III. OPERATION

1. Introduction

In operation, it is important to be aware of the operating mode selected for the CNC. There are six operating modes and one simulation mode in this control. The operating mode is selected with the six buttons labeled:

EDIT	To edit a program already in memory
MEM	To run a program stored in memory
MDI / DNC	To directly run manually entered program or to select DNC mode
HANDLE JOG	To use jog keys or jog handle
ZERO RET	To establish machine zero
LIST PROG	To list, send, or receive programs

The Graphics simulation mode is entered with the DISPLAY select buttons.

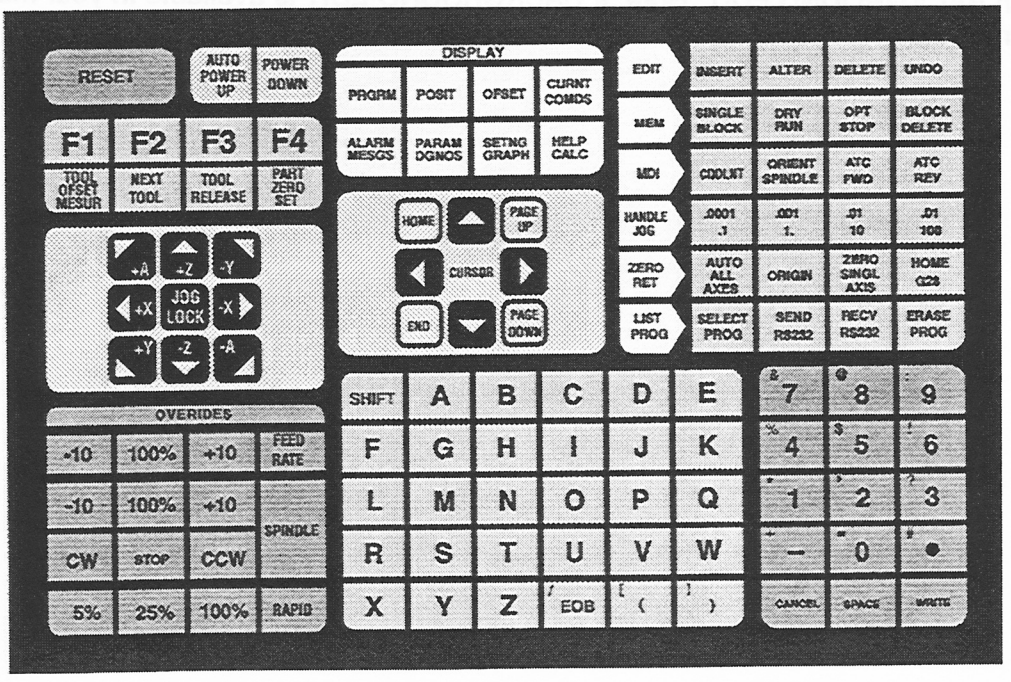

Fig. 1-1 *Control panel keypad with operating and display keys highlighted.*

In MEM or MDI mode, a program can be started with the CYCLE START button. While a program is running, you cannot change to another mode; you must wait until it finishes or press RESET to stop the program.

When already in MDI, a second push of the MDI button will select DNC if the DNC mode is enabled by settings and parameters in your machine.

In any of the above modes, you can select any of the following displays using the six DISPLAY buttons:

PROGRAM	To show the program selected
POSIT	To show the axes positions
OFSET	To show or enter working Offsets
CURNT COMDS	To show Current Commands and times
ALARM / MESGS	To show Alarms and user messages
PARAM / DGNOS	To show Parameters or Diagnostic data
SETNG / GRAPH	To show or enter Settings or to select Graphics simulation mode
HELP / CALC	To show the Help data and calculator

In addition to the above displays, when a program is already running, you may press LIST PROG to select a list of the programs in memory. This is useful to determine what programs can be edited in BACKGROUND EDIT. Note, if BACKGROUND EDIT is available in your machine, it is selected from the PROGRAM DISPLAY.

2. Operator's Control Panel

All operation of the CNC is controlled from the operator's panel. The control panel is composed of the CRT display, the keypad, On/Off switch, Load meter, Handle, EMERGENCY STOP, CYCLE START, and FEED HOLD buttons.

The **keypad** is a flat membrane type that requires approximately eight (8) ounces of pressure. The **SHIFT** button changes the function of these buttons with a blue character in the upper left corner. The **SHIFT** button must be pressed once before each shifted character. Pressing the **SHIFT** button twice will turn off shift.

The **load meter** measures the power to the spindle motor. At 100%, the spindle motor can be operated continuously. The 150% level can be sustained for no more than five (5) minutes. At 180%, the spindle may begin to slow and even stall. A 180% load should be reduced to 150% by reducing spindle speed or decreasing the feed rate.

The **Handle** is used to jog one of the axis. Each step of the crank can be 0.0001, 0.001, or 0.01 inch. The rotary axis is 0.001, 0.01, or 0.1 degree per step. The handle has 100 steps per rotation. When using metric, the smallest handle step is 0.001 mm and the largest is 0.1 mm. As an option, the handle can also be used to move the screen cursor while in EDIT mode.

The EMERGENCY STOP button will instantly stop all motion of the machine including the servo motors, the spindle, the tool changer, and the coolant pump. It will also stop any auxiliary axes.

CYCLE START will start a program running in MEM or MDI mode, continue motion after a FEED HOLD, or continue after a SINGLE BLOCK stop.

FEED HOLD will stop all axis motion until the CYCLE START is pressed. FEED HOLD will not stop the spindle, the tool changer, or the coolant pump. It will not stop motion of any auxiliary axes.

The SINGLE BLOCK button on the keypad will turn on and off the SINGLE BLOCK condition. When in SINGLE BLOCK, the control will operate one block and stop. Every press of the START button will then operate one more block.

The RESET button on the keypad will always stop motion of the servos, the spindle, the coolant pump, and tool changer. It will also stop the operation of a running program. This is not, however, a recommended method to stop the machine as it may be difficult to continue from that point. SINGLE BLOCK and FEED HOLD provide for continuation of the program. RESET will not stop motion of any auxiliary axes but they will stop at the end of any motion in progress.

The CRT is the only display or readout device in the control. All status and position data is shown on the CRT.

The F1, F2, F3, and F4 buttons perform different functions depending on what display and mode is selected. The following is a quick summary of the **Fn** buttons:

F1 In EDIT mode and PROGRAM DISPLAY, this will start a block definition.

In LIST PROG mode, F1 will duplicate a program already stored and give it a new name from the command line.

In offsets display, F1 will set the entered value into the offsets.

F2 In EDIT mode, PROGRAM DISPLAY, this will end a block definition.

F3 In EDIT and MDI modes, the F3 key will copy the highlighted circular help line into the data entry line at the bottom of the screen. This is useful when you want to use the solution developed for a circular motion. Push INSERT to add that circular motion command line to your program.

In the calculator Help function, this button copies the value in the calculator window to the highlighted data entry for Trig, Circular, or Milling Help.

F4 In MEM mode and PROGRAM DISPLAY, this will select either BACKGROUND EDIT or PROGRAM REVIEW. BACKGROUND EDIT is selected by entering **Onnnn** with the program number to edit. Program review is selected with just F4. Program review shows the running program on the left half screen and allows the operator to review the program on the right half screen.

In the calculator Help function, this button uses the highlighted Trig, Circular, or Milling data value to load, add, subtract, multiply, or divide with the calculator.

2.1 Keyboard

The control panel keyboard consists of 123 keys and is divided into nine separate regions. They are:

RESET keys	Three (3) keys
FUNCTION keys	Eight (8) keys
JOG keys	Nine (9) keys
OVERRIDES	Twelve (12) keys
DISPLAYS	Eight (8) keys
CURSOR keys	Eight (8) keys
ALPHA keys	Thirty (30) keys
MODE keys	Thirty (30) keys
NUMERIC keys	Fifteen (15) keys

A detailed description of how and where these keys are used can be found through use of the index. The following are short descriptions of the control panel keys' usage.

RESET KEYS: The RESET keys are in the upper left corner of the control panel.

RESET	Stops all machine motion and places the program pointer to the top of the current program.
AUTO POWER UP	Automatically initializes the machine at power up.
POWER DOWN	Automatically positions axes to machine zero and prepares the machine for power down condition.

FUNCTION KEYS: Below the reset keys are the function keys. There are eight function keys. They are used to execute special functions implemented throughout the control software.

F1-F4	Used in editing, graphics, background edit, and the help/calculator to execute special functions.
TOOL OFSET MESUR	Used to record tool length offsets in the offset page during part setup.
NEXT TOOL	Used to select the next tool during part setup.
TOOL RELEASE	Releases the tool from the spindle when in MDI mode. (The remote TOOL RELEASE button is located on the front of the cover to the spindle head. It operates the same as the one on the keypad. It must be held for ½ second before the tool will be released, and the tool will remain released for ½ second after the button is released. While the tool is unclamped, air is forced down the spindle to clear chips, oil, or coolant away from the tool holder.
PART ZERO SET	Used to automatically set work coordinate offsets during part setup.

JOG KEYS: The jog keys are on the left below the function keys. These keys select which axes the jog handle sends signals to and provides for continuous jogging. When a key is pressed briefly, that axis is selected for use by the jogging handle. When a key is pressed and held down, that axis is moved as long as the key is held down. If a "+" key is pressed and held, the axis is moved so that the tool position is changed in a positive direction relative to the work coordinates. If a "-" key is pressed and held, the axis is moved so that the tool position is changed in a negative direction relative to the work coordinates.

+A, -A	Selects the A axis. Selects the B axis when used with the shift key and control is configured with a fifth-axis option.
+Z, -Z	Selects the Z axis.
+Y, -Y	Selects the Y axis.
+X, -X	Selects the X axis.
JOG LOCK	When pressed prior to one of the above keys, the axis is moved in a continuous motion without the need to hold the axis key depressed. Another press of the key stops jogging motion.

OVERRIDES: The overrides are at the lower right of the control panel. They give the user the ability to override the speed of rapid traverse motion, as well as programmed feeds and spindle speeds.

FEED RATE	Not a key.
-10	Decreases current feed rate by 10 percent.
100%	Sets control feed rate to programmed feed rate.
+10	Increases current feed rate by 10 percent.
SPINDLE	Not a key.
-10	Decreases current spindle speed by 10 percent.
100%	Sets spindle speed to programmed speed.
+10	Increases current spindle speed by 10 percent.
CW	Starts the spindle in the clockwise direction.
STOP	Stops the spindle.
CCW	Starts the spindle in the counterclockwise direction.
RAPID	Not a key.
5%	Limits rapid traverse to 5 percent of maximum.
25%	Limits rapid traverse to 25 percent of maximum.
100%	Allows rapid traverse to feed at its maximum.

DISPLAYS: The display keys are in the center at the top. These eight keys provide access to the different displays and operational information and help routines available to the user. Some of these keys are multi-action keys in that they will display different screens when pressed multiple times. The current display is always displayed on the top left line of the video screen.

PRGRM	Displays the currently selected program.
POSIT	Displays the position of the machine axes. Pressing PAGE UP and PAGE DOWN will show operator, machine, work, and distance-to-go formats in large letter format.
OFSET	Displays the tool length and radius offsets. PAGE UP will display the values of the axes work offsets.
CURNT COMDS	Displays the current program, modal program values, and position during run time. Succeeding presses of the PAGE DOWN key will display modal values, system timers, macro variables, tool life and tool load information.
ALARM MESGS	Shows the full text of an alarm when the alarm message is flashing. Pressing the left or right arrow keys will display an alarm history.

Pressing PAGE DOWN will display a page for user messages and notes.

PARAM DGNOS — Displays and allows changing of parameters that define machine character. Pressing PAGE UP will display lead screw compensation values. Successive PAGE DOWN presses will display general parameters as well as the X, Y, Z, A and B parameters. A second press of the PARAM key will display the first page of diagnostic data. The first page of diagnostic data is discrete inputs and outputs. Pressing PAGE DOWN will display the second page of diagnostic data that consists of additional inputs and analog data.

