 62.01.
 - b) If the line number does not exist in the program the message PARAMETER ERROR will appear on the screen just above the command line.

Example 2:

To delete a number of lines from the program:

- 1) From the command mode type DE,1st #,2nd # (where 1st # and 2nd # are the lines to be deleted from and through) then press the ENTER button.
 - a) Type DE,73,85 then press the ENTER button. This will delete all lines from line 73 through 85 including 73 and 85.
 - b) If the line numbers do not exist in the program the message PARAMETER ERROR will appear on the screen just above the command line.



WARNING:

There is NO confirm message before the DE command (delete) is executed. If an error in line numbers is entered it will be deleted if the line numbers exist. See the PA mode, *Delete*.

DNC COMMAND

Direct Numerical Control, Using the DNC Command

When a program is longer than the memory capacity of the control, an extended memory board can be installed in the control. If the program is longer than the extended memory capabilities, the DNC mode can be used which will allow a computer to link directly with the control. As the control needs more information the computer will send more information to the control on demand. This requires DNC software in the computer.

- 1) The baud rate at the control must be set at the same rate of the computer. (see BAUD RATE, USING THE CHANGE DEVICE (CD) COMMAND)
- 2) From the command mode type DNC then press the ENTER button.
- 3) Start the DNC software at the computer.
- 4) At the control press the AUTO or START button to begin execution of the program.
- 5) Use the feed rate potentiometer to slow the rate of execution. The screen should have at least two program lines showing while the program is running. Hesitation will occur if the tool is moving fast enough to only have one line of the program showing on the screen. (For -2 software, see also G91.1 for high speed execution.)

Note: The CNC program in the computer does not need sequence numbers, the control will establish them. The CNC program in the computer can have sequence numbers, however they are not needed.

Note to programmers

See also, G91.1 in the User's Manual.

Direct Numeric Control, Using a Server Program

The server program works similar to a main program with subprograms. The importance of the server program is that it gives the programmer the ability to direct the program to other files rather than having one long DNC file. Using the server program also facilitates the operator when attempting a mid-program start. It helps because the server program is usually short in comparison to a normal DNC files. It increases the ability to identify the mid-program start point.

The server program is no more than a normal NC program with a line describing a path and file name to be executed. Return to the server program is automatic after execution of the called up subprogram file is complete. Mid-program searches only search through the server program.

Note: Comments in the subprogram files must appear on a line with program coding.

```
(SERVER FOR P/N 123454 RIGHT ANGLE FLANGE DETAIL
M6 T1 (TOOL #1, 3/4 BALL NOSE)
G0 G90 S2000 M3 E1 X0 Y0
H1 M7 Z3.
F30.Feed rates
C:\DATA\PROG1   Execute Program 1
Z3. G0
E2 X0 Y0   Fixture offsets
G51.1 X0   Mirror
G68 R0+45. X0 Y0   Rotation
C:\DATA\PROG1   Execute Program 1
G50.1
G69
Z3. G0
M5 M9
G0 G49 G90 Z0
M6 T2 (TOOL #2, 5/8 BALL NOSE)
G0 G90 S2300 M3 E2 X0 Y0
H2 M7 Z3.
G51.1 X0
G68 R0+45. X0 Y0
C:\DATA\PROG2   Execute Program 2
G50.1
G69
Z3. G0
E1 X0 Y0
C:\DATA\PROG2
M5 M9
G0 G90 H0 Z0
M6 T1
M2
```

FO COMMAND

Fixture Offsets, Using the FO Command

Fixture offset information can be inserted or changed using the FO command. (See also: FIXTURE OFFSETS, USING THE UT COMMAND)

To find and insert fixture offset information:

- 1) Use an edge finder or indicator to find the home position (X & Y) for each fixture. All of these locations must be distances from the position where the SET(axis) commands were used (X and Y home positions; see following note).

Note: When using fixture offsets a common procedure to follow is:

- a) After using the CS command at power on, remain at that position. (see POWER ON)
- b) Press the JOG button, turn the selector switch to jog the Y axis and jog the Y axis .200 from the positive axis limit. Do Not move the X or Z axes from the CS position.
- c) From the command mode type SETH and press the ENTER button. This will reset the home position for all axes.

Note: Fixture offset 0 could also be referred to as "Program Home"

- d) This position is known as fixture number zero. From this position establish all the distances to all the other fixtures.

For Example:

Distance from fixture 0 to fixture 1 X-12.3256 Y-9.5723

Distance from fixture 0 to fixture 2 X-4.0162 Y-9.5014

Distance from fixture 0 to fixture 3 X7.8743 Y-9.4902

- e) The advantage of having Fixture zero near the Y axis positive limit when the program ends and returns to this position is: a) The table is out and close to the operator making it easier to load parts. b) The coolant is dripping away from the fixtures.

- 2) From the command mode type FO,#,X distance,Y distance then press the ENTER button. (Where # is the fixture number.)

Example:

(From distances recorded above)

FO,1,-12.3256,-9.5723

FO,2,-4.0162,-9.5014

FO,3,7.8743,-9.4902

Note: The second parameter is for the X distance and the third parameter is for the Y distance. Do not include the letter X or Y. Parameters are numbers only.

- 3) Set all tool length offsets at fixture #1, even if the program is a multiple operation program. When moving from one fixture to the next the control will automatically compensate for the Z difference from fixture #1. For Example if tool #4 is only used at fixture #3, the tool length offset is still established at fixture #1. When the program instructs the tool to move to fixture #3, the difference in height between fixture #1 and fixture #3 is made and then all Z axis moves are relative to the gauge point of fixture #3.
- 4) Establish the differences in the Z axis between fixture #1 and the other fixtures.
 - a) Place an indicator in a tool holder.
 - (1) Jog the indicator down to the point of fixture #1.
 - (2) Zero the dial to the pointer.
 - (3) Write the position of the Z axis, from the screen, down on paper.
 - (4) Jog to the gauge point of the other fixtures until the pointer of the indicator is at the same zero dial reading. Note: the Z axis position from the screen.
 - b) Compare the Z axis positions for each fixture with the position of fixture #1. Subtract to find the difference between fixture #1 and each fixture.

Example 1:

Fixture Z offsets for

#1 Z-10.3457

#2 Z-10.3498

#3 Z- 8.8024

#2 10.3498

#1 -10.3457

= .0041 Fixture #2 is below #1 -.0041

#1 10.3457

#3 - 8.8024

= 1.5433 Fixture #3 is above #1 +1.5433

- 5) From the command mode type FO,#,,,Z difference then press the ENTER button. (Where # is the fixture number.)

Example 2:

(From work above)

FO,2,,,-.0041

FO,3,,,1.5433

Note: Because all Z differences are from fixture #1, there is no Z offset for fixture #1.

**Viewing the
Fixture Offset
Table**

From the command mode type DF then press the ENTER button.

**Changing the
Fixture Offset
Information**

From the command mode type FO, new X dimension, new Y dimension, new Z difference then press the ENTER button. (Where # is the fixture number.)

Example:

FO,#, new X dimension

FO,1,-12.3234 This changes only the X dimension.

IN COMMAND

Insert Program Lines, Using the IN Command

To insert program coding into the program currently in memory, or to start entering a new program into memory, use the IN command. Insertion can be between existing lines by using decimal line numbers. The control will accept 999 lines between two whole line numbers (i.e. N1, N1.001 - N1.999, N2)

When the control is instructed to insert over an existing line, the old line is deleted and the new inserted line takes its place. The original line will be lost.

Example 1:

To insert a new program:

- 1) From the command mode type NE then press the ENTER button.
 - a) This will delete the current program from memory.
 - (1) Press the letter Y (Yes) then press the ENTER button to delete the program.

