Section 8: Commands

Table 1: Command List

DESCRIPTION	COMMAND	PARAMETERS
AUTO:	AU,	From, to, dry run option, direct block start
BACKLASH:	BL,	Axis no., amount at center, at - limit, at + limit
CHANGE DEVICE:	CD,	Baud rate, line feed option, command echo, device option
CHANGE PROGRAM BLOCKS:	CH,	From, through
COPY PROGRAM BLOCKS:	CO,	From, through, to just after
COMMAND LOCK:	CL	
COLD START:	CS	
DISPLAY BUCKET #:	DD	
DELETE PROGRAM BLOCKS:	DE,	From, through
DIAGNOSTIC MODE:	DI	(For use by trained maintenance personnel only)
DISPLAY FIXTURE OFFSETS:	DF	
DISPLAY FEED FORWARD	DFF	
DISPLAY TOOL TABLE:	DT	
DISPLAY TOOL TIME:	DTT	
DIRECT NUMERICAL CONTROL:	DNC (DNCX),	Video option, error option, dry run, start block number
DRAW:	DR,	Displaying from, through, CRC option, list option
DISPLAY VARIABLE TABLE:	DV	
FIXTURE OFFSET:	FO,	Number, (X amount), (Y amount), (Z amount)
HOME ALL AXES:	HO	
INSERT PROGRAM BLOCKS:	IN,	From, increment
LEARN MODE:	LE,	First block number, increment
LIST PROGRAM BLOCKS:	LI,	From, through
MACROS:	MA	
MEMORY:	ME	
MENU:	MU	
MANUAL DATA INPUT:	MD	
NEW PROGRAM:	NE	(Caution: this deletes the currently active program, see PR)
NUMBER PROGRAM:	NU,	Increment for renumbering
PROGRAM MAINTENANCE:	PR,	Program number
PROGRAM PAGE EDIT:	PA	
PUNCH PROGRAM TAPE:	PU,	Data option, code option, TTY option
REINITIALIZE:	RI	

Table 1: Command List

DESCRIPTION	COMMAND	PARAMETERS
SET(parameter):	SET	Parameter code (SETX, SETY, SETZ, SETA, SETB, SETHO, SETME, SETIN)
SAVE PARAMETERS	SP	Parameter#, option#
SET(pallet):	SETPA/SETPB	This command is used to tell the control which pallet is loaded in the machine and only occurs at start-up
SETTO:	SETTO	
SETTO,#:	SETTO,#	
SET LENGTH OFFSET:	SL,	Tool number, optional change value
SUM PROGRAM:	SU,	Displaying from, through, CRC option, list option
SURVEY:	SV	(For use by trained maintenance personnel only)
SYSTEM PARAMETERS:	SETP	
TAPE READER INPUT:	TA,	Device option, error option, add at end
TOOL CHANGER HOME:	TC,	Option
TOOL PARAMETER DEFINITION:	TO,	Tool number, diameter, length off.
UTILITY:	UT,	Tool Number
VERIFICATION OF TAPE:	VT	

Auto

AU, From, To, Dry Run, Direct Block Start   This command is used instead of the AUTO key when a mid-program start or a dry run is desired. The "From" parameter specifies the first block to be executed. If it is zero, the first program block of the main program is assumed. For mid- program starts, all machine axes are automatically positioned to the location they would have been prior to the block specified, and all modal function codes specified before the starting block are automatically in effect (Spindle ON, Coolant ON, Absolute Mode, etc.). The "To" parameter specifies the block to end program execution. If it is zero, the program is executed until an M2 or M30 (Format 2) end of program. If the third parameter is a 1, 2, or 3, the program will be executed in a DRY RUN mode. In this mode, all rapid moves are under control of the feed rate override pot.

EXAMPLE: **DRY RUN OPTIONS:** If the third parameter is 1, the interpolation moves are made at the programmed feed rates and point-to-point moves are at 150 IPM.

If the third parameter is 2, the interpolation moves are made at 150 IPM and point-to-point moves are at 150 IPM.

If the third parameter is 3, the interpolation moves are at 75 IPM. and point-to-point moves are at 300 IPM.

If the fourth parameter is a 1, execution begins directly and the control will not search for modal function codes specified before the block number in the first parameter; caution must be taken. If the fourth parameter is greater than 1, the CNC begins the modal code search starting at the block # specified by the fourth parameter.

> *WARNING:* The low way lube message is not displayed when continuously looping a program in the Auto mode. The operator MUST monitor the way lube level to ensure proper fluid levels during these continuous operations.

Backlash
BL, Axis No., Amount At Center, At - Limit, At + Limit

This command is used to display axis backlash. It is also used to enter an amount of backlash for each axis into the memory of the CNC. Each axis is addressed by a number.

X = 1, Y = 2, Z = 3, A = 4, B = 5

The backlash is specified by units of one ten-thousandth of an inch. Therefore having a value of 5 would equal .0005 in decimal inches. Example: Having .0004" backlash at center of the Y axis.

Enter: BL,2,4

Change Device
CD, Baud Rate, Line Feed Option, Command Echo Option, Device Option

The primary use of this command will be to prepare the RS-232-C serial I/O port to send or receive data to or from another device such as a tape punch or another computer (see Section 14, Communications).

EXAMPLE: **BAUD RATE:**

1=110 baud	*5=1200 baud*	*9=19,200 baud*
2=150 baud	*6=2400 baud*	*10=38,400 baud*
3=300 baud	*7=4800 baud*	*11=57,600 baud*
4=600 baud	*8=9600 baud*	*12=115,200 baud*

Baud rates above 9600 should only be used with Xmodem protocol. This protocol uses error checking that is more suitable for the higher baud rates. See the communications section for an explanation of protocol types.

Note: The 57,600 and 115,200 baud rates can only be established from the Command Mode.

EXAMPLE: **LINE FEED OPTION:** 1=NO LINE FEEDS TRANSMITTED TO THE RS-232 PORT

EXAMPLE: **COMMAND ECHO OPTION:** 1=NO COMMAND ECHO TO THE RS232 PORT

EXAMPLE: **DEVICE OPTION:** 0=THE EXTERNAL COMMUNICATIONS PORT IS ACTIVE.

1=THE INTERNAL COMMUNICATIONS PORT IS ACTIVE. PC programs on the 32 MP control may use COM2 when using this option. Type BYE or CD,# to return the system to the machine RS-232 port.

EXAMPLE: *CD,3 Set the baud rate to 300*
Send data with line feeds.
Echo all commands entered at terminal.

CD,3,1 Set the baud rate to 300
Send data without line feeds.
Echo all commands entered at terminal.

CD,3,1,1,1 Set the baud rate to 300
Send data without line feeds.
Commands entered at the terminal will not be
echoed back to the terminal.
The internal communications port is active.

**Change Program
CH, From, Through**

This is a command used to change one or more blocks of the program. The CNC displays the block of data starting with the "From" parameter and proceeds by pressing the ENTER key until the "Through" parameter