SETNG GRAPH — Displays and allows changing of user settings. Pressing the SETNG key twice enables graphics mode where the user can debug the current program and view the program's generated tool path.

HELP CALC — Displays a brief, on-line, manual. Pressing HELP a second time will display the help calculator. There are three pages of calculator help. Pressing the PAGE DOWN key will display milling and tapping help, triangle help, or circle help.

CURSOR KEYS: The cursor keys are in the center of the control panel. They give the user the ability to move to various screens and fields in the control. They are used extensively for editing of CNC programs.

HOME — Context-sensitive key that generally moves the cursor to the topmost item on the screen. In editing, this is the top block of the program. In graphics, it will display the entire table in the view window after F2 is pressed.


(UP ARROW) The up arrow moves up one item, a block, or field. In graphics, the zoom window is moved up.

PAGE UP — Used to change displays, move up one page in the editor, or zoom out when in graphics.


(LEFT ARROW) Used to select individually editable items within the editor, moves cursor to the left. It selects optional data in fields of the settings page and moves the zoom window left when in graphics.


(RIGHT ARROW) Similar to the LEFT arrow, but moves right.

END — Context-sensitive key that generally moves the cursor to the bottom most item on the screen. In editing, this is the last block of the program.


(DOWN ARROW) Similar to the UP arrow but moves down.

PAGE DOWN — Used to change displays, move down one page in the editor, or zoom closer when in graphics.

ALPHA KEYS: The alpha keys allow the user to enter the 26 letters of the alphabet along with some special characters.

SHIFT — The shift key provides access to the blue characters on the keyboard. Pressing SHIFT and then the blue character will result in that character being sent to the control. When a control has a fifth-axis installed, the B axis is selected for jogging by pressing SHIFT and then the +,-A keys.

EOB — This is the END-OF-BLOCK character. It is displayed as a semicolon on the screen and it signifies the end of a programming block. It is the same as a carriage return and then a line feed.

( ) — The parenthetical brackets are used to separate CNC program commands from user comments. They must always be entered as a pair and may or may not have additional characters separating them. Any time an invalid line of code is received through the RS-232 port while receiving a program, it is added to the program between these two brackets.

/ The right-leaning slash is used as a block delete flag. If this symbol is the first symbol in a block and a BLOCK DELETE is enabled, then that block is ignored at run time. The symbol is also used for division in macro expressions.

[and] Square brackets are used in macro expressions and functions.

MODE KEYS: The mode keys are in the upper right part of the control panel. These keys change the operational state of the CNC machine tool. There are six major operation modes. The user can enter a specific mode by pressing the desired "arrow" shaped key on the left. The keys in the same row as the pressed mode key are then made available to the user. Otherwise, these keys are not available. The current mode is always displayed on the top line just to the right of the current display on the video screen.

EDIT	Selects edit mode.
INSERT	Inserts the text in the input buffer after the current cursor location. Also used to copy blocks of code in a program.
ALTER	Changes the item that the cursor is on to the text in the input buffer. Places an MDI program in the program list.
DELETE	Deletes the item that the cursor is on.
UNDO	Backs out or undoes up to the last 10 edit changes.
MEM	Selects MEM mode.
SINGLE BLOCK	Turns single block **on** so that when the cycle start button is pressed, only one block of the program running is executed.
DRY RUN	Used to check actual machine movement without cutting a part. Programmed feeds are replaced by the speed keys in the handle jog row.
OPT STOP	Turns on optional stops. If an M01 code is encountered in the program then a stop is executed.
BLOCK DELETE	Blocks with a slash "/" as the first item are ignored or not executed when this option is enabled.
MDI	Selects MDI mode.
COOLNT	Turns the coolant on and off.
ORIENT SPINDLE	Rotates the spindle to a known position and then locks the spindle. Can be used during setup to indicate parts.
ATC FWD	Rotates the tool turret forward to the next sequential tool. If **Tnn** is in the input buffer, the turret will advance to tool **nn**.
ATC REV	Rotates the tool turret backward to the previous tool. If **Tnn** is in the input buffer, the turret will advance to tool **nn**.
HANDLE JOG	Selects Jogging mode.
.0001, .1	.0001 inches or .001 mm for each division on the jog handle. For dry run .1 inches/min.
.001, 1.	.001 inches or .01 mm for each division on the jog handle. For dry run 1. inches/min.
.01, 10.	.01 inches or .1 mm for each division on the jog handle. For dry run 10. inches/min.
.01, 100.	.01 inches or .1 mm for each division on the jog handle. For dry run 100. inches/min.
ZERO RET	Selects Zero Return mode.
AUTO ALL AXES	Searches for all axes' machine zero.
ORIGIN	Zeros out various displays and timers.
ZERO SINGL AXIS	Returns the axis that is specified in the input buffer to machine zero.
HOME G28	Returns all axes to machine zero. Does not search.
LIST PROG	Selects Program List mode and displays a list of the programs in the control.
SELECT PROG	Makes the highlighted program on the program list the current program. The current program will have an asterisk preceding it in the program list.

SEND RS232	Transmits programs out the RS232 serial port.
RECV RS232	Receives programs from the RS232 serial port.
ERASE PROG	Erases the highlighted program or the program specified in the input buffer.

NUMERIC KEYS: The numeric keys give the user the ability to enter numbers and a few special characters into the control.

CANCEL	The Cancel key is used to delete the last character entered during editing or field input.
SPACE	This is a space and can be used to format comments placed into programs.
WRITE	This acts as the general purpose enter key. Any time that user needs to change any information in the control, this key is pressed.
-, .	Used to negate numbers, or provide decimal precision.
+, =, #, and *	These symbols are accessed by first pressing the SHIFT key and then the key with these symbols. They are used in macro expressions.
?, %, $, !, &, @,and :	These are additional symbols, accessed by pressing the SHIFT key, They can be used in program comments.

3. Power On/Off and Setup

3.1 Power On

There is only one way to turn on this CNC. This is by the green "On" button at the top-left of the control panel. The main breaker at the rear of the mill must be on before this button will turn on the mill. Any interruption to power will turn the mill off and this button must be used to turn power back on again.

Upon power up, the machine must find its fixed reference point before any operations can occur. After power on, pressing the AUTO POWER UP will establish this point. The ZERO RET mode and AUTO ALL AXES button may also be used to initialize the system after all alarms are cleared. A single axis can be selected by first pushing the **X**, **Y**, **Z**, or **A** key and then the ZERO SINGL AXIS key. The position thus found is used as machine zero. Note that the Z-axis will shift downwards about five inches as the search for zero is finished; so, keep clear.

CAUTION !!!! After power on, the machine does not know its home position or stored stroke limits until it has been zero returned by the AUTO POWER UP key or the ZERO RET/AUTO ALL AXES key. It is possible to jog the machine with the handle or jog keys at the lower feeds. If it is jogged unchecked in the negative directions, you may damage the sheet metal covers or overload the ball screws. To avoid this, always properly ZERO RET the machine immediately after power on before doing anything else.

After initializing, all Position displays are reset to zero. The ORIGIN key will zero out the operator display only when that display is selected and in JOG mode. During ZERO RET, the Distance To Go display contains a diagnostic value.

The HOME G28 key should be used any time after the initial power up. This will return the Z-axis first and then the **X**, **Y**, and **A** axes all at rapid rate. If the Z-axis is positioned above the machine zero, the **X**, **Y**, and **A** axes are moved first. This key will work in any of the operating modes. The manual G28 button does not use any intermediate return point the way the programmed G28 does. Any auxiliary axes (**B**, **C**, ...) are returned to home after **X**, **Y**, **Z**, and **A**.

Note: Repairs to the motor, ball screw, or home switch will affect the zero return point and must be done only by a factory trained technician. Serious damage to the ball screw, way covers, linear guides, or tool changer may occur if the zero return point is not properly set.

3.2 Power Off

There are several ways of turning the machine off. By pushing the red POWER OFF button, power to the machine is removed instantly. Another preferable method is to use the POWER DOWN key. The POWER DOWN key will put the machine in the proper order for turn on the next time and will also sequence an orderly machine shut down. When this key is pushed, the axes will zero return, tool one will be put in the spindle, and the machine will turn off in 30 seconds.

The machine can also be programmed to turn off at an end of cycle (M30) or after a preset amount of time that the machine sees no activity. These are settings 1 and 2 on the Setting page.

A sustained overvoltage condition or sustained overheat condition will also shut this machine off automatically. If either of these conditions remains true for 4.5 minutes, the machine will start the 30 second auto-shutdown as above. Alarm 176 is displayed when an overheat shutdown begins and alarm 177 is displayed when an overvoltage shutdown begins.

Any power interruption, including the rear cabinet main circuit breaker, will also turn this machine off. Power must be restored and the POWER ON button pressed to restore operation.

3.3 Setup Procedures

The following sequence of operations is strongly recommended for setting up this machine:

1) Load a program into memory. This is done either manually entered or downloaded from a CAM package via the RS-232 interface.

2) Determine tools needed and get them ready.

3) Use a vise or fixture to hold the part and mount on mill.

4) Locate the **X** and **Y** zero points of your program on the part. Usually these points will coincide with the print reference point where dimensioning begins and needs to be clearly indicated by the programmer. Use an edge finder or indicator to locate this point with the handle function. After locating the programmed zero point, push the display offset key and page down until the work coordinate page appears. Use the cursor to get to work set G54 **X**. Push the PART ZERO SET button and the X-axis machine value will be stored at this location. Cursor to the G54 **Y** location and repeat the above. You have now told the machine where part zero is located. Usually **Z** and **A** values will not have to be set and should be zero.

5) Remove any tools from changer and MDI a T1 M6 command to install tool #1 into spindle (it should be empty). Put your tool #1 into the spindle using the TOOL RELEASE button. Push the OFSET key and page down to get to the tool offset page and cursor to tool #1. Do not install any tools directly into the carousel. Use MDI or ATC FWD/REV to retrieve tools.

6) Push the Z- JOG key until you are close to the top of your part. (The top of your part should be Z0). Use the Handle to accurately position the tool edge to Z0. Push the TOOL OFSET MESUR key and the **Z** machine value will be stored in tool offset #1. Note that this automatic offset measurement works with G43 only and the work **Z** offset must be zero.

7) Push the NEXT TOOL key and the Z-axis will retract to tool change and tool #2 (empty) will be installed in the spindle. Put your tool #2 into the spindle and jog to **Z** zero as you did for tool #1. The cursor will automatically be on offset #2. Push TOOL OFSET MESUR.

8) Repeat this procedure until all tools are measured and installed.

9) MDI a T1 M6 to return to tool #1.

10) You are now ready to run your program.

11) Please note that in order to load and measure all of your tools, you do not need to use any buttons other than JOG, TOOL OFSET MESUR, and NEXT TOOL. Also note that this automatic offset measurement works with G43 only.

4. Manual Operation

4.1 MDI

Manual data input allows you to enter data that can be executed on a line by line basis instantly without having to use the EDIT and MEM modes. In this control, MDI is actually a scratch pad memory that can execute many lines of instruction without having to disturb your main program in memory. The data in MDI will be retained even when switching modes or in power off.

Editing with MDI is the same as memory editing.

The MDI mode also allows for manual operation of coolant, spindle, and tool changer.

A program in MDI can be saved as a normal named program in memory by placing the cursor at the beginning of the first line (HOME), typing **Onnnn** (new program number), then pushing ALTER. This will add that name to the program list and clear MDI.

The entire MDI program may be cleared by pressing the ERASE PROG key while in MDI.

A fast way to select a tool is to type **Tnn** and, instead of INSERT, press either ATC FWD or ATC REV. This will directly select that tool.

When DNC is available in your machine, a second push of the MDI button will put the control into DNC mode.

When the Parameter 57 flag DOOR STOP is set to 1, manual tool change operations are not allowed with the doors open. In addition to this, the maximum spindle speed is 100 RPM.