Note: Save the program currently in memory before inserting a new program. (see OUTPUT, TO TAPE PUNCH OR COMPUTER)

- 2) From the command mode type IN then press the ENTER button. Inserting will begin at line number one and step sequentially by one.
 - N 1 This will appear in the bottom left corner of the screen.
 - a) Type the information desired for the line then press the ENTER button.

N 1(TOOL #1 1/2 INCH 2FL EM (CRC))

Press ENTER

N 2 and the control will display the next line, allowing for input on the next line.
 - b) To terminate the insert mode press the MANUAL button, or press the ENTER button.

Example 2:

To insert a line anywhere in the program:

- 1) From the command mode type IN,# (where # is a line number) then press the ENTER button.

Note: Line numbers can have decimal points. A line number of N .015 or N 2.5 for Example. N .015 is a line before N 1 and N 1.5 is a line between N 1 and N 2.

- a) Type IN,.001 then press the ENTER button.

N 0.0010 This will appear in the lower left hand portion of the screen. Type in the line and press the ENTER button.

N 0.0010 O12(P/N 67-37754 SK)

N 1.001 The next line will appear. Because no increment was give as the second parameter, the insert command used an increment of one.

- b) Press the MANUAL button, or the ENTER button, to terminate the insert mode.

Example 3:

To insert a number of lines anywhere in the program:

- 1) From the command mode type IN,#,increment (where # is a line number and increment is the amount to step by for each line) then press the ENTER button.

Note: Line numbers can have decimal points. A line number of N.015 or N2.5 is acceptable. N.015 is a line before N1 and N2.5 is a line between N2 and N3. Increments for sequence numbers can also have decimals. Using an increment of .001 will cause the numbers to step by .001: for Example N.001, N.002, .003 ... N.999.

- a) Type IN,62.01,.01 then press the ENTER button. This will allow inserting of 99 lines between line 62 and 63.

N 62.01 This will appear in the lower left hand portion of the screen. Type in the line and press the ENTER button.

N 62.01G82G99R0.05Z-.22F50.P24

N 62.02 The next line will appear. Type in the next line and press the ENTER button.

N 62.01G82G99R0.05Z-.22F50.P24

N 62.02L101

N 62.03 The next line will appear. Type in the next line and press the ENTER button.

- b) Press the MANUAL button, or the ENTER button, to terminate the insert mode.

LE COMMAND

Learn Mode (digitizing), Using the LE Command

The learn mode is used when the programmer needs to jog the tool and insert the tool position into the program currently active in memory.

The learn mode is commonly used to find the moves required to clean out pockets, or any other moves in areas not defined by the print. The programmer can program the moves along walls and run the program. Then using the jog mode to cut the remaining material, the learn mode will insert all these moves directly into the program.

- 1) For the first parameter of the LE command, determine the sequence number where the positions will be inserted into the program.

Example:

N50 X1.5 Y-2.

N51 X2.5 Z-.245 G1 F20.

N52 (DIGITIZE POCKET HERE)When writing the program this message was placed at this line to mark the place where to insert the moves.

N53 Z.1 G0

- 2) The second parameter of the LE command is the increment used for the sequence number steps when each position is inserted into the program. Be careful to use an increment that will allow plenty of positions. An increment of .01 is usually adequate, allowing for 99 positions.
- 3) The tool length offset, of the tool used when in the learn mode, must be in the tool table. All digitized Z axis moves will be in relation to the gauge point (Z0) of the part according to the tool number in the third parameter.

Note: All positions inserted into the program will be in absolute (G90).

Example:

Using the program above:

- a) From the manual data mode establish the proper RPM for the tool.
- b) Put the tool in the spindle.
- c) From the command mode type LE,52.01,.01,1 then press the ENTER button. This will start inserting positions at line N52.01, increment by .01 (the next line will be N52.02), and all Z axis moves will be in relation to the gauge point of tool number one.

- d) The message at the bottom of the screen will read "PRESS JOG TO CONTINUE OR MANUAL TO EXIT."
 - e) Press the JOG button.
 - f) Start the spindle using the SPINDLE ON/OFF button and the SHIFT key.
 - g) Jog to the first position for the tool to move to. (In the case above X2.5 Y-2. Z-.245, or approximately this point.)
 - h) Press the MANUAL button to get out of the jog mode.
 - i) The line number that will be inserted, is listed on the second line from the bottom. The line on the bottom with the cursor is allowing the operator to change the line above it, exactly like the change program line command. Press the ENTER button after making changes. (see CHANGE PROGRAM LINES, USING THE CH COMMAND)
 - j) The message at the bottom of the screen will read "PRESS JOG TO CONTINUE OR MANUAL TO EXIT."
 - k) Press the JOG button and move to the next position, then repeat steps h - k.
- 4) When all the positions are found press the MANUAL button.
- A technique for digitizing around bosses or radii, is to jog to a starting position where a radius is desired, and insert that position. Then jog to the end point of the radius, change the G1 to a G2 or G3, and add R0# (use radius designation, see users manual) to go around the boss or radius. Go around a boss in at least two or three arcs.

LI COMMAND

List Program

The control can list the currently active program to the screen by using the list command. The list mode is only for listing the program. All other editing functions are handled by other commands.

LISTING SPEED CONTROL: The speed (or rate) at which the control will list the program is controlled by pressing a number key. The 0 key will stop the listing, to start listing again press a number key 1 - 9. The slowest listing speed is 1 the fastest listing speed is 9. When the control has listed the entire program, it goes directly into the command mode. The control can start listing and stop listing as often as needed. Scrolling is forward, to see a program line that has scrolled off the screen, press the manual button and list the program again. The control starts listing at speed 5.

- 1) From the command mode type LI then press the ENTER button.

The program will start listing from the beginning of the program at speed 5.

The control can start listing from any sequence number, and stop listing from any sequence number by using the from and through parameters.

- 2) From the command mode type LI,# (where # is the sequence number to start listing from) then press the ENTER button.

Example:

LI,52 The program will begin listing at line 52.

- 3) From the command mode type LI,#,# (where the first # is where to start listing from and the second # is the last line to list) then press the ENTER button.

This will start listing the program from the First # to the second #.

Note: See also PAGE,USING THE PA COMMAND.

MD COMMAND

MDI, Manual Data Input

Manual Data Input is used to execute program coding, as it is typed in at the keyboard and the ENTER button is pressed. When coding is typed, and the ENTER button is pressed, the coding is not placed into memory, it is only executed.

Any function allowed in the currently active program is allowed in MDI except for cutter radius compensation (unless it is used in a subroutine and an H or D word is used).

The first time the enter button is pressed when in the MDI mode, the waiting message will appear. After executing the first instruction, whenever the ENTER button is pressed the coding is executed without going into the waiting stage.

FORMAT 1 The default codes are in effect until other modal codes are used. See the users manual for the list of default codes.

FORMAT 2 The last modal codes used will be in effect in the MDI mode.

Example 1:

Using MDI to change a tool:

- 1) From the command mode type MD then press the ENTER button, or press the MANUAL button at the command mode.
- 2) Type M6T# (where # is a turret location) then press the ENTER button.

Example 2:

M6 T5 This will do a tool change and pick up the tool in turret location five when executed:

- 3) The waiting message will flash on the screen, press the START button to execute the tool change.
- 4) Press the MANUAL button to exit the MDI mode or type the next program coding to be executed.

Example 3:

Using a subroutine in the currently active program, called up from MDI:

- 1) From the command mode press the AUTO button.
- 2) When the waiting message appears press the MANUAL button.
- 3) Type MD then press the ENTER button.