4.2 Handle/Jog

Manually moving the axes is accomplished by pressing the mode button labeled HANDLE JOG and then by using the JOG keys or the Handle to move the axis. Both the JOG buttons and the Handle are enabled simultaneously without needing to select between them.

Jog feed rate or handle resolution is selected by the four keys to the right of the HANDLE JOG key. Jog feeds from .1 inch per minute to 100 inch per minute or handle divisions from .0001 inch to .01 inch are selectable. Auxiliary axes cannot be manually jogged from the front panel. The single axis control jog button should be used for this.

During jogging, the FEED RATE override buttons will adjust the rates selected from the keypad. This allows for very fine control of the jog speed. It does not change the handle step size.

In the center of the jog buttons is a key labeled JOG LOCK. This key will cause the axis you are jogging to continue jogging even after you release the key. Press this key and then press the selected axis motion key to start. Motion will stop as soon as the JOG LOCK button is pressed again, another axis is selected or RESET is pressed. This feature is handy when you are slow milling the soft jaws of a vise as an example.

In order to select another axis for jogging while using the Handle, use +/- **X**, **Y**, **Z**, or **A** buttons. When one of these buttons is pressed, that axis is selected for HANDLE JOG but does not move unless the button is held down for more than ½ second. After ½ second, that axis is moved in the selected direction and at the selected feed rate.

All aspects of handle jogging for the fifth axis work as they do for the other axes. The exception is the method of selecting jog between axis **A** and axis **B**.

By default the '+A' and '-A' keys, when pressed, will select the **A** axis for jogging. The display will show "JOGGING A AXIS HANDLE .01" while you are jogging the **A** axis. The **B** axis can be selected for jogging by pressing the 'shift' key, and then pressing either the '+A' or '-A' key. When this is done the control will switch to jogging the **B** axis and the display will change to "JOGGING B AXIS HANDLE .01".

The axis assigned to the '+A' and '-A' keys will remain selected for jogging even if the operating mode is changed or if the machine is turned off. The selected axis for '+A' and '-A' can be toggled by pressing the shift key prior to pressing the '+A' or '-A' keys.

5. Automatic Operation

5.1 Operation Mode

There are six modes of operation of the VF Series CNC Mill. They are:

EDIT	Used to make manual changes to a part program.
MEM	Used to run a users part program stored in memory.
MDI	Used to quickly manually enter and run a program.
HANDLE/JOG	Used to move the axes with the handle or JOG buttons.
ZERO RET	Used to search for machine zero and to return to machine zero automatically.
LIST PROG	Used to list, send, receive and delete programs.

Changes to the mode are made by pressing of the buttons on the top right quadrant of the keypad that have the above labels. If an operation is started, such as running a program, you cannot change modes until the operation is stopped. The six mode selection buttons are arranged vertically and, generally, the keys to their right apply only in that selected mode.

5.2 Program Selection

Program selection is done from the LIST PROG mode. This mode will list all of the programs stored in memory and allow you to select one as the main program. This is the program that will be run when you press START in MEM mode. On the LIST PROG display, the program with the " * " is the main program. The selected program is the one that you will see on the EDIT display and is the one that will be run when you press START in MEM mode.

To select an existing program, press the CURSOR **up** or **down** buttons until the program you want is highlighted (bright) and then press the SELECT PROG button. The " * " will move to that program.

To select a new program (create a new program) or to select an existing program, you may also enter **Onnnn** from the keyboard and then the press SELECT PROG button.

There is a maximum of 200 programs stored in this control at a time.

5.3 Starting Automatic Operation

Before you can run a program, it must be loaded in the current memory. To select a program, push the LIST PROG mode key. Use the cursor to find the desired program and then push SELECT PROG. The program list includes the program name and the first comment. If the control was turned off while running, that program will automatically be in current memory and selected.

If the machine has just powered up, you need to first push the AUTO POWER UP key. This will initialize all axes and the tool changer, display the Current Commands, and go to MEM mode with the control ready to run. Pushing the CYCLE START button in the lower left of the control panel will begin execution.

To start a program other than at the beginning, scan to the block number using the keypad and the **down** arrow or PAGE DOWN until you reach the desired start place. Push the MEM key and CYCLE START to begin. The Program Restart function, selected from the Setting page 36, will change the way a program operates if you start from other than the first block. The setting called Program restart "ON" will ensure that the correct tool and axis positions are selected when you start from part way through a program.

Any errors in your program will cause an alarm and stop the running of the program. Typical alarms are travel limits and missing **I**, **J**, and **Q** codes. Attempts to move outside of the limits of travel will also cause an alarm.

When cutting materials that produce hot chips, use coolant to prevent damage to the plastic windows and enclosure.

At any time that a program is running, the bottom left corner of the CRT will show RUNNING. If it does not show this, the program has completed, has been stopped by the operator, or has been stopped by a fault condition.

5.4 Program Restart

The Program Restart function may be selected from the Setting page. It allows a program to be restarted from other than the first block. You do this by using the CURSOR **up** and **down** keys in MEM mode to select the block to start operation and pressing CYCLE START. If Program-Restart is on, program interpretation will begin with the first block but no motion of the machine will occur until execution gets to the selected restart block. When it gets to the restart block, the axis and tools will be moved to the correct position and normal operation will proceed from there.

5.5 Stopping Automatic Operation

There are several ways a program can be stopped. They include both normal stops and abnormal, or alarm caused, stops. The normal stops are:

1) Normal completion at M00, M01, M02, or M30

2) A FEED HOLD stop by the operator. This is continued by pressing CYCLE START again.

3) A SINGLE BLOCK stop when operator selected. This is continued by pressing START again.

4) Door Hold stop caused by operator opening the enclosure doors. This continues when doors close.

The abnormal stops are:

1) **Operator Reset**
This stops all axes' motion, stops the tool changer, turns off the spindle, and turns off the coolant pump. Program operation cannot be continued from the stopping point. If Setting 31 is On, the program pointer is reset to the beginning.

2) **Emergency Stop**
This stops all axes' motion, disables the servos, stops the tool changer, turns off the spindle, and turns off the coolant pump. Program operation cannot be continued from the stopping point. This will also stop any auxiliary axes' motion. RESET must be used at least twice to remove the alarms and start again.

3) **Alarm condition**
This can occur any time an alarm comes on during program operation. Since a program cannot be restarted until RESET is pressed, a program execution cannot be continued from the stopping point. Alarms can be caused by programming errors or machine faults. Use the Graphics simulation mode to test your program first for errors.

4) **Power-off**
This will stop all motors within one second but does not guarantee any conditions when the machine is powered-on again.

6. Override Functions

6.1 Feed/Rapid/Spindle Overrides

The feed rate can be varied from 10% to 200% of the programmed value while in operation. This is done with the feed rate +10%, -10% and 100% buttons. The FEED RATE override is ineffective during G74 and G84 tapping cycles. FEED RATE override does not change the speed of any auxiliary axes.

During manual jogging, the feed rate override will adjust the rates selected from the keypad. This allows for fine control of the jog speed.

The spindle speed can also be varied using the SPINDLE override as above and is also ineffective for G74 and G84. In the SINGLE BLOCK mode, the spindle may be stopped. It will automatically start up upon continuing the program.

Rapid moves (G00) may be limited to five or 25% of maximum. If the **100%** rapid is too fast, it may be set to **50%** of maximum by Setting 10 on the Setting page.

In the Setting page, it is possible to disable the override keys so that the operator cannot select them. This is Setting 19, 20 and 21.

The FEED HOLD button acts as an override button as it sets the rapid and feed rates to zero when it is pressed. The CYCLE START must be pressed to proceed after a FEED HOLD. When in a FEED HOLD, the bottom left of the screen will indicate this. The door switch on the enclosure also has a similar result but it will display "Door Hold" when the door is opened. When the door is closed, machine operation will continue normally. Door hold can be prevented with Setting 51. Door Hold and FEED HOLD do not stop any auxiliary axes.

When Parameter 57 flag DOOR STOP SP is set to 1, the door switch will stop the servos and the spindle. In addition, the override setting does not work, and manual tool changes cannot be done with the door open.

There is also an override function for the coolant supply. This is done from the Setting 32. The "NORMAL" setting checks the low coolant alarm and turns the pump on and off with **M** codes. The "OFF" setting ignores the coolant alarm but will alarm if an attempt is made to turn the coolant on. The "IGNORE" setting is used to ignore all coolant commands and the low coolant alarm.

At any time a program is running, the operator may override the coolant setting by pressing the MDI Coolant button. The pump will remain either on or off until the next **M** command or operator action.

6.2 Dry Run Operation

The DRY RUN function is used to check a program quickly without actually cutting parts. DRY RUN is selected by pressing the DRY RUN button while in MEM or MDI mode. When in DRY RUN, all rapids and feeds are run at the DRY RUN speed selected from the JOG speed buttons. The bottom of the screen will display the rate as 100, 10, 1.0 or 0.1 inches per minute.

DRY RUN cannot be turned on while a program is running. It can only be turned on or off when a program has completely finished or is reset. The first push of the DRY RUN button turns on this function and the second push will turn it off again. DRY RUN will still make all of the requested tool changes. The speed used in DRY RUN can be changed at any time and the operator can then check that the motions that are programmed are exactly what were intended. Note that Graphics mode is just as useful and may be even safer since it does not begin moving the machine before the program is checked.

8. What To Do When Alarms Occur

8.1 Alarms

Any time an alarm is present, the lower right hand corner will have a blinking "ALARM". Push the ALARM display key to view the current alarm. All alarms are displayed with a reference number and a complete description. If the RESET key is pressed, one alarm will be removed from the list of alarms. If there are more than 18 alarms, only the last 18 are displayed and the RESET must be used to see the rest. The presence of any alarm will prevent the operator from starting a program.

Note that tool changer alarms can be easily corrected by first correcting any mechanical problem, pressing RESET until the alarms are clear, selecting ZERO RET mode, and selecting AUTO ALL AXES. Some messages are displayed while editing to tell the operator what is wrong but these are not alarms. See the editing topic for those errors.

8.2 Alarm List

The following alarm list shows the alarm numbers, the text displayed along with the alarm, and a detailed description of the alarm, what can cause it, when it can happen, and how to correct it.

Alarm number and text:	Possible causes:
102 Servos Off	Indicates that the servo motors are off, the tool changer is disabled, the coolant pump is off, and the spindle motor is stopped. Caused by EMERGENCY STOP, motor faults, tool changer problems, or power fail.
103 X Following Error Too Large	Too much load or speed on X-axis motor. The difference between the motor position and the commanded position has exceeded a parameter. The motor may also be stalled, disconnected, or the driver failed. The servos will be turned off and a RESET must be done to restart. This alarm can be caused by problems with the driver, motor, or the slide being run into the mechanical stops.
104 Y Following Error Too Large	same as 103.
105 Z Following Error Too Large	same as 103.
106 A Following Error Too Large	same as 103.
107 Emergency Off	EMERGENCY STOP button was pressed. Servos are also turned off. After the E-STOP is released, the RESET button must be pressed at least twice to correct this; once to clear the E-STOP alarm and once to clear the Servo Off alarm.
108 X Servo Overload	Excessive load on X-axis motor. This can occur if the load on the motor over a period of several seconds or even minutes is large enough to exceed the continuous rating of the motor. The servos will be turned off when this occurs. This can be caused by running into the mechanical stops but not much past them. It can also be caused by anything that causes a very high load on the motors.
109 Y Servo Overload	same as 108.
110 Z Servo Overload	same as 108.
111 A Servo Overload	same as 108.