- 4) Type the required codes necessary to use the subroutine in the currently active program. (H#, S#, G#, F#, M#)
- 5) Call up the subroutine using the L word, the sub number and the amount of repetitions.

ME COMMAND

Memory Free, Using the ME Command

The ME command is used to determine the amount of memory available to the user. This information is presented in the form of a percentage. The amount of the percentage indicates the amount of memory free to load other user programs.

Note: As a reminder, the memory of the machine is not a proper program storage device. It is recommended to preserve the user programs on disk or tape. The memory of the machine is maintained by a battery on the memory board. This battery cannot prevent an operator from using a command to clear certain portions of memory.

NE COMMAND

Memory Clear, Using the NE Command

Using the NE command will clear from memory the currently active program only.

- 1) From the command mode type NE then press the ENTER button.
 - a) The question will appear on the screen.
"DO YOU WANT TO REMOVE PROG." ENTER Y OR N.

- (1) Press the letter Y then press the ENTER button to clear the program from memory.
- (2) Press the letter N then press the ENTER button to keep the program.

Note: This command clears the program from the machine memory completely. It does not retain the program in the program library. To use the program again it must be reloaded into the control.



WARNING:

Ensure that the program has been saved to the floppy disk or tape before using the NE command or the program will be lost.

NU COMMAND

Number Program Lines, Using the NU Command

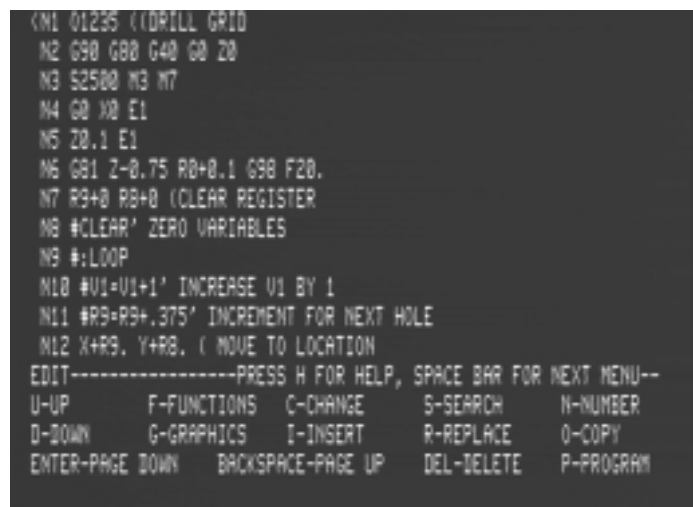
The lines in the program can be numbered by using the NU command. Each line in the program must be numbered. Because decimal numbers are allowed, the lines can be numbered in increments of 1 and insertion of lines can be accomplished by using decimal line numbers. It is easier to work with the program if the lines are numbered with whole numbers. It is suggested to number the lines before transmitting the program to a disk file or tape. Then get a new listing (hard copy) of the program.

- 1) From the command mode type NU,# (where # is the increment used to number the lines by) then press the ENTER button.
 - a) NU This will number the lines by ones.
 - b) NU,10 This will number the lines by tens.

PA COMMAND

Uses

The page command can be used to list, search, replace, number, copy, insert, delete, change, edit, run individual lines, and begin a direct mid-program start from the currently active program.



```
<N1 01235 (DRILL GRID
N2 G90 G80 G40 G0 Z0
N3 S2500 N3 M7
N4 G0 X0 E1
N5 Z0.1 E1
N6 G81 Z-0.75 R0+0.1 G98 F20.
N7 R9+0 R0+0 (CLEAR REGISTER
N8 #CLEAR' ZERO VARIABLES
N9 #:LOOP
N10 #U1=U1+1' INCREASE U1 BY 1
N11 #R9=R9+.375' INCREMENT FOR NEXT HOLE
N12 X+R9. Y+R9. ( MOVE TO LOCATION
EDIT-----PRESS H FOR HELP, SPACE BAR FOR NEXT MENU--
U-UP          F-FUNCTIONS  C-CHANGE    S-SEARCH     N-NUMBER
D-DOWN        G-GRAPHS    I-INSERT   R-REPLACE    O-COPY
ENTER-PAGE DOWN  BACKSPACE-PAGE UP  DEL-DELETE  P-PROGRAM
```

Figure 10-2 Page command screen.

PROGRAM EDITING USING PA

Decimals and Signs

- 1) Decimals are required for all dimensions, feed rates, and angles.
 - a) Each axis designation cannot exceed three digits to the left and four digits to the right or two digits to the left and five digits to the right with no more than seven digits total allowed.
INCHES: X Y Z ###.#### (or ##.##### for higher precision)
METRIC: X Y Z ####.### (or ###.##### for higher precision)
 - b) Each feed rate cannot exceed three digits to the left of the decimal and two digits to the right.
F ####.##
 - c) Each A axis angular move cannot exceed four digits to the left and three digits to the right.
A ####.###
 - d) Each B axis angular move cannot exceed three digits to the left of the decimal and three to the right.
B ####.###

If decimals do not appear with dimensions, feed rates, or angles, the control will add them. For Example if an X1 is entered it is accepted as X0.0001, F40 would be accepted as F0.4 and A100 would be accepted as A0.1

- 2) Signs must be typed after the axis designation or variable.

Example 1:

X-1. or R0-3.4

- 3) Positive signs are assumed for all designations but are required for R variables in format 2 mode to separate the variable from the value.

Example 2:

X1. Y2. Z3. A100. B15. R1+4.5 R2+5.6

Background Edit

From the AUTO mode the operator can change the program, offsets, and dry run options.

- 1) From the AUTO mode press the space bar.
 - a) The currently active program can now be edited.
 - b) Or by pressing 1 or 2 the dry run options or the offsets can be edited.

c) The 3 key displays the page editor keys used to edit the program.

2) Press the manual key to exit from the background edit mode.

Note: The edits made in the background edit mode are subject to the look-ahead / binary buffer. Any changes made will not come into affect until the current information in the buffer has been executed. If changes made to the program or offsets are already in the buffer, the changes will not be executed until the next time the program is executed.

To list the program:

- 1) From the command mode type PA then press the ENTER button. The Program will appear, as typed, on the screen.
 - a) The ENTER button is used to page down.
 - b) The BACK SPACE button is used to page up.
 - c) The U button is used to move the cursor up.
 - d) The D button is used to move the cursor down.

The cursor is the < symbol.

```
N73 M6 T2
N74 G0 G90 S8500 M3 E1 X-.5 Y-.42
<N75 H2 M7 Z-.395
```
 - e) The T button is used to move the cursor to the top of the program.
 - f) The B button is used to move the cursor to the bottom of the program.
- 2) The program may also be listed from a desired line number. Type PA,# (where # is the line number to begin the listing from) and press the ENTER button.

To insert between lines:

- 1) Move the cursor to the line to insert after.

Example:

```
N5 X2.374 Move the cursor to line N5 to insert  between lines N5 and
N6.
N6 Y1.275
```

- 2) Press the I button.
 - a) A new line number will appear.

Example:

N5 X2.374
N 5.1
N6 Y1.275

- b) After typing the new line press the ENTER button and the next line will appear.

Example:

N5 X2.374
N5.1 X2.874
N5.2
N6 Y1.275

Press the ENTER button at this point to exit the insert mode and return to the PA mode.

To search the program:

- 1) From the command mode type PA then press the ENTER button. The Program will appear, as typed, on the screen.
 - a) Press the S button.
 - b) The message "ENTER SEARCH:DEFAULT= : " will appear in the lower left hand corner of the screen.

Note: The search word refers to program coding. Words in a comment cannot be searched for.