112 No Interrupt	Electronics fault. Call your dealer.
113 Shuttle In Fault	Tool changer not completely to right. During a tool changer operation the tool in/out shuttle failed to get to the in position. Parameters 62 and 63 can adjust the time-out times. This alarm can be caused by anything that jams the motion of the slide or by the presence of a tool in the pocket facing the spindle. A loss of power to the tool changer can also cause this, so check CB5 and relays 1-8, 2-1, and 2-2.
114 Shuttle Out Fault	Tool changer not completely to left. During a tool changer operation the tool in/out shuttle failed to get to the out position. Parameters 62 and 63 can adjust the time-out times. This alarm can be caused by anything that jams the motion of the slide or by the presence of a tool in the pocket facing the spindle. A loss of power to the tool changer can also cause this, so check CB5 and relays 1-8, 2-1, and 2-2.
115 Turret Rotate Fault	Tool carousel motor not in position. During a tool changer operation the tool turret failed to start moving or failed to stop at the right position. Parameters 60 and 61 can adjust the time-out times. This alarm can be caused by anything that jams the rotation of the turret. A loss of power to the tool changer can also cause this, so check CB5 and relays 1-8, 2-3, and 2-4.
116 Spindle Orientation Fault	Spindle did not orient correctly. During a spindle orientation function, the spindle is rotated until the lock pin drops in; but the lock pin never dropped. Parameters 66, 70, 73, and 74 can adjust the time-out times. This can be caused by a trip of circuit breaker CB4, a lack of air pressure, or too much friction with the orientation pin.
117 Spindle High Gear Fault	Gearbox did not shift into high gear. During a change to high gear, the spindle is rotated slowly while air pressure is used to move the gears but the high gear sensor was not detected in time. Parameters 67, 70 and 75 can adjust the time-out times. Check the air pressure, the solenoids circuit breaker CB4, and the spindle drive.
118 Spindle Low Gear Fault	Gearbox did not shift into low gear. During a change to low gear, the spindle is rotated slowly while air pressure is used to move the gears but the high gear sensor was not detected in time. Parameters 67, 70 and 75 can adjust the time-out times. Check the air pressure, the solenoids circuit breaker CB4, and the spindle drive.
119 Over Voltage	Incoming line voltage is above maximum (about 255 volts when wired for 240 or 235 when wired for 208). The servos will be turned off and the spindle, tool changer, and coolant pump will stop. If this condition remains for 4.5 minutes, an automatic shutdown will begin.
120 Low Air Pressure	Air pressure dropped below 80 PSI for a period defined by parameter 76. Check your incoming air pressure for at least 100 PSI and ensure that the regulator is set at 85 PSI.
121 Low Lub or Low Pressure	Way lube is low or empty or there is no lube pressure or too high a pressure. Check tank at rear of mill and below control cabinet. Also check connector P5 on the side of the control cabinet. Check that the lube lines are not blocked.
122 Control Overheat	The control internal temperature is above 150 degrees F. This can be caused by almost anything in the control overheating. But is usually caused by overheat of the two regen resistors for servos and spindle

	drive. This alarm will also turn off the servos, spindle drive, coolant pump, and tool changer. One common cause of this overheat condition is an input line voltage too high. If this condition remains for 4.5 minutes, an automatic shutdown will begin.
123 Spindle Drive Fault	Overheat or failure of spindle drive or motor. The exact cause is indicated in the LED window of the spindle drive inside the control cabinet. This can be caused by a stalled motor, shorted motor, overvoltage, undervoltage, overcurrent, overheat of motor, or drive failure.
124 Low Battery	Memory batteries need replacing within 30 days. This alarm is only generated at power on and indicates that the 3.3 volt Lithium battery is below 2.5 volts. If this is not corrected within about 30 days, you may lose your stored programs, parameters, offsets, and settings.
125 Shuttle fault	Tool shuttle not initialized at power on, CYCLE START or spindle motion command. This means that the tool shuttle was not fully retracted to the Out position.
126 Gear Fault	Gearshifter is out of position when a command is given to rotate the spindle. This means that the two speed gear box is not in either high or low gear but is somewhere in between. Check the air pressure, the solenoids circuit breaker CB4, and the spindle drive.
127 No Turret Mark	Tool carousel motor not in position. The turret motor only stops in one position indicated by a switch and cam on the Geneva mechanism. This alarm is only generated at power-on. The AUTO ALL AXES button will correct this but be sure that the pocket facing the spindle afterwards does not contain a tool.
128 Tool In Turret	Pocket opposite spindle has tool in it. Future option not yet implemented.
129 M Fin Fault	M-Fin was active at power on. Check the wiring to your **M** code interfaces. This test is only performed at power-on.
130 Tool Unclamped	Tool release piston is energized at power up. This is a possible fault in the air solenoids, relays on the IO Assembly, the draw bar assembly, or wiring.
131 Tool Not Clamped	Tool Release Piston is not Home. This is a possible fault in the air solenoids, relays on the IO Assembly, the draw bar assembly, or wiring.
132 Power Down Failure	Machine did not turn off when an automatic power-down was commanded. Check wiring to POWIF card on power supply assembly, relays on the IO assembly, and the main contactor K1.
133 Spindle Locked	Shot pin did not release. This is detected when spindle motion is commanded. Check the solenoid that controls the air to the lock, relay 2-8, the wiring to the sense switch, and the switch.
134 Tool Clamp Fault	Tool did not release from spindle when commanded. Check air pressure and solenoid circuit breaker CB4. Can also be caused by misadjustment of draw bar assembly.
135 X Motor Over Heat	Servo motor overheat. The temperature sensor in the motor indicates over 150 degrees F. This can be caused by an extended overload of the motor such as leaving the slide at the stops for several minutes.

136 Y Motor Over Heat	same as 135.
137 Z Motor Over Heat	same as 135.
138 A Motor Over Heat	same as 135.
139 X Motor Z Fault	Encoder marker pulse count failure. This alarm usually indicates that the encoder has been damaged and encoder position data is unreliable. This can also be caused by loose connectors at P1-P4.
140 Y Motor Z Fault	same as 139.
141 Z Motor Z Fault	same as 139.
142 A Motor Z Fault	same as 139.
143 Spindle Not Locked	Shot pin not fully engaged when a tool change operation is being performed. Check air pressure and solenoid circuit breaker CB4. This can also be caused by a fault in the sense switch that detects the position of the lock pin.
144 Time-out - Call Your Dealer	Time allocated for use prior to payment exceeded. Call your dealer.
145 X Limit Switch	Axis hit limit switch or switch disconnected. This is not normally possible as the stored stroke limits will stop the slides before they hit the limit switches. Check the wiring to the limit switches and connector P5 at the side of the main cabinet. Can also be caused by a loose encoder shaft at the back of the motor or coupling of motor to the screw.
146 Y Limit Switch	same as 145
147 Z Limit Switch	same as 145
148 A Limit Switch	Normally disabled for rotary axis.
149 Spindle Turning	Spindle not at zero speed for tool change. A signal from the spindle drive indicating that the spindle drive is stopped is not present while a tool change operation is going on.
150 Z and Tool Interlocked	Tool changer not at home and **Z** is not either at machine home or above tool. If RESET, E-STOP, or POWER OFF occurs during tool change, Z-axis motion and tool changer motion may not be safe. Check the position of the tool changer and remove the tool if possible. Re-initialize with the AUTO ALL AXES button but be sure that the pocket facing the spindle afterwards does not contain a tool.
151 Low Coolant	Coolant supply is below about five gallons or P7 is disconnected. To run without coolant, Setting 32 can be set to IGNORE.
152 Self Test Fail	Control has detected an electronics fault. All motors and solenoids are shut down. This is most likely caused by a fault of the processor board stack at the top left of the control. Call your dealer.
153 X-axis Z Ch Missing	Broken wires or encoder contamination. All servos are turned off. This can also be caused by loose connectors at P1-P4.
154 Y-axis Z Ch Missing	same as 153.
155 Z-axis Z Ch Missing	same as 153.
156 A-axis Z Ch Missing	same as 153.

157 Motor Interface PCB Failure	Internal circuit board problem. The MOTIF PCB in the processor stack is tested at power-on. Call your dealer.
158 Video/Keyboard PCB Failure	Internal circuit board problem. The VIDEO PCB in the processor stack is tested at power-on. This could also be caused by a short in the front panel membrane keypad. Call your dealer.
159 Keyboard Failure	Keyboard shorted or button pressed at power on. A power-on test of the membrane keypad has found a shorted button. It can also be caused by a short in the cable from the main cabinet or by holding a switch down during power-on.
160 Low Voltage	The line voltage to control is too low. This alarm occurs when the AC line voltage drops below 190 when wired for 230 volts or drops below 165 when wired for 208 volts.
161 X-axis Over Current or Drive Fault	Current in **X** servo motor beyond limit. Possibly caused by a stalled or overloaded motor. The servos are turned off. This can be caused by running a short distance into a mechanical stop. It can also be caused by a short in the motor or a short of one motor lead to ground.
162 Y-axis Over Current or Drive Fault	same as 161.
163 Z-axis Over Current or Drive Fault	same as 161.
164 A-axis Over Current or Drive Fault	same as 161.
165 X Zero Ret Margin Too Small	This alarm will occur if the home/limit switches move or are misadjusted. This alarm indicates that the zero return position may not be consistent from one zero return to the next. The encoder **Z** channel signal must occur between 1/8 and 7/8 revolution of where the home switch re leases. This will not turn the servos off but will stop the zero return operation.
166 Y Zero Ret Margin Too Small	Same as 165.
167 Z Zero Ret Margin Too Small	Same as 165.
168 A Zero Ret Margin Too Small	Not normally enabled for A-axis.
169 Spindle Direction Fault	Problem with rigid tapping hardware. The spindle started turning in the wrong direction.
170 Phase Loss L1-L2	Problem with incoming line voltage between legs L1 and L2. This usually indicates that there was a transient loss of input power to the machine.
171 Phase Loss L2-L3	Problem with incoming line voltage between legs L2 and L3.
172 Phase Loss L3-L1	Problem with incoming line voltage between legs L3 and L1.
173 Spindle Ref Signal Missing	The **Z** channel pulse from the spindle encoder is missing for hard tapping synchronization.

174 Tool Load Exceeded	The tool load monitor option is selected and the maximum load for a tool was exceeded in a feed. This alarm can only occur if the tool load monitor function is installed in your machine.
175 Ground Fault Detected	A ground fault condition was detected in the 115V AC supply. This can be caused by a short to ground in any of the servo motors, the tool change motors, the fans, or the oil pump.
176 Over heat Shutdown	An overheat condition persisted for 4.5minutes and caused an automatic shutdown.
177 Over voltage Shutdown	An overvoltage condition persisted for 4.5 minutes and caused an automatic shutdown.
178 Divide by Zero	Software Error; Call your dealer.
179 Low Pressure Spindle Coolant	Spindle coolant oil is low or low pressure condition in lines.
180 Tool Arm Rotation Fault	For Side Mount Tool Changer, the tool exchange operation did not sense the 180 degree rotation switch.
181 Tool Pot Position Fault	For Side Mount Tool Changer, the tool pot positioning mechanism is not working.
182 X Cable Fault	Cable from X-axis encoder does not have valid differential signals.
183 Y Cable Fault	Same as 182.
184 Z Cable Fault	Same as 182.
185 A Cable Fault	Same as 182.
186 Spindle Not Turning	Status from spindle drive indicates error.
196 Coolant Spigot Failure	Spigot failed to achieve commanded location after two (2) attempts.
201 Parameter CRC Error	Parameters lost maybe by low battery. Check for a low battery and low battery alarm.
202 Setting CRC Error	Settings lost maybe by low battery. Check for a low battery and low battery alarm.
203 Lead Screw CRC Error	Lead screw compensation tables lost maybe by low battery. Check for CRC Error low battery and low battery alarm.
204 Offset CRC Error	Offsets lost maybe by low battery. Check for a low battery and low battery alarm.
205 Programs CRC Error	Users program lost maybe by low battery. Check for a low battery and low battery alarm.
206 Internal Program Error	Software Error; Call your dealer.
207 Queue Advance Error	Software Error; Call your dealer.
208 Queue Allocation Error	Software Error; Call your dealer.