- c) Type M6 and press the ENTER button.
 - d) The program will be listed, and the cursor will be placed next to the line that contains the search word. The line with the searched for information will be displayed as the first line on the screen unless it is on the last page of the program.

Example:

N57 M6 T1 This is the first line on the screen.
N58 G0 G90 S10000 M3 E1 X1.34 Y.4
N59 H1 M7 Z-.245
N60 G4 P1000

- 2) To search for the word again, press the S button and the message "ENTER SEARCH: DEFAULT=M6 : " in the lower left hand corner of the screen. Notice the default is M6.

Example:

From the preceding Example, if the ENTER button is pressed, the cursor will be placed next to the next line with an M6 in it because the default search is M6.

N73 M6 T2 This is the first line on the screen.

N74 G0 G90 S8500 M3 E1 X-.5 Y-.42

N75 H2 M7 Z-.395

To change a line:

- 1) Move the cursor next to the line to be changed.
- 2) Press the C button.
 - a) This will place the control in the change mode.
 - b) The line can now be changed, the same way the change command would work. (See also CHANGE PROGRAM LINES, USING THE CH COMMAND.)

To delete a line:

- 1) Move the cursor next to the line to be deleted.
- 2) Press the DEL button.
 - a) Two options will be available.
 - 1-CURRENT LINE this will delete the current line selected.
 - 2-MULTIPLE LINES Selecting this option allows the operator to designate the starting line and ending line to be deleted.

To run a single line from the program:

- 1) Place the cursor next to the line to be run by using the search method, or by using the cursor up or down method for cursor placement.
- 2) Press the A button.
 - a) Press the AUTO button.
 - (1) The WAITING message will start flashing on the screen.
 - b) Press the START button.
 - (1) Only the line that the cursor was next to will be executed. Then the control will return to the page listing.
 - c) The process can be repeated as often as needed.

Note: The process for running individual lines may be used when doing a mid-program start in format 2. Certain lines of coding may contain important information, needed for running the portion of the program in a direct mid-program start. For Example: In a situation where a fixed cycle must be initialized and then the 80th hole of a 100 hole pattern is the place where the direct mid-program will begin, the lines the where spindle is turned on and the tool length offset is called up and the line with the fixed cycle is used, can each be run individually. Then the cursor can be placed to the line of the 80th hole location and the direct start can begin from that location.

**To do a DIRECT
MID-PROGRAM
START:**

- 1) Place the cursor next to the line to begin running the program from. Use the search method, or the cursor up or down method for cursor placement.
- 2) Press the AUTO button.
 - a) Three options will be available:
 - 1-START AT BEGINNING the program will start at the beginning line number.
 - 2-START AT CURSOR LINE the program will start at the line selected by the cursor.
 - 3-SEARCH MODALS FROM & START AT CURSOR the program will be searched for applicable modal codes and start at line selected by cursor.
 - b) Press the AUTO button, and the program will begin processing according to the option selected.
 - c) After processing is completed the waiting light will flash.
 - d) Press the AUTO button to start the program.

PR COMMAND

Program Library, Using the PR Command

The program library is used to store different programs in memory, while one program is currently active. All editing, listing, and execution is done with the program currently active in memory. To work with any of the other programs, the PR command may be used to switch the currently active program with one of the programs in the library.

To differentiate one program from the other an O word in the first line of each program is needed. The O word must be the only coding, other than comments, in the line. The O word must be numbered as between O1 and O9999.

Example:

N1 O206(P/N 1234)

In the Example above the program has been given the number 206. The O word can have a comment in the line, which will show up to sixteen characters when option 2 (display program numbers) is used from the PR menu. This comment can be used to tag the program numbers with a part number or part name.

To select another program:

- 1) From "ENTER NEXT COMMAND" type PR then press ENTER the following menu will appear:

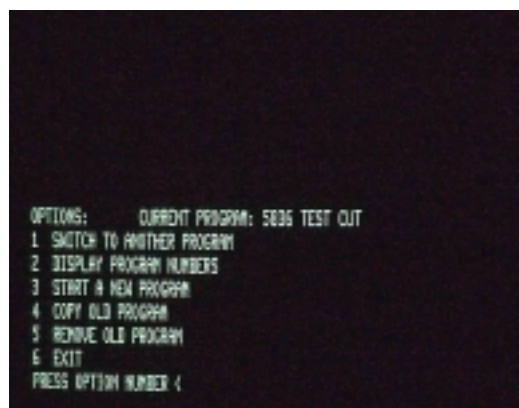


Figure 10-3 Display from PR Command

**To use the
program library
menu:**

- 1) Type PR then press the enter button. If the message appears "PLEASE PUT AN O-WORD AT THE FIRST OF THE CURRENT PROGRAM THE FOLLOWING PROGRAMS ARE IN MEMORY:", an O word must be inserted into the program as the first line of the program.
 - a) Use the IN command to insert the O word.
 - (1) Type IN,.001 and press enter.
 - (2) Type O# (where # is an unused program number) and press enter.
 - (3) Press the manual button.
 - (4) Type PR and press the enter button to use the program library menu.
- 2) When in the program library menu the words ENTER OPTION NUMBER appear an option number must be entered or the manual button can be pressed to exit this mode.
 - a) Option #1: This option is used to switch the currently active program with one of the programs in the program library. Before using option #1, use option #2 which will display the program numbers in memory. Then use the PR command again, after the control displays the programs in memory, to switch programs.
 - b) Option #2: This option will display all program numbers in memory. It will also display the comment placed with the O word. (see above)
 - c) Option #3: This option is used only when the IN command is going to be used to start a new active program, and if the currently active program has an O word in the first line and it is desired to keep the currently active program in the library. When the words appear ENTER PROGRAM NUMBER , enter only the program number, do not include the letter O. The control will display DUPLICATE NAME if the number entered for the program is the duplicate of a program in the library. The IN command can now be used to begin entering the next program.
 - d) Option #4: This option is used to copy an entire program in the program library (old program), and give it another program number (new program). This may be used for editing purposes.
 - e) Option #5: This option is used to delete a program from the program library (old program).

RI COMMAND

Memory Clear, Using the RI Command

The RI command allows the operator to completely clear the machine memory. The operator can choose to delete or leave in memory all fixture offsets and the tool table.

- 1) From the command mode type RI then press the ENTER button.
 - a) The first question to appear will read,
"DO YOU WANT TO ZERO TOOL TABLE? (HIT Y OR N, THEN ENTER)"
 - (1) Press the letter Y then press the ENTER button to clear all tool offsets.
 - (2) Press the letter N then press the ENTER button to keep all tool offsets.
 - b) The second question to appear will read, "DO YOU WANT TO ZERO FIXTURE OFFSETS? (HIT Y OR N, THEN PRESS ENTER)."
 - (1) Press the letter Y then press the ENTER button to clear all fixture offsets.
 - (2) Press the letter N then press the ENTER button to keep all fixture offsets.
 - c) The third question to appear will read, "DO YOU WANT TO REINITIALIZE MEMORY? (HIT Y OR N, THEN PRESS ENTER). This will clear all CNC programs in the program library and the currently active program."
 - (1) Press the letter Y then press the ENTER button to clear all program memory.
 - (2) Press the letter N then press the ENTER button to keep all program memory.

Note: Answering yes to the third question will cause the machine to ask for the CS command again. (See POWER ON)

USING THE SETTO COMMAND

Turret (ATC), Establish Tool Order; Using the SETTO Command

The turret is not numbered, so it is up to the operator to establish turret location number one. Then the control will sequentially number the remaining locations (CW viewed from the top). This allows for multiple tool groups from various jobs to be placed in the turret at one time. At the beginning of each job, the operator will rotate the turret and establish a new turret location number one for each new job.