209 Queue Cutter Comp Error	Software Error; Call your dealer.
210 Insufficient Memory	Not enough memory to store users program. Check the space available in the LIST PROG mode and possibly delete some programs.
211 Odd Prog Block	Software Error; Call your dealer.
212 Program Integrity Error	Software Error; Call your dealer.
213 EPROM CRC Error	Electronics fault; Call your dealer.
214 No. of Programs Changed	Indicates that the number of programs disagrees with the internal variable that keeps count of the loaded programs. Call your dealer.
215 Free Memory PTR Changed	Indicates the amount of memory used by the programs counted in the system disagrees with the variable that points to free memory. Call your dealer.
216 EPROM Speed Failure	Indicates that an EPROM internal driver has weakened so that data read from that EPROM may be unreliable. Call your dealer.
240 Empty Prog or No EOB	Software Error; Call your dealer.
241 Invalid Code	RS-232 load bad. Data was stored as comment. Check the program being received.
242 No End	Software Error; Call your dealer.
243 Bad Number	Data entered is not a number.
244 Missing)	Comment must end with a ") ".
245 Unknown Code	Check input line or data from RS-232. This alarm can occur while editing data into a program or loading from RS-232.
246 String Too Long	Input line is too long. The data entry line must be shortened.
247 Cursor Data Base Error	Software Error; Call your dealer.
248 Number Range Error	Number entry is out of range.
249 Prog Data Begins Odd	Software Error; Call your dealer.
250 Program Data Error	Same as 249.
251 Prog Data Struct Error	Same as 249.
252 Memory Overflow	Same as 249.
253 Program Data Error	Same as 249.
254 Program Data Error	Same as 249.
255 Program Data Error	Same as 249.
256 Program Data Error	Same as 249.
257 Program Data Error	Same as 249.
258 Invalid DPRNT Format	Macro DPRNT statement not structured properly.

302 Invalid R In G02 or G03	Check your geometry with the HELP page. **R** must be less than or equal to half the distance from start to end within an accuracy of 0.0010 inches.
303 Invalid X, Y, or Z In G02 or G03	Check your geometry with the HELP page.
304 Invalid I, J, or K In G02 or G03	Check your geometry with the HELP page. Radius at start must match radius at end of arc within 0.0010 inches.
305 Invalid Q In Canned Cycle	**Q** in a canned cycle must be greater than zero.
306 Invalid I, J, K, or Q In Canned Cycle	**I, J, K,** and **Q** in a canned cycle must be greater than zero.
307 Subroutine Nesting Too Deep	Subprogram nesting is limited to nine levels. Simplify your program.
308 Nest Full	Software Error; Call your dealer.
309 ExceededMaxFeedRate	Use a lower feed rate.
310 Invalid G Code	**G** code not defined and is not a macro call.
311 Unknown Code	Possible corruption of memory by low battery. Call your dealer.
312 Program End	End of subroutine reached before M99. Need an M99 to return from subroutine.
313 No P Code In M97, M98, or G65	Must put subprogram number in **P** code.
314 Subprogram or Macro Not In Memory	Check that a subroutine is in memory or that a macro is defined.
315 Invalid P Code In M97, M98 or M99	The **P** code must be the name of a program stored in memory without a decimal point for M98 and must be a valid **N** number for M99.
316 X Over Travel Range	X-axis will exceed stored stroke limits. This is a parameter in negative direction and is machine zero in the positive direction. This will only occur during the operation of a user's program.
317 Y Over Travel Range	same as 316.
318 Z Over Travel Range	same as 316.
319 A Over Travel Range	Not normally possible with A-axis.
320 No Feed Rate Specified	Must have a valid **F** code for interpolation functions.
321 Auto Off Alarm	A fault turned off the servos automatically; occurs in debug mode only.
322 Sub Prog Without M99	Add an M99 code to the end of program called as a subroutine.
324 Delay time Range Error	**P** code in G04 is over 1000.0 or over 9999.
325 Queue Full	Control problem; call your dealer.
326 G04 Without P Code	Put a **Pn.n** for seconds or a **Pn** for milliseconds.

327 No Loop For M Code Except M97, M98	**L** code not used here. Remove **L** Code.
328 Invalid tool number	Tool number must be between 1 and 16.
329 Undefined M Code	That **M** code is not defined and is not a macro call.
330 Undefined Macro Call	Macro name O90nn not in memory. A macro call definition is in parameters and was accessed by user program but that macro was not loaded into memory.
331 Range Error	Number too large.
332 H and T Not Matched	This alarm is generated when Setting 15 is turned ON and an **H** code number in a running program does not match the tool number in the spindle. Correct the **Hn** codes, select the right tool, or turn off Setting 15.
333 X-axis Disabled	Parameters have disabled this axis. Not normally possible in VF Series CNC Mill.
334 Y-axis Disabled	same as 333.
335 Z-axis Disabled	same as 333.
336 A-axis Disabled	Parameters have disabled this axis. Must enable A-axis to program it or remove programming of A-axis. The A-axis can be disabled permanently by Parameter 43 or temporarily by Setting 30.
337 Line Referenced By **P**, not Found	Subprogram is not in memory, or **P** code is incorrect.
338 Invalid IJK and XYZ in G02 or G03	There is a problem with circle definition; check your geometry.
339 Multiple Code	Only one **M**, **X**, **Y**, **Z**, **A**, **Q**, etc. allowed in any block or two **G** codes in the same group.
340 Cutter Comp Begin With G02 or G03	Select cutter comp earlier.
341 Cutter Comp End With G02 or G03	Disable cutter comp later.
342 Cutter Comp Path Too Small	Geometry not possible. Check your geometry with the HELP page.
343 Display Queue Record Full	A block exists that is too long for displaying queue. Shorten title block.
344 Cutter Comp With G18 and G19	Cutter comp only allowed in XY plane (G17).
345 Diff Step Ratio On G17 Plane	Parameters 5 and 19 must be same value.
346 Diff Step Ratio On G18 Plane	Parameters 5 and 33 must be same value.
347 Diff Step Ratio On G19 Plane	Parameters 19 and 33 must be same value.

348 Illegal Spiral Motion	Linear axis path is too long. For helical motions, the linear path must not be more than the length of the circular component.
349 Prog Stop W/O Cancel Cutter Comp	Information message only. Fix or Ignore.
350 Cutter Comp Look Ahead Error	There are too many non-movement blocks between motions when cutter comp is being used. Remove some intervening blocks.
351 Buffered Block Range Error	Software error. Call your dealer.
352 Aux Axis Power Off	Aux **B**, **C**, **U**, **V**, or **W** axis indicate servo off. Check auxiliary axes. Status from control was OFF.
353 Aux Axis No Home	A ZERO RET has not been done yet on the aux axes. Check auxiliary axes. Status from control was LOSS.
354 Aux Axis Disconnected	Aux axes not responding. Check auxiliary axes and RS-232 connections.
355 Aux Axis Position Mismatch	Mismatch between VMC and aux axes position. Check aux axes and interfaces. Make sure no manual inputs occur to aux axes.
356 Aux Axis Travel Limit	Aux axes are attempting to travel past their limits.
357 Aux Axis Disabled	Aux axes are disabled.
358 Multiple Aux Axis	Can only move one auxiliary axis at a time.
359 Invalid I, J, or K In G12 or G13	Check your geometry with the HELP page.
360 Tool Changer Disabled	Check Parameter 57. Not a normal condition for VF Series CNC Mill.
361 Gear Change Disabled	Check Parameter 57. Not a normal condition for VF Series CNC Mill.
362 Tool Usage Alarm	Tool life limit was reached. To continue, reset the usage count in the Current Commands display and press RESET.
363 Coolant Locked Off	Override is off and program tried to turn on coolant.
364 No Circ Interp Aux Axis	Only rapid or feed is allowed with aux axes.
365 Cutter Comp Interference	G02 or G03 cut cannot be done with tool size.
366 Cutter Comp Interference	Tool doesn't fit inside of cut.
367 Cutter Comp Interference	G01 cannot be done with tool size.
368 Groove Too Small	Tool too big to enter cut.
369 Tool Too Big	Use a smaller tool for cut.

370 Pocket Definition Error	Check geometry for G150.
371 Invalid I, J, K, OR Q	Check G150.
372 Tool Change In Canned Cycle	Tool change not allowed while canned cycle is active.
373 Invalid Code in DNC	A code found in a DNC program could not be interpreted because of restrictions to DNC.
374 Missing XYZA in G31 or G36	G31 skip function requires an **X**, **Y**, **Z**, or **A** move.
375 Missing Z or H in G37	G37 auto offset skip function requires **H** code, **Z** value, and tool offset enabled. **X**, **Y**, and **A** values not allowed.
376 No Cutter Comp In Skip	Skip G31 and G37 functions cannot be used with cutter compensation.
377 No skip in Graph/Sim	Graphics mode cannot simulate skip function.
378 Skip signal found	Skip signal check code was included but skip was found when it was not expected.
379 Skip Signal Not Found	Skip signal check code was included but skip was not found when it was expected.
380 X, Y, A, or G49 not allowed in G37	G37 may only specify Z-axis and must have tool offset defined.
381 G43 or G44 not allowed in G36 or G136	Auto work offset probing must be done without tool offset.
382 D code required in G35	A **Dnn** code is required in G35 in order to store the measured tool diameter.
383 Inches Is Not Selected	G20 was specified but settings have selected metric input.
384 Metric Is Not Selected	G21 was specified but settings have selected inches.
385 Invalid L, P, or R Code In G10	G10 was used to changes offsets but **L**, **P**, or **R** code is missing or invalid.
386 Invalid Address Format	An address A...Z was used improperly.
387 Cutter Comp Not Allowed With G103	If block buffering has been limited, Cutter comp cannot be used.
388 Cutter Comp Not Allowed With G10	Coordinates cannot be altered while cutter comp is active. Move G10 outside of cutter comp enablement.
389 G17, G18, G19 Illegal in G68	Planes of rotation cannot be changed while rotation is enabled.
390 No Spindle Speed	**S** code has not been encountered. Add an **S** code.
403 RS-232 Too Many Progs	Cannot have more than 100 programs in memory.

404 RS-232 No Program Name	Need name in programs when receiving ALL; otherwise has no way to store them.
405 RS-232 Illegal Prog Name	Check files being loaded. Program name must be **Onnnn** and must be at beginning of a block.
406 RS-232 Missing Code	A receive found bad data. Check your program. The program will be stored but the bad data is turned into a comment.
407 RS-232 Invalid Code	Check your program. The program will be stored but the bad data is turned into a comment.
408 RS-232 Number Range Error	Check your program. The program will be stored but the bad data is turned into a comment.
409 RS-232 Invalid N Code	Bad Parameter or Setting data. User was loading settings or parameters and something was wrong with the data.
410 RS-232 Invalid V Code	Bad parameter or setting data. User was loading settings or parameters and something was wrong with the data.
411 RS-232 Empty Program	Check your program. Between **%** and **%** there was no program found.
412 RS-232 Unexpected End of Input	Check Your Program. An ASCII EOF code was found in the input data before program receive was complete. This is a decimal code 26.
413 RS-232 Insufficient Memory	Program received doesn't fit. Check the space available in the LIST PROG mode and possibly delete some programs.
414 RS-232 Buffer Overflow	Data sent too fast to CNC. This alarm is not normally possible as this control can keep up with even 38400 bits per second.
415 RS-232 Overrun	Data sent too fast to CNC. This alarm is not normally possible as this control can keep up with as much as 38400 bits per second.
416 RS-232 Parity error	Data received by CNC has bad parity. Check parity settings, number of data bits and speed. Also check your wiring.
417 RS-232 Framing error	Data received was garbled and proper framing bits were not found. One or more characters of the data will be lost. Check parity settings, number of data bits and speed.
418 RS-232 Break	Break condition while receiving. The sending device set the line to a break condition. This might also be caused by a simple break in the cable.
419 Invalid Function For DNC	A code found on input of a DNC program could not be interpreted.
420 Program Number Mismatch	The **O** code in the program being loaded did not match the **O** code entered at the keyboard. Warning only.
501 Too Many Assignments In One Block	Only one assignment "=" is allowed per block. Divide block in error into multiple blocks.
502 [Or = Not First Term In Expressn	An expression element was found where it was not preceded by "[" or "=", that start expressions.