- 1) Open the ATC and use either the TURRET CW or TURRET CCW button to rotate the turret to the number one location. (see TURRET (ATC), OPEN FOR LOADING; USING THE TC,1 COMMAND)
- 2) Press the MANUAL button to close the ATC. This will bring the head down. If a tool is in location number one, the tool will be loaded into the spindle and the ATC will move back to its home position.
- 3) From the command mode type SETTO then press the ENTER button. The SETTO command will establish this turret location as turret location number one.

Note: The RI command will also establish tool order when the tool table is cleared.

SETP COMMAND

Using the SETP Command

The SETP command is usually used by the factory and by the setup person who installs the machine. It can also be used by the end user to establish programming, baud rate, and output preferences.

Note: The factory set parameters can be found on the inside of the pendant door.

- 1) Move the machine to the cold start position if changes will be made to the parameter settings. The machine Cold Start must be accomplished when parameters are changed.
- 2) From the command mode type SETP then press the ENTER button.
- 3) The menu prompt at the bottom of the screen defines the use of the keyboard to move the cursor to the parameter to change. When the parameter to change has the cursor to it's left, the parameter choices will be displayed at the bottom of the screen.
- 4) Select the parameter choice by pressing the number corresponding to the choice and press the ENTER button. The parameter will display the change and the cursor will move to the right.
- 5) Press the MANUAL button to exit and save the parameter setting mode.

Note: If changes are made to any of the parameters, in most cases the CNC control will require a COLD START. Follow the power on procedure. (See POWER ON)

The following pages have the recommended format settings for the FADAL FORMAT 1 and FORMAT 2 6MB,10M,11M compatibility.

SET(AXIS) COMMAND

Position (PART), Establish; Using the SET(AXIS) Command

The CS command will establish the cold start position where X, Y, Z, A, and B are zero. This point is also known as machine zero. (see POWER ON)

The cold start position is used as a fixed reference point to establish the part home from. The SET(axis) commands are used to establish the part or program home location.

Establishing part home using an edge finder:

- 1) From the command mode type MD then press the ENTER button.
- 2) Now in the manual data input mode, type S1400 then press the ENTER button.
- 3) The waiting message will appear and the blue light will be flashing.
- 4) Press the START button, which will establish an RPM of 1400. (The spindle will not be on at this time.)
- 5) Press the MANUAL button.
- 6) Jog the edge finder close to the 1st edge (X or Y) of the part.
- 7) Start the spindle manually by depressing one of the SHIFT buttons as the SPINDLE ON/OFF button is pressed. The SHIFT button must be depressed first then the SPINDLE ON/OFF button can be pressed.
- 8) Jog the edge finder until it finds the 1st edge. And immediately press the MANUAL button.
- 9) From the command mode type SETX or SETY then press the ENTER button. This establishes the current X or Y position as zero.
- 10) Follow steps 6 - 8 for the other axes.
- 11) From the command mode type SETY or SETX for the remaining axis then press the ENTER button.
- 12) Jog the edge finder up, above the top of the part.

13) Jog the X and Y axes to correct for the radius of the edge finder.

14) Press the MANUAL button.

15) From the command mode type SETX then press the ENTER button. Then type SETY and press the ENTER button. This will correct the home position for the radius of the edge finder.

Example:

If the upper left hand (X- Y+) corner of the part is picked up, using a .200 diameter edge finder.

- a) Select the X axis and .01 increments.
- b) Jog the X axis to .1 by looking at the position read-out on the screen.
- c) Select the Y axis.
- d) Jog the Y axis to -.1 by looking at the position read-out on the screen.



WARNING:

The SETH command will establish the current position of all axes to zero (home). Do not use SETH unless all axes are at home.

**Establishing part
home using an
indicator:**

- 1) Remove the indicator from the spindle.
- 2) From the command mode type MD then press the ENTER button.
- 3) Now in the manual data mode, type S.2 M3 M5 then press the ENTER button.
- 4) The waiting message will appear and the blue light will be flashing.
- 5) Press the START button, which will unorient the spindle.
- 6) Press the MANUAL button.

Note: Steps 1 - 6 are to unorient the spindle and put the belts in the high range making it easier to turn the spindle.

- 7) Place a holder with an indicator, in the spindle.

- 8) Jog over and indicate the location boss, pin or hole.
- 9) Press the MANUAL button.
- 10) From the command mode type SETY then press the ENTER button.
- 11) From the command mode type SETX then press the ENTER button.

SETH COMMAND

Home Position (Part), Correction Using SETH

To make an adjustment to the home position use the following steps:

- 1) From the command mode type HO then press the ENTER button. The waiting message will appear, then press the START button. If the Z axis is below the Z axis home position, the Z axis will move home first, then the X and Y axes will move home. If the Z axis is above the Z axis home position the X and Y axes will move first, then the Z axis will move home.
- 2) Press the JOG button and jog the axes to the new correct location for the next run.

Example:

If the home position needs to be corrected by the value of X.0013 and Y-.0009:

- a) Select the X axis and by looking at the position read-out on the screen jog the X axis to .0013.
- b) Jog the Y axis to -.0009 by looking at the position read-out on the screen.
- 3) Press the MANUAL button.
- 4) From the command mode type SETH then press the ENTER button. This will correct the home position for the next run. The SETH command will establish the current position of all axes to zero (home).

SL COMMAND

Tool Length Offset, Setting

The point where the tools will be set, is called a **gauge point**. This is a common starting position for all the tools. This is where the programmer has established the Z axis zero position for the part program. (Not to be confused with Z zero at the cold start position.)

Commonly the programmer will choose a gauge point that is at the top of the part; at .100 above the top of the part; or at 1.0 above the top of the part.

The method of setting the TLO (tool length offset) described below is a two step method. The TLO is first set (inputting of the TLO into the tool table) above the gauge point. The TLO in the table is then corrected to its proper location.

Note: Use the RI command to clear the tool table.

- 1) Locate all tools specified for the program and load the tools into tool holders. Place these holders close to the machine.
- 2) Load tool #1 into the spindle using the TOOL IN/OUT button. (CAUTION: Read, "Spindle, Manually loading & unloading a holder")
- 3) Place a gauge block, of any available size, on the top of the part.
- 4) Press the JOG button and use either the hand wheel or the keyboard to jog the tip of the tool below the top of the gauge block.
- 5) Slide the block carefully up to the tool, and in .001 increments jog the tool up until the block slides under the tip of the tool.
- 6) Remove the block from under the tool and press the MANUAL button.
- 7) Type the command SL,# (where # is the number of the tool in the spindle) and press the ENTER button. This inputs the current Z axis location into the tool table. To see the tool table, type DT and press the ENTER button.

Example:

SL,1 This will input the TLO for tool #1

SL,17 This will input the TLO for tool #17

(see also TURRET (ATC), OPEN TO LOAD AND UNLOAD and TURRET (ATC), ESTABLISH TOOL ORDER)

- 8) The TLO is now set at the gauge block height above the part. Jog the Z axis to a safe point and remove the tool from the spindle and repeat steps two through eight. If this is not the gauge point the tool table must be corrected (see the next step), then repeat steps two through nine for each tool.
- 9) To correct the tool table for the gauge block, type SL,#,correction (where # is the number of the tool length to be corrected, and correction is the plus or minus distance the tool length offset must change) then press the ENTER button.

Example:

SL,1,-1. This will correct the TLO in the tool table for tool #1, making the tool go deeper one inch.

SL,17,-.9 This will correct the TLO in the tool table for tool #17, making the tool go deeper 900 thousandths.