503 Illegal Macro Variable Reference	A macro variable number was used that is not supported by this control, use another variable.
504 Unbalanced Paren. In Expression	Unbalanced brackets, "[" or "]", were found in an expression. Add or delete a bracket.
505 Value Stack Error	The macro expression value stack pointer is in error. Call your dealer.
506 Operand Stack Error	The macro expression operand stack pointer is in error. Call your dealer.
507 Too Few Operands On Stack	An expression operand found too few operands on the expression stack. Call your dealer.
508 Division By Zero	A division in a macro expression attempted to divide by zero. Re-configure expression.
513 Var. Ref. Not Allowed With N Or O	Alphabetic addresses N and O cannot be combined with macro variables. Do not declare N#1, etc.
514 Illegal Macro Address Reference	A macro variable was used incorrectly with an alpha address. Same as 513.
515 Too Many Conditionals In a Block	Only one conditional expression is allowed in any WHILE or IF-THEN block.
516 Illegal Conditional Or No Then	A conditional expression was found outside of an IF-THEN, WHILE, or M99 block.
517 Exprsn. Not Allowed With N Or O	A macro expression cannot be concatenated to N or O. Do not declare O[#1], etc.
518 Illegal Macro Exprsn Reference	An alpha address with expression, such as A[#1+#2], evaluated incorrectly. Same as 517.
519 Term Expected	In the evaluation of a macro expression an operand was expected and not found.
520 Operator Expected	In the evaluation of a macro expression an operator was expected and not found.
521 Illegal Functional Parameter	An illegal value was passed to a function, such as SQRT[or ASIN[.
522 Illegal Assignment Var Or Value	A variable was referenced for writing. The variable referenced is read only.
523 Conditional Reqd Prior To THEN	THEN was encountered and a conditional statement was not processed in the same block.
524 END Found With No Matching DO	An END was encountered without encountering a previous matching DO. DO-END numbers must agree.
526 Command Found On DO/END Line	A G-code command was found on a WHILE-DO or END macro block. Move the G-code to a separate block.

527 = Not Expected Or THEN Required	Only one Assignment is allowed per block, or a THEN statement is missing.
528 Parameter Precedes G65	On G65 lines all parameters must follow the G65 G-code. Place parameters after G65.
529 Illegal G65 Parameter	The addresses G, L, N, O, and P cannot be used to pass parameters.
530 Too Many I, J, or K's In G65	Only 10 occurrences of I, J, or K can occur in a G65 subroutine call. Reduce the I, J, or K count.
531 Macro Nesting Too Deep	Only four levels of macro nesting can occur. Reduce the amount of nested G65 calls.
532 Unknown Code In Pocket Pattern	Macro syntax is not allowed in a pocket pattern subroutine.
533 Macro Variable Undefined	A conditional expression evaluated to an UNDEFINED value, i.e. #0. Return True or False.
534 DO Or END Already In Use	Multiple use of a DO that has not been closed by and END in the same subroutine. Use another DO number.
535 Illegal DPRNT Statement	A DPRNT statement has been formatted improperly, or DPRNT does not begin block.
536 Command Found On DPRNT Line	A G-code was included on a DPRNT block. Make two separate blocks.
537 RS-232 Abort On DPRNT	While a DPRNT statement was executing, the RS-232 communications failed.
538 Matching END Not Found	A WHILE-DO statement does not contain a matching END statement. Add the proper END statement.

9. Part Program Storage and Edit

When using anything other than HELP or Messages function, alphanumeric key entries are displayed along the bottom line of the CRT. This is called the data entry line. When the line contains what you want to enter, press the WRITE, ALTER, or INSERT key as appropriate.

When the HELP display is selected, the alphanumeric keys are used to select one of the topics; so they are not displayed on the data entry line of the CRT.

When the Message function is selected, the cursor is positioned on the screen and you type directly into the display.

9.1 Creating Programs

To create a new program, you must be in the PROGRAM DISPLAY and LIST PROG mode. Enter **O** (letter, not number) and a four digit program number and press SELECT PROG. The selected program is the "Main" program and is the one you will see on the MEM and EDIT modes. Press EDIT to show the new program. A new program will consist of only the **Onnnn** and an EOB (;). All further entries are made by typing a letter followed by a numeric value and pressing INSERT, ALTER, or WRITE. All items entered into a program are either addressed data (a letter of the alphabet followed by a number), a comment (text surrounded by parenthesis), or the End-Of-Block (EOB or ;).

The CURSOR **up** and **down** keys can be used to search for the entered value. Simply enter the value to search for on the bottom line and press the CURSOR **up** or **down** keys. The CURSOR **up** key will search for the entered item backwards to the start of the program. The CURSOR **down** key will search forward to the end of the program. Searching also works in MEM mode. If you enter a letter without a number, the search will stop on the first use of that letter with any value.

Note that when INSERT is pressed, the new data is put in after the highlighted (reverse video) data. The CURSOR **up**, **down**, **left**, and **right** keys are used to select the highlighted item. The PAGE UP and PAGE DOWN keys move farther distances and the HOME and END keys go to the start or end of the program. All of these keys work in EDIT, MEM, and MDI modes.

A comment can be edited without entering the entire comment again. Simply move the highlighted cursor to the characters you wish to change, enter the new characters, and press ALTER, INSERT, or DELETE.

After creating a program, the name can be very easily changed by simply altering the **Onnnn** on the first line. If the maximum number of programs are already present, the message "DIR FULL" will de displayed and the program cannot be created. The maximum number of programs in memory is 200.

9.2 Editing Programs

The EDIT mode is used to make changes to a program already in memory. If a program does not exist yet, the LIST PROG mode is used to create it. A newly created program contains only the program **Onnnn** name and an EOB.

To enter the EDIT mode, press the EDIT mode key. The screen will display the current program. If no program file exists, program O0000 will be displayed. To change a program name, move the cursor to the existing **Onnnn**, type in the letter **"O"** followed by a four digit number, such as O1234, and press the ALTER key. The upper right hand screen will display the new program number. Your data will first appear in the lower left screen and will be input to the upper screen upon pressing an EDIT key button.

To enter a program from the keypad, type in the data you wish and press the INSERT key. More than one code, such as **X**, **Y**, and **Z**, can be entered before you press INSERT. After a program is entered, you may wish to change the data. Use the CURSOR keys to move the cursor to the word you wish to edit. Input your desired change in the lower left screen and then press INSERT, ALTER, or DELETE to alter the data. Use the UNDO button to reverse any changes. The UNDO button will work for the last nine entries.

The CURSOR **up** and **down** keys can be used to search for the entered value. Simply enter the value to search for on the bottom line and press the CURSOR **up** or **down** keys. The CURSOR **up** key will search for the entered item backwards to the start of the program. The CURSOR **down** key will search forward to the end of the program. Searching also works in MEM mode. If you enter a letter without a number, the search will stop on the first use of that letter with any value.

You can change to a different program while in the EDIT mode by using the CURSOR **up** and **down** keys, enter **Onnnn** on the input line and then press the CURSOR **up** and **down** keys or the **F4** key. **Onnnn** is the program you wish to change to.

As an option, the jog handle can be used to move the cursor during editing. Parameter 57 is used to turn this function on. If enabled, the handle will act like the CURSOR **left** and **right** buttons.

Background editing is also possible with this machine and is now a standard feature. If background editing is available on your control, all of the above editing functions can be used while a program is running in MEM. See Section 17 for a description of background editing.

Editing error messages:

Guarded Code	You tried to remove the **Onnnn** from start of a program.
Bad Code	A line contained invalid data or comment over 80 characters.
Editing Error	Some previous edit was not completed; fix the problem or press UNDO.
Bad Name	Program name **Onnnn** is invalid or missing.
Invalid Number	The number with an alphabet code was invalid.
Block Too Long	A block may only be 256 characters.
No Code	An insert was done without any data to insert.
Can't Undo	May only use undo for previous nine changes.
End Of Prg	End of prog EOB cannot be deleted.

9.3 Special Function Keys

The F1, F2, F3, and F4 buttons perform different functions depending on what display and mode is selected. The following is a quick summary of the **Fn** buttons:

F1 In EDIT mode and PROGRAM DISPLAY, this will start a block definition.
In LIST PROG mode, F1 will duplicate a program already stored and give it a new name from the command line.

In OFSET display, F1 will set the entered value into the offsets.

F2 In EDIT mode, PROGRAM DISPLAY, this will end a block definition.

F3 In EDIT and MDI modes, the F3 key will copy the highlighted circular help line into the data entry line at the bottom of the screen. This is useful when you want to use the solution developed for a circular motion. Push INSERT to add that circular motion command line to your program. In the calculator HELP function, this button copies the value in the calculator window to the highlighted data entry for Trig or Circular Help.

F4 When a program is not running, you are in EDIT mode, and you have entered **Onnnn** in the input line, pressing **F4** will change the currently–edited program to **Onnnn**. When in MEM mode and PROGRAM DISPLAY, this will select either BACKGROUND EDIT or PROGRAM REVIEW. You can enter BACKGROUND EDIT only when a program is running. BACKGROUND EDIT is selected by entering **Onnnn** with the program number to edit; PROGRAM REVIEW is selected with just **F4**. PROGRAM REVIEW shows the running program on the left half screen and allows the operator to review the program on the right half screen. In the calculator HELP function, this button uses the highlighted Trig, Circular, or Milling data value to load, add, subtract, multiply, or divide with the calculator.

9.4 The UNDO Key

A very powerful keyboard button available in this control is the UNDO button. When editing, this button will allow you to basically undo any changes or edits you have made but wish you hadn't. Any time you use the INSERT, ALTER, or DELETE buttons, the condition of the original block is saved and can be restored with the UNDO button. In fact, the previous nine changes can be undone in the opposite order that they were entered by pressing the UNDO button for each change that is to be backed out.

The UNDO button can be used in EDIT, BACKGROUND EDIT, and MDI. But if you change operating modes between EDIT and MDI, you cannot use the UNDO button as the list of saved data is cleared.

9.5 Block Operations

Block operations can be performed on a group of one or more blocks of the program. These operations include block duplicate, block move, and block delete. Prior to a block being defined, the bottom right of the screen show how to define a block; the F1 key is pressed when the cursor is on the first line of the block and the F2 key is pressed when the cursor is on the last line of the block.

Once a block is defined, it is displayed in reverse video and the lower right of the screen shows how to manipulate the block; the INSERT key is used to duplicate the defined block wherever the cursor is positioned, the DELETE key is used to delete the block, the ALTER key is used to move the block, and the UNDO key cancels the block definition.

When a block is defined, the cursor is indicated by the " > " symbol and is always at the beginning of a line. When a block is copied or moved, the lines are added after the block with the cursor. Only whole command lines may be move with the block functions.

11. Displays

You can select any of the following displays using the eight DISPLAY select buttons:

PRGRM	To show or edit the program selected.
POSIT	To show the axes positions.
OFSET	To show or enter working offsets.
CURNT COMDS	To show current commands and times.
ALARM / MESGS	To show alarms and user messages.
PARAM / DGNOS	To show parameters and diagnostic data.
SETNG / GRAPH	To show or enter settings OR to select graphics simulation mode.
HELP / CALC	To show the help data and calculator.

In addition to the above displays, when a program is already running, you may press LIST PROG to select a list of the programs in memory. This is useful to determine what programs can be edited in BACKGROUND EDIT. Note, if BACKGROUND EDIT is available in your machine, it is selected from the PROGRAM DISPLAY.

11.1 CRT Displays

The CRT will ALWAYS show some of the current conditions selected in the control. These are fixed status displays that describe the condition of the machine. The things displayed on the screen are:

The present selected display in top left corner,
The present selected mode in parentheses,
The presently selected program in top right corner,
The most recent line number in top right corner,
Up to 18 lines of variable display data,

SPIND %	If spindle speed override is active.
FEED %	If feed rate override is active.
RAPID %	If rapid override is active.
SINGBK	If SINGLE BLOCK is turned on.
DRYRUN	If DRY RUN is selected.
OPTSTP	If OPTIONAL STOP is turned on.
BLKDEL	If BLOCK DELETE is turned on.
FEED HOLD	If a FEED HOLD is active.
DOOR HOLD	If an open door has stopped program.
ALARM	Blinking in lower right corner when alarm occurs.
BUF	When next block is ready in continuous path.
FEED	When a feed motion in progress.
XYZA-MIR	When these axes are set to mirror image.
TOOL UNCLP	Reverse video when the tool is unclamped.
RUNNING	When a program is running.
DWELL	When a G04 is being performed.
SINGBK STOP	When a program is stopped in SINGLE BLOCK.

Error response status if button pressed is not allowed. The following are in response to operator action.

FUNCTION LOCKED	Function attempted is locked from settings.
SERVO IS ON!	Parameter change was made with servo on. This is dangerous!
WRONG MODE	Function requested is available only in another mode.
WAIT OR RESET	Cannot perform requested function until program finishes or you reset.
DISABLED AXIS	Cannot jog a disabled axis.
PLEASE WAIT	Wait until spindle is stopped.
SENDING...	RS-232 output is in process.
WAITING...	Waiting for RS-232 input.
LOADING...	Got some RS-232 input and loading it.
RS-232 ABORT	RS-232 was aborted by operator action.
RS-232 ERROR	RS-232 error (shown in alarms).
RS-232 DONE	RS-232 operation is complete.
BAD NAME	Name entered is not **Onnnn.**
ALARM ON	Cannot start an operation until alarms are reset.
PROGRAM END	Cannot remove last EOB in program.
INVALID NUMBER	Number entered is invalid.
DIVIDE BY ZERO	An attempt to divide by zero in calculator mode.
SPINDLE IN USE	Spindle is controlled by program - manual controls not available.
BLOCK TOO LONG	Block being edited would be too long.
MEMORY LOCKED	Memory lock is set in settings.
MEMORY FULL	Memory space is full.
GUARDED CODE	Cannot remove **Onnnn** at start of program.
BAD CODE	Code entered is not understood.
SERVO IS OFF	When servos are **off**, you cannot start a program.
NOT IN DRYRUN	The function requested applies to DRY RUN but not in DRY RUN mode.
NO INPUT	Cannot alter unless enter something first.
ONE PROG ONLY	Program name being selected cannot be ALL.
DIR FULL	Maximum number of programs exceeded.
PROG EXISTS	Cannot receive RS-232 an existing program.
MACRO LOCKED	Macros 9000 to 9099 are locked by setting.
PROG NOT FOUND	Requested program not in memory.
PROG READY	has been received and is ready to run.
END FOUND	End of program has been received.
NO PROG YET	Cannot Cycle Start until receives a program.
NO ZERO X	Cannot run machine until search for zero is complete on X-axis.
NO ZERO Y	Cannot run machine until search for zero is complete on Y-axis.
NO ZERO Z	Cannot run machine until search for zero is complete on Z-axis.
NO ZERO A	Cannot run machine until search for zero is complete on A-axis.
NOT AVAILABLE	Function requested is not available at that time.
DISPLAYS OFF	Indicates that M76 was used to turn off displays.
AUX AXIS BUSY	One or more auxiliary axes are busy in an operation.

There are several more responses made only in graphics mode.

M30 FOUND	End of program found and execution stopped.
LINEAR	A linear motion is being performed.
RAPID	A rapid motion is being performed.
CIRCULAR	A circular motion is being performed.

In addition to the above, the CRT display can show one of eight types of data in the 18 lines of variable display. They are:

Program Displays:
The PROGRAM DISPLAY is used to show your program while in either MEM, EDIT, or MDI modes.

Position Display:
The position display is used to select the **X**, **Y**, **Z**, or **A** axes positions in any of several coordinate systems. The PAGE UP and PAGE DOWN keys select between these.

Offsets Display:
The Offsets display is used to enter and display tool length offsets, tool radius offsets, and work offsets. The PAGE UP and PAGE DOWN keys select between these.

Current Commands Display:
The Current Commands display is used to display the Program Command Check, the Current Commands, Running Timers, Tool Life Timers, and Tool Load Monitor. The PAGE UP and PAGE DOWN keys select between these.

Alarms / Messages Displays:
The Alarms/Messages display is used to display alarms and to enter and display user messages. The second push of the ALARM button will select messages display. The CURSOR **up** and **down** buttons will display additional alarms if there is more than will fit on one page.

Parameters / Diagnostics Displays:
The Parameters display show all of the machine dependent control parameters and the Diagnostic data. The second push of the PARAM DGNOS button will select the diagnostic display. The PAGE UP and PAGE DOWN buttons will select additional data for display.

Settings / Graphics Displays:
The Settings display is used to display and change user controlled parameters. The second push of the SETNG GRAPH button will select the Graphics display. The cursor and PAGE UP and PAGE DOWN buttons will select additional settings.

Help / Calculator Displays:
The HELP display shows a mini-manual on the CRT along with a directory of available help information. Each alphabet button will select a different topic within the HELP display. The second push of the HELP button will select the Calculator display. The PAGE UP and PAGE DOWN buttons will select different calculator functions.

11.2 Program Displays

The PROGRAM DISPLAY is used to show a program being edited in EDIT mode or a program being run in MEM. In MEM mode, there is also a PROGRAM REVIEW display available.

The PROGRAM DISPLAY uses 18 lines of the text display area of the CRT to show the command blocks of a CNC program. The display is 40 positions wide and blocks that are longer than 40 positions are continued on the next line of the display.

While you are running a program, the PROGRAM REVIEW function is available. This allows you to review the program that is running. This is selected by pressing F4 while in MEM mode and PROGRAM DISPLAY. The screen is changed to an 80 column display with the normal MEM display on the left and PROGRAM REVIEW on the right. The CURSOR and PAGE UP and PAGE DOWN keys can be used to change the right hand display to a different part of the program. The left side display will show the progress of the running program. To exit PROGRAM REVIEW, select any other display.

While you are running a program, the BACKGROUND EDIT function is available as a standard feature. BACKGROUND EDIT allows you to edit any named program in memory while any program is being run in memory. BACKGROUND EDIT is selected from MEM mode in PROG display by entering **Onnnn** with the program number and pressing F4. The display will change to the selected program while still running the first program. BACKGROUND EDIT is enabled by parameters if it is available in your machine.

11.3 Position Displays

There are five position displays in this control:

Machine display:
This display is the machine coordinate system that is automatically set upon power up and the first ZERO RET. It cannot be changed by the operator or any work coordinate systems and will always tell you how far from machine zero you are. It can be used by a non modal G53 command.

Work display:
This display tells you how far your tool is away from your **X**, **Y** and **Z** zero of your programmed part. Upon power up, it will display the value in work offset G54 automatically. It can only be changed by G54 thru G59, G110 thru G129 or by a G92 command. The machine uses this coordinate system to run your part.

Distance to go:
This display is an incremental display that tells you the travel distance remaining to go before the axes stop. When the mode is in ZERO RET, this display shows a diagnostic value. When in JOG mode, this display shows the total distance jogged. In rigid tapping, this number decreases to zero at the bottom of the hole and then increases again as the reverse stroke occurs.

Operator display:
This display is for the operator/setup person to use as desired. It can be Origin'ed out as needed to show position relative to where you zero it. It is not used by the control for any positioning functions. The ORIGIN button is used to set the zero position when the Operator position display is selected and the control is in JOG mode.

A fifth display shows all of the above four in small characters. All of the others are displayed in large characters. The PAGE UP and PAGE DOWN keys will change displays. The last display selected will be shown in CURNT COMDS and SETNG GRAPH displays when they are selected.

11.4 Help Function

The HELP function is selected by pressing the HELP display button. This will bring a mini-manual up on the CRT. There are 26 topic areas selectable with the A-Z keys. This also includes a directory of the topics. The areas covered are:

A .....MACHINE INSTALLATION
B .....START/RUN//BGEDIT/POWER DOWN
C .....G/M/S/T COMMAND CODES
D .....RETURN TO THIS DIRECTORY
E .....EDITING PROGRAMS
F.....SETTING PAGE
G.....SPECIAL G CODES
H.....TROUBLE SHOOTING
I .....MDI / MANUAL DATA INPUT
J.....JOGGING / HANDLE FUNCTION
K.....CRT DISPLAY / KEYBOARD
L.....ALARMS / MESSAGES
M.....MAINTENANCE REQUIREMENTS
N.....SET UP PROCEDURES
O.....OVERRIDES: FEED/SPIN/COOLANT
P .....PARAMETERS / DIAGNOSTICS
Q.....POSITION DISPLAYS
R.....RECV / SEND PROGRAMS
S .....SAMPLE PROGRAM

T.....TOOL OFS/TOOL LIFE/LOAD
U.....GRAPHIC FUNCTION
V.....TOOL CHANGER
W.....WORK COORDINATES
X.....CREATING PROGRAMS
Y.....SPECIAL FUNCTIONS
Z.....ZERO RETURN

When the HELP display is selected, the alphanumeric keys are used to select one of the above topics; so they are not displayed on the data entry line of the CRT.

11.5 Calculator Function

The calculator function is selected by pressing the HELP key a second time. There are three calculator pages. Trig Help, Circular Help, and Milling/Tapping Help. All of these have a simple calculator and an equation solver. The **Fn** keys also allow moving of data from other displays to/from the calculator. Trig Help, Circular, and Milling Help are selected using the PAGE UP and PAGE DOWN keys.

All of the Calculator Help functions have a simple calculator for simple add, subtract, multiply, and divide operations. The operation being performed is selected with the **left** and **right** cursor arrows. The value being operated is typed on the bottom of the screen and the WRITE button is used to perform the operation.

The calculator functions only work when the calculator value window is highlighted. The CURSOR **up** and **down** buttons are used to select the Trig, Circular, or Milling data values for entry with the WRITE button.

11.6 Trigonometry Help Function

The Trig Help page will help you solve a triangular problem. You enter the lengths and the angles of a triangle and when enough data has been entered, the control will solve for the triangle and display the rest of the values. Use the CURSOR **up** and **down** buttons to select the value to be entered with Write. For inputs that have more than one solution, entering the last data value a second time will cause the next possible solution to be displayed. The F3 and F4 buttons perform special data import and export functions:

F3 In EDIT and MDI modes the F3 key will copy the highlighted circular help line into the data entry line at the bottom of the screen. This is useful when you want to use the solution developed for a circular motion. Push INSERT to add that circular motion command line to your program.

In the calculator Help function, this button copies the value in the calculator window to the highlighted data entry for Trig or Circular Help.

F4 In the calculator Help function, this button uses the highlighted Trig data value to load, add, subtract, multiply, or divide with the calculator.

11.7 Circular Interpolation Help

The Circular Help page will help you solve a circle problem. You enter the center, radius, angles, start, and end points and when enough data has been entered, the control will solve for the circular motion and display the rest of the values. In addition, it will list the four ways that such a

move could be programmed with a G02 or G03. Those four lines can be selected using the CURSOR **up** or **down** buttons and the F3 button will import the highlighted line into a program you are editing. Use the CURSOR **up** and **down** buttons to select the value to be entered with WRITE.

For inputs that have more than one solution, entering the last data value a second time will cause the next possible solution to be displayed. The CW/CCW entry is changed to the other value by pressing WRITE. The **F3** and **F4** buttons perform special data import and export functions:

F3 The **F3** key will copy the highlighted circular help line into the data entry line at the bottom of the screen. This is useful in EDIT or MDI modes where you can then push Insert to add that circular motion command line to your program.

F4 In the calculator Help function, this button uses the highlighted circular data value to load, add, subtract, multiply, or divide with the calculator.

11.8 Milling/Tapping Help

The Milling/Tapping Help page will help you solve three equations relating to milling and tapping. They are:

1) SFM = (CUTTER DIAMETER IN.) * RPM * 3.14159 / 12

2) (CHIP LOAD IN.) = (FEED IN./MIN.) / RPM / #FLUTES

3) (FEED IN./MIN.) = RPM / (THREAD PITCH)

With all three equations, you may enter all but one of the values and the control will compute the remaining value and display it. Note that the RPM value for equations 1 and 2 are the same entry.