To view the tool table:

- 1) From the command mode type DT then press the ENTER button.

WARNING:

Using the SETZ or SETH command may alter the Z axis zero position (see HOME POSITION, ESTABLISHING). This Z axis zero position is used to establish a starting position for the tool length offsets to begin from. Usually the Z axis zero position is at the Z axis cold start position. The Z axis zero position will be changed to do a dry run of the program (see DRY RUN). If the SET commands are used, wherever the Z axis is set to zero, the tool length offsets in the tool table remain the same. If the Z axis is set to zero below the original position where the tool length offsets were established from, the tools will go deeper into the part.

**Tool Length
Offsets, Purpose
and Use in
Program**

When tools are loaded into tool holders and placed into the ATC (Automatic Tool Changer), each one of the tools extends out of the holder at different lengths. The travel distance from the tool change position to the part, is different for each tool. The tool length offset is used to compensate for this difference.

The tool length offset will bring each tool used in a program to a common starting point. This point is the Z axis home position (Z0) for the part. This point is known as the gauge point.

The gauge point is chosen by the programmer, usually it is the same point as the Z0 datum established by the print. Using the same point as the print makes inspection of the program easier, because the dimensions on the print will be the same as the Z moves in the program. There is only one gauge point on the part and it is used by all the tools in the program.

The distance from the machine Z0 position to the part Z0 position is known as the tool length offset. The tool length offset for each tool is entered into the machines tool table. In the tool table each tool length offset is numbered. Usually the number in the tool table is the same number for the tool. For example tool length offset number one is for tool number one, and tool length offset number ten is for tool number ten, and so on.

In the program the H word is used to call up the TLO (tool length offset) from the tool table. The H word is always associated with a number. The number is the TLO number from the tool table. For Example: H#, where # is the TLO # from the tool table (see example below).

```
N1 (PROG # 65 P/N 56-4312 CO. NAME)
N2 M6 T1
N3 ( TOOL #1 1/2 INCH 2FL HSS EM (CRC) )
N4 G0 G90 S10000 M3 E1 X2.56 Y-1.9
N5 H1 M7 Z.1At this line, the program is calling
N6 G4 P1000 up TLO #1 from the tool table with the
N7 G1 F50. H word.
N8 X3.56 Z-.245
N9 M5 M9
N10 G0 G49 G90 Z0
```

In the program example above, line N2 does a tool change picking up the tool in turret location number one. And in line N5, TLO (tool length offset) number one is called up from the tool table (with the H1) which moves the tool .100 above the gauge point, because of the Z.1 in the same line.

When a Z axis move appears in the same line along with the H word, the two moves occur simultaneously.

The H word in a program will (1) call up the TLO from the tool table and (2) move the tool the distance in the table and (3) transfer the Z0 position from the tool change position to the part zero.

A Z axis move does not need to appear along with the H word. However this can be used to bring the tool to a position other than the Z zero position.

Example:

H8 Z1.5 This will bring the tool 1.5 inches above the gauge point.

SP COMMAND

Save Parameters, Using the SP command

This command is used to input the machine parameter list through the RS-232 port.

To output parameter information.

- 1) Get a computer ready to receive information through the RS-232 port.
- 2) Type the PU,4 command and press the ENTER button.

To receive parameter information:

- 1) Type the TA,1 command and press the ENTER button.
- 2) Send the file the parameter information was saved under to the machine.

SU COMMAND

Sum Program, Using the SU Command

The sum command is used to analyze the program without machine motion. The control will process the program as if it is moving the machine, without regard to feed rate.

Example 1:

Using the sum program command to display error messages in the program.

Note: See the FADAL USER MANUAL for information concerning the messages appearing on the screen.

- 1) From the command mode type SU then press the ENTER button.
 - a) The cursor will not be on the screen, meaning the control is processing the program.
 - b) Any error messages, while processing the program, will appear on the screen.
- 2) When messages appear on the screen, press the MANUAL button to stop the summing process.
- 3) Correct the first message that appears and use the SU command again.
- 4) Follow steps 1-3 until the program can run the SU command and without any error messages. This will be indicated with the X, Y, and Z axis locations being displayed and the control returning to the command mode.

Example 2:

Using the sum program command to display all moves from the program.

Note: See the FADAL USER MANUAL for information concerning the messages appearing on the screen.

- 1) From the command mode type SU,,,1 then press the ENTER button.
 - a) On the left hand side of the screen, the control will display all moves in the program in incremental terms. Regardless if the program is written in absolute. All positioning decisions are made in incremental.

- b) All lines in the program which require intermediary moves (i.e. circles, fixed cycles, fixed subroutines) will be shown on the screen as decimal pointed lines when sum program is displaying the moves.
- c) The listing can be stopped and started similar to the list mode. Use zero to stop the listing, and one through nine to start to list at different speeds, one being the slowest and nine being the fastest.

Example 3:

N57 G73 G99 R+0.05 Z-0.15 F25. Q0.15 X0 Y0 In the program line 57 is a drill cycle. The sum program command will display line 57 in five steps, showing the drill peck moves.

```
N57.001 Z-0.15F25.G1
N57.002 Z0.05G0
N57.003 Z-0.05G0
N57.004 Z-0.05F25.G1
N57.005 Z0.2G0
```

Example 4:

Using the sum program command to display all moves from the program and the absolute position after each move.

Note: See the FADAL USER MANUAL for information concerning the messages appearing on the screen.

- 1) From the command mode type SU,,,2 then press the ENTER button.
 - a) On the left hand side of the screen, the control will display all moves in the program in incremental terms. Regardless if the program is written in absolute. All positioning decisions are made in incremental. And the absolute position of each move is displayed on the right side of the screen.
 - b) All lines in the program which require intermediary moves (i.e. circles, fixed cycles, fixed subroutines) will be shown on the screen as decimal pointed lines when sum program is displaying the moves.
 - c) The listing can be stopped and started similar to the list mode. Use 0 to stop the listing, and 1 through 9 to list at different speeds, 1 being the slowest and 9 being the fastest.

Example 4:

```
N55 G0 G90 S2000 M3 E1 X0 Y0
N56 H2 M7 Z.05
```

```
N57 G73 G99 R0+0.05 Z-0.15 F25. Q0.15 X0 Y0
```

In program line 57 is a drill cycle. The sum program command will display line 57 in five steps, showing the drill peck moves.

N55.M3.S2000.	X = 0 Y = 0 Z = 0
N56.Z0.05D1H1G0	X = 0 Y = 0 Z = 0.0500
N57.001Z-0.15F25.G1.	X = 0 Y = 0 Z = -0.1000
N57.002Z0.05G0	X = 0 Y = 0 Z = -0.0500
N57.003Z-0.05G0	X = 0 Y = 0 Z = -0.1000
N57.004Z-0.05F25.G1.	X = 0 Y = 0 Z = -0.1500
N57.005Z0.2G0	X = 0 Y = 0 Z = 0.0500

Example 5:

Using the sum program command to display all moves from the program, the absolute position after each move, and the current G code status.

- 1) From the command mode type `SU,,,3` then press the ENTER button.

Example 6:

The first parameter is used to determine what line to begin the sum listing. If the program is summed up to a particular line in the program, it is not necessary to have the beginning portion listed again. Doing this will save time during the summing process.

- 1) From the command mode type `SU,#,,,1` (where # is the line number to start listing from) then press the ENTER button.

Example 7:

The second parameter can be used in combination with the first parameter to start displaying at any line and stop displaying at any line.

- 1) From the command mode type `SU,1st #, 2nd #,,1` (where 1st # is the first line to display, and 2nd # is the last number to display) then press the ENTER button.