When Metric units are selected, the units displayed change to millimeters, mm per minute, threads per mm, and meters respectively.

The **F3** and **F4** buttons perform special data import and export functions:

F3 The F3 key will copy the highlighted circular help line into the data entry line at the bottom of the screen.

This is useful in EDIT or MDI modes where you can then push INSERT to add that circular motion command line to your program.

F4 In the calculator Help function, this button uses the highlighted milling or tapping data value to load, add, subtract, multiply, or divide with the calculator.

11.9 Graphic Display Function

The Graphics function is a visual dry run of your part program without the need to move the axes and risk tool damage from programming errors. This function is far more powerful than using the DRY RUN mode because all of your work offsets, tool offsets, and travel limits can be checked before any attempt is made to move the machine. The risk of a crash during setup is greatly reduced.

To run a program in Graphics, you must be in either MEM or MDI mode.

After loading the program into memory, select MEM (or MDI) and press the "SETNG/GRAPH" key twice to select the Graphics Simulation mode. This function operates the same as if running a program on the machine except no physical machine action occurs.

The graphics screen is composed of the following areas:

DISPLAY TITLE AREA The title area is on the top left line of the screen and indicates the display (GRAPHICS), the mode you are in (MEM or MDI), the program number, and the current program line being executed. It is the same as the top line of all displays.

KEY HELP AREA The right side of the top line is the function key help area. Function keys that are currently available are displayed here with a brief description of their usage.

LOCATOR WINDOW The lower right part of the screen has two functions: it can display the whole table area and indicate where the tool is currently located during simulation, or it can be used to display four lines of the program that is being executed. The F4 key can be used to toggle between these twomodes.

TOOL PATH WINDOW In the center of the display is a large window that represents a look down perspective of the X-Y axis. It displays tool paths during a graphics simulation of a CNC program. Rapid moves are displayed as coarse dotted lines, while feed motion is displayed as fine continuous lines. The rapid path can be disabled by Setting 4. The places where a drill can or canned cycle can be executed are marked with an X. The drill mark can be disabled by Setting 5.

The tool path window can be scaled. After running a program, you can scale any portion of the tool path by pressing F2 and then using the PAGE DOWN key and the ARROW keys to select the portion of the tool path that you want to see enlarged. During this process, a reticle will appear within the TOOL PATH window and the Locator window indicating what the TOOL PATH window will represent when the zoom process is complete. The locator window always portrays the entire table with an outline of where the TOOL PATH window is zoomed to. The PAGE UP key unzooms the reticle one step. After sizing or moving the reticle, pressing the WRITE key will complete the zoom process and re-scale the TOOL PATH window. Pressing F2 and then the HOME key will expand the TOOL PATH window to cover the entire table. After the TOOL PATH window is re-scaled, the TOOL PATH window is cleared and you must rerun the program, or a portion of it, to see the tool path. The tool path is not retained in the control.

The scale and position of the TOOL PATH window is saved in Settings 65 through 68. Any scaling performed on the TOOL PATH window is retained. You can leave graphics to edit your program and when you return, your previous scaling is still in effect.

Z AXIS WINDOW A long window on the rightmost part of the screen shows the location of the Z-axis and indicates spindle movement. A horizontal line in the top part of this window represents the tool change position.

CONTROL STATUS The lower left portion of the screen displays control status. It is the same as the last four lines of all other displays.

POSITION WINDOW The location of all enabled axes can be viewed in this window. By default it is OFF. This window can be opened by pressing the F3 key. Additional presses of the F3 key will display the various position formats that the control keeps track of. This window also displays the current scale of the tool path window and the current simulated tool number. The value represented by the vertical dimension of the Tool Path window is labeled

Y-SIZE. At power-on, this will be the full Y-axis table travel. When you zoom into a table area, this value will become smaller, indicating that you are viewing a smaller portion of the table. In addition to the above, a perspective 3D graphics view is also selected by setting 3.

To exit the Graphic mode, select any other display or mode. When you exit Graphics, the graphics image is lost and must be built again by running the program.

11.10 Current Command Display

The Current Commands Display provides four types of displays. The PAGE UP and PAGE DOWN keys are used to select among:

Program Command Check,
Current Display Command,
Operation Timers,
Tool Life Timers, and
Tool Load Monitor.

The Program Command Check display will show both the programmed speed for the spindle (the last **Snnnn** command) and the actual speed command being sent to the spindle. This is shown in the Program Command Check page of the Current Commands. To get to this page, use the PAGE UP or PAGE DOWN keys. In addition to the speed, this page shows the CW, CCW, or stopped command being sent to the spindle and the current gear position. The Program Check also displays the axes' positions. The coordinates displayed are selected from the POSIT display using PAGE UP and PAGE DOWN.

11.11 Run Hours and Parts Number Display

The running hour displays are selected by PAGE UP or PAGE DOWN from the CURNT COMDS display. You may use the ORIGIN key to zero any of the three displays of on-time, cycle start time, and feed time. The CURSOR **up** and **down** buttons move the highlighted title and the ORIGIN button will zero the selected item.

Listed below the run hour display are two M30 counters that are used for counting completed parts. They may be set to zero independently to provide for the number of parts per shift and total parts. Both counters are increased when an M30 is operated.

11.12 Leaving Messages

You may leave an electronic note to yourself or anyone else by using this feature. The note may be for the operator to change tools after running a number of parts or it may be a diary for machine maintenance intervals that are performed. Data is automatically stored and maintained even in a power off state.

To enter messages, press the ALARM MESGS button twice. You may now enter data by simply typing directly onto the screen. The cancel and space keys can be used to remove existing messages. The DELETE button can be used to remove an entire line.

11.17 Diagnostic Data Display

The diagnostic data display can be selected at any time by pressing the PARAM DGNOS button a second time. There are two pages of diagnostic data and the PAGE UP and PAGE DOWN buttons are used to select between them.

A five-axis control has additional diagnostic data to be aware of:
The first page of diagnostic data has two discrete outputs that control the rotary axes brakes; they are labeled "A BRAK" and "B BRAK". When motion is commanded to either the **A** or **B** axis, the brake for that axis must first be released. To release a brake on the HAAS control, a relay is activated. These two outputs represent the activation of the brake relays. If the **A** or **B** axis brakes are disengaged, then these outputs will read high (logical 1). Normally these bits will read low (logical 0).

The second page of diagnostic data show the status of inputs from the motor interface board. Additional inputs for the expanded motor interface board are listed under "INPUTS4". These bits are monitored by the control to determine if the interface for the **B** axis is working correctly. Refer to the **Service Manual** for a description of the diagnostic page inputs.

11.18 Settings Display

The settings display can be selected at any time by pressing the SETNG GRAPH button. When the settings are displayed, changes can be made to any of the settings. There are some special functions in the settings; See section 10 for a more detailed description.

11.19 Alarms Display

The Alarms display can be selected at any time by pressing the ALARM MESGS button. When there are no alarms, the display will show NO ALARM. If there are any alarms, they will be listed with the most recent alarm at the bottom of the list. The CURSOR and PAGE UP and PAGE DOWN buttons can be used to move through a large number of alarms. The CURSOR **right** and **left** buttons can be used to turn on and off the ALARM history display.

11.20 Message Display

The Message display can be selected at any time by pressing the ALARM MESGS button a second time. This is an operator message display and has no other effect on operation of the control. Any message can be typed into the message display and called up later.

16. Direct Numerical Control (DNC)

As a standard feature, this machine is shipped with a DNC capability. With DNC, there is no limit to the size of your CNC programs. The programs are directly executed by the control as they are sent over the RS-232 interface. Note, that this is the first serial port or the top connector. Do not confuse DNC with RS-232 uploading and downloading which is described in section 13.

If you wish to use DNC, it is enabled by Parameter 57 and Setting 55. When enabled, DNC is selected by pressing MDI a second time when already in MDI. When DNC is selected, the PROGRAM DISPLAY will show:

WAITING FOR DNC...

This means that no DNC data has been received yet and you may begin sending data. You must start sending the program to the control before the START button can be pushed. After the beginning of the program is seen by the control, the display will show part of the program and a message at the bottom, left of the CRT will show DNC PROG FOUND. After the program is found, you may push CYCLE START just like running any other program from Memory.

If you try to press START before receiving a program, you will get the message: NO DNC PROG YET. The reason for not allowing the command of START before receiving the DNC program is for safety. If the operation is allowed to start from a remote location, the operator may not be present to ensure that the machine is operating safely.

While a DNC program is executing, you are not allowed to change modes. You must first press RESET to stop the program.

When the end of the DNC program is received, the message DNC END FOUND is displayed. When the DNC program is finished running, the PROGRAM DISPLAY will show the last few lines of the program. You must press RESET or exit the DNC mode before you can run any other programs. If you try to press START before RESET of the previous DNC, you will get the message: RESET FIRST.

DNC supports DRIPMODE. The control will execute one block at a time from the RS-232 port. Each block entered will be executed immediately with no block lookahead buffering. The exception is that Cutter Compensation requires three blocks of motion commands to be buffered prior to a compensated block being executed.

There are several restrictions on what can be in a DNC program. An M98 **Pnnn** may not be programmed to jump to another part of your program. An M30 is not allowed as it is not possible to start over at the beginning. The program must begin with a **%** just like any other program sent over RS-232 and the program must end with a **%**. The data rate selected for the RS-232 port by settings must be fast enough to keep up with the rate of block execution of your program. If the data rate is too slow, the tool may be stopped in a cut when you might otherwise expect continuous cutter motion. The highest RS-232 data rate available is 38400 bits per second. If the displays are turned off (M76) the maximum DNC rate is 150 blocks per second.

It is recommended that DNC be run with parity selected because an error in transmission will then be detected and will stop operation of the DNC program without crashing. The settings page is used to select parity. The recommended RS-232 settings for DNC are:

9600 or 19200 BITS PER SECOND
EVEN PARITY
2 STOP BITS
XON/XOFF

Full duplex communication during DNC is possible by using the G102 command to output axes coordinates back to the controlling computer. When DNC is running, BACKGROUND EDIT is not available.

17. Background Edit

As a standard feature, this machine is shipped with a BACKGROUND EDIT capability. With BACKGROUND EDIT, you may edit a program in memory while any other program is being run. BACKGROUND EDIT can be enabled and disabled by Parameter 57.

BACKGROUND EDIT is selected from MEM mode when in PROGRAM DISPLAY by typing **Onnnn** for the program you want to edit and pressing F4. If you do not enter the **Onnnn**, you will instead get the PROGRAM REVIEW display.

While in BACKGROUND EDIT, you may perform any of the operations available in the EDIT mode. The last five lines of the CRT will, however, display the status of the running program and the top line will show the name and line number of the running program.

Selecting any other display or pressing F4 will exit from BACKGROUND EDIT. In order to list the programs that are in memory, a new display function has been added to view the program memory list while a program is running. This display is called LIST. It is selected by pressing the LIST PROG button while a program is running. The display is just like the LIST PROG mode display but it does not allow any send, receive, copy, select, or erase functions.

All of the changes made during BACKGROUND EDIT are saved in a different memory area until the running program stops. This means that you can even edit the program that is running, or any of its subprograms, and those changes will not effect the running program.

The first time you select a program for BACKGROUND EDIT, you will get the message PROG EXISTS if the program is already in memory or NEW PROG if it is not. The NEW PROG message means that the program is being created and will be initially empty. In either case, you will then be able to edit that program. The second time you select a program for BACKGROUND EDIT without stopping the running program, you will get the message SECOND EDIT.

When you are in BACKGROUND EDIT and the running program finishes, the display will automatically change to the PROGRAM DISPLAY and will show the program that just finished running. To continue editing your program, you must select it with LIST PROG and then display it in EDIT mode.

BACKGROUND EDIT is not available from MDI or from DNC operating modes.

◆◆◆◆ End Of Programming and Operation Manual ◆◆◆◆