Example 8:

When summing a program that uses cutter radius compensation, it is best to first sum the program without cutter radius compensation.

- 1) This could be done by removing all of the tool diameters in the tool table. Or by using the third parameter of the sum command.
 - a) From the command mode type `SU,,,1` or `SU,,,1,1` or `SU,,,1,2` or `SU,,,1,3` then press the ENTER button.

Note: Removing the diameters from the tool table will now allow the summing of programs with fixed subroutines.

TA COMMAND

Input, From a Tape Punch or Computer

The control can accept programs or tool data into its memory by using the tape input command.

- 1) The BAUD RATE of the control and the device it is connected to, must be set at the same rate. (see BAUD RATE, USING THE CD COMMAND)
- 2) From the command mode type TA,1,# (where # is an error option see below), then press the ENTER button.
- 3) Prepare the tape punch or the computer, connected to the control, to send the program or tool data.

OPTION 1: For # type 1. Using this error option for input will cause the control to halt the input process if an error is detected in the program. Example: Using the letter O for the number 0, double letters (XX.152), minus sign misplaced (-Y.45), a comment without a preceding * or (, a dimension without an axis word (X.125 .756). Observe, on the screen of the control which line the transmission was halted at, and correct this line at the text editor or Teletype. After this line is corrected repeat steps 1-3 above. If no errors are detected the message "TAPE IS GOOD" will appear.

OPTION 2: For # type 2. When this option is used the control will continue to accept data even after an error is detected. When transmission of the data is complete, a message is displayed on the screen "# ERROR(S) READING TAPE." If no errors are detected the message "TAPE IS GOOD" will appear. This option is only recommended when the program being transmitted has sequence numbers. With sequence numbers in the program the operator can list the program, if errors are detected, and look for the missing sequence numbers which were dropped because of errors in the line. Use the insert (IN) command to replace the missing lines. (see INSERT PROGRAM LINES, USING THE IN COMMAND)

OPTION 3: For # type 3. This option is used when the program to be transmitted is from another control. The control will accept the program but may not be able to run the program in its present form. Some of the coding may have to be edited to the coding format required by the control.

4) Start output from the tape punch or the computer.

Note: If the program currently in memory has an O word as the first line of the program, the program is considered a part of the program library. When the new program is transmitted to the control, the program currently in memory will be pushed into the library and retained. If the program currently in memory does not have an O word as the first line in the program it is not considered to be a part of the program library. When the new program is transmitted to the control, the program currently in memory is deleted and the new program replaces it.

Adding a Program to the End of the Program in Memory

The tape input (TA) command is used to add another program to the end of the program currently active in the control.

Using the add at end option:

- 1) The BAUD RATE of the control and the device it is connected to, must be set at the same rate. (see BAUD RATE, USING THE CD COMMAND)
- 2) From the command mode type TA,1,#,1 (where # is an error option see INPUT, FROM A TAPE PUNCH OR COMPUTER), then press the ENTER button.
- 3) Prepare the tape punch or the computer, connected to the control, to send the program or tool data.
- 4) The program being added to the end can have sequence numbers (any sequence). After the program has been added to the current program in memory, from the command mode type NU then press the ENTER button. This will give the entire program new sequence numbers. (see NUMBER PROGRAM LINES, USING THE NU COMMAND)

Note: If the NU command is not used the edit features of the PA mode may not properly function.

**Output, to Tape
Punch or Computer**

The control can output the program(s) in its memory, through the RS-232 port, by using the punch command.

- 1) The BAUD RATE of the control and the device it is connected to, must be set at the same rate. (see BAUD RATE, USING THE CD COMMAND)
- 2) Prepare the tape punch or the computer, connected to the control, to receive the program.
- 3) From the command mode type PU,# (where # is a data option), then press the ENTER button.

Below is a list of data options for # above.

- 0= for program and tooling data
- 1= for tool data only
- 2= for program only
- 3= for all programs in memory
- 4= for parameters & backlash

Note: The SETP command will allow the operator to determine the output format. When using a tape punch set the output format for tape punch. When using a computer set the output format for a computer. The tape punch format will output leader information before transmitting data and it will output a tape tail after transmitting data. (see DEFAULT PARAMETERS, USING THE SETP COMMAND)

WARNING:

Do not use the control as a program storage device. Whenever a change has been made to the program, output the program to a paper tape or store it on a floppy disk. This will insure backup of the program. When possible, make a back-up copy of the tape or floppy disk.

- 4) To output in the EIA format, type PU, #, 1 (where # is the data option) then press the ENTER button. The number one, for the second parameter is what establishes the EIA output.
- 5) To output to a Teletype use the third parameter. Type PU, #,, 1 then press the ENTER button.

Note: This command is not used when utilizing the FADAL Assist software.
(See also the Communications section of this manual.)

TC,1 COMMAND

Turret (ATC), Open for Loading

- 1) From the command mode type TC,1 then press the ENTER button. The ATC will open and remain open until the MANUAL button is pressed. If the Z axis is not at the cold start position the control will enter the waiting state. Press the START button to return to the cold start position then ATC will open.
- 2) Use the TURRET CW and the TURRET CCW buttons to rotate the turret.
- 3) Press the MANUAL button to close the ATC. This will bring the head down. If a tool is under the spindle, the tool will be loaded into the spindle and the ATC will move back to its home position.

Note: When loading tools into the turret to run a program, place tool number one in the turret and use the TURRET CW button to put the next tool in the turret until all the tools are placed in the turret. Rotate the turret until the first tool is in the number one position before setting the tool order. (See TURRET (ATC), ESTABLISH TOOL ORDER; USING THE SETTO COMMAND to establish tool order.)

Tool Changer Crash

- 1) Press the JOG button to get the turret into home position.
- 2) Move to the cold start position using either JOG or the power off procedure: Type SETCS and press ENTER. Type HO and press Enter, then press START.
- 3) Type CS and press the ENTER button. See POWER ON.
- 4) Use the TURRET CCW or CW button to rotate the turret until there is an open turret location directly opposing the spindle.
- 5) From the command mode type SETTO. This will establish this location as tool number one.
- 6) Type TC,1 to open the tool changer.

- 7) Rotate the turret with the TURRET CCW or CW buttons until tool number one is directly under the spindle, then press the MANUAL button to pick up tool number one.
- 8) Type SETTO to establish this location as tool number one.

**Turret Loading,
Putting Tools in
the Tool Changer**

When placing tools in the turret, the location of tool number one can be any convenient turret location. (See TURRET (ATC), ESTABLISH TOOL ORDER; USING THE SETTO COMMAND)

- 1) After the turret is open, move the turret around until there is an open spot directly in front of the open door.
 - a) Or take the tool out of the location directly in front of the open door. (See the TC,1 command, to open the turret door.)
- 2) Place tool #1 in the turret lid by aligning the clip with the V flange of the tool holder.
 - a) The tongue in the back of the clip must go into the key slot on the tool holder, for the tool to be seated properly.
- 3) Push the TURRET CW button, and then place the next tool in the turret lid.
 - a) The remaining tools are all placed into the turret lid, in sequential order. Press the TURRET CW button, after each tool is placed into the lid.
- 4) Use either the TURRET CW or TURRET CCW button to rotate the turret lid around so that tool number one, from the program, is directly under the spindle.
- 5) Press the MANUAL button to bring the head down over tool number one.
- 6) From the command mode type, SETTO then press the ENTER button. This will establish this location as tool number one. (See TURRET (ATC), ESTABLISH TOOL ORDER; USING THE SETTO COMMAND)

TO COMMAND

Tool Diameter Input, Using the TO Command

Tool diameters are required for the fixed subroutines except for sub 91 and 93 and when using cutter radius compensation in the program.

- 1) From the command mode type TO,#,DIAMETER VALUE (where # is the tool number) then press the ENTER button.

Example:

TO,1,.375 This will enter into the tool table (use command DT to see the table), a tool diameter of .375 for tool number one.

- 2) To change an existing tool diameter, use step one. This will overwrite the current diameter in the table.

To view the tool table

Note: Tool diameters can be entered by using the DT command.
(See also DISPLAY TOOL TABLE, USING THE DT COMMAND)

Note: In format 2 with the parameter set to radius the amount entered with the TO command is input as a radius amount.

Note: To make incremental changes the DT command must be used.

UT COMMAND

Utility Option - UT - Setting Tool Length Offsets

The UT command may be used to get tools from the turret and automatically enter the tool length offsets into the tool table. The height of the block used to establish the tool length offset is automatically compensated for when the lengths are entered into the table.

- 1) Put the tools in the turret. (See: TURRET LOADING, PUTTING TOOLS IN THE TOOL CHANGER.)
- 2) Place the first tool in the spindle with the tool turret in the proper ATC position.
- 3) Establish the tool order. (See: MEMORY CLEAR, USING THE NE AND RI COMMAND.)

***To clear the tool
table and establish
tool order.***

(See also: TURRET (ATC), ESTABLISH TOOL ORDER; USING THE SETTO COMMAND.)

- 4) From the command mode type UT and then press the ENTER button.

This menu will appear in the lower left of the screen.

OFFSET UTILITY OPTIONS:

- 1) TOOL SETTING CYCLE
- 2) FIXTURE SETTING CYCLE
- 3) TEST TS-20 PROBE
- 4) TEST MP PROBE
- 5) PALLET CHANGER
- 6) CLOCKS
- 7) EXIT

PRESS OPTION NUMBER<

Option #1: TOOL SETTING CYCLE

- 1) After typing the UT command and pressing ENTER, type the number 1 and press the ENTER button. The screen will immediately display the current tool and offset information.

TOOL SETTING CYCLE
TOOL NUMBER = 1
DIAMETER OFFSET = 0.0000
LENGTH OFFSET = 0.0000
Answer the following questions:
ENTER STARTING TOOL NUMBER <
ENTER THE ENDING TOOL NUMBER<

SETTING TOOL LENGTH OFFSETS - UT -

- 2) The next question deals with the method used for setting the lengths. This portion of the manual assumes the operator will JOG TO POSITION.

TOOL SETTING OPTIONS:

- 1) JOG TO POSITION
 - 2) PROBE LEFT MOUNT
 - 3) PROBE RIGHT MOUNT
- PRESS OPTION NUMBER<

- a) Press number 1 and immediately the next question appears.
ENTER THE HEIGHT BLOCK SIZE<
- b) Type the height of the block used for setting the tool length offset and press the ENTER button.
 - (1) The number to type in at this point is the distance from the top of the block to the gauge point on the part.
 - (2) If the gauge point is below the top of the block, then use a positive number, if it is above the block use a negative number.

The tool data for the first tool and the tool setting options menu will appear in the lower left corner of the screen.

TOOL NUMBER = #
DIAMETER OFFSET = #
LENGTH OFFSET = #
TOOL SETTING OPTIONS:
1) ENTER A TOOL DIAMETER
2) LOCATE TOOL
3) EXIT TOOL SETTING
PRESS OPTION NUMBER:

- 3) To enter a tool diameter for the current tool in the spindle press the 1 button.
 - a) Immediately this message will appear at the bottom of the screen:
"ENTER THE DIAMETER OFFSET FOR TOOL #: <."
 - b) Type the diameter and press the ENTER button.

(The dia. can be changed at this time by pressing the 1 button again.)

4) Press the 2 button.

SETTING TOOL LENGTH OFFSETS - UT -

5) The message, "PRESS JOG AND MOVE TO HEIGHT BLOCK OR PRESS MANUAL TO EXIT" appears at the bottom of the screen.

a) Press the JOG button.

b) Jog the tool down to the block.

c) Press the MANUAL button after the tool is at the block.

d) The message "PRESS START TO LOAD TOOL #

1....M6,T#," appears at the bottom of the screen. This message only occurs for the first tool change.

6) The waiting message will flash on the screen. Press the ENTER button to make the tool change.

7) Repeat steps 3-5 for the remaining tools.

VT COMMAND

Verify Punched Tape, Using the VT Command

Because devices used to punch tapes for program storage malfunction periodically, the VT command is used to verify that the tape has been punched properly.

- 1) After punching a tape (see OUTPUT, TO TAPE PUNCH OR COMPUTER), place the tape in the tape reader.
- 2) From the command mode type VT then press the ENTER button.
- 3) Start the reader.
- 4) After the tape has been read look at the screen on the control for any messages.
 - a) TAPE IS GOOD : This message indicates the program output to the tape punch is the same on the tape.
 - b) TAPE DID NOT VERIFY : This message indicates a problem with the reader or the punch itself.
 - (1) Clean the reader, place the tape back in the reader and follow steps 2-4 again.
 - (2) Clean the tape guides for anything that would push the tape over, causing the holes to be punched off to one side of the tape. (Follow steps 1-4 again.)

CNC 88HS GRAPHICS MENU

Menu

The graphics menu of the page editor has been designed to allow the user to view the part path of the current program in memory. The graphics can be accessed by pressing the G key from the page editor or by entering the command DR.

A second menu will appear, allowing the user to choose from several options (See Figure 10-4). All of these options can be pressed while plotting is taking place.

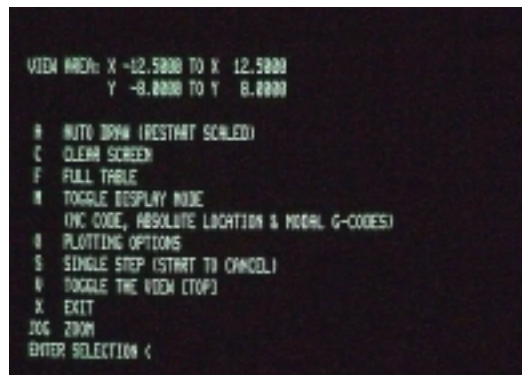


Figure 10-4 Options menu

Plotting options

A = AUTO

Pressing the A key runs the current program completely through the part path showing interpolation moves only (movement programmed at a feed rate G1,G2,G3).

C = CLEAR

Pressing the C key clears the screen and continues auto part path draw at full table plotting.

F = FULL TABLE

Pressing the F key clears the screen and continues auto part path draw at full table plotting. This is used after the part path plot has been ZOOMED inward and the user wished to see the whole part path again on a full table display.

M = TOGGLE DISPLAY MODE

Pressing the M key will toggle the options differences displayed along with the graphics plot. Toggle display options are incremental moves, absolute positions, and modal codes. The M key can be pressed while plotting in order to view the various modes.

O = OPTIONS PLOTTING

Pressing the O key display an additional menu allowing the user to choose from:

- I** Ignore G41 G42 compensation
- L** Plot only subroutines
- O** Plot only subprogram
- P** Plot total program
- T** Plot tool

Once the option key has been pressed, the plotting continues.

S = SINGLE STEP

Pressing the S key, one program line will be plotted. Repeated pressing of the S key allows the user to step through the program in line by line execution. This can be canceled at any time by pressing the START button. During single step plot the current program line will also appear on the screen in G91 incremental value.

V = VIEW TOP OR ISOMETRIC

The V Key can be pressed at any time during plotting to change the view from top to simple isometric view. Plotting restarts from the beginning. This view may not be rotated.

Z = ZOOM

Move box with the pulse generator press enter to zoom box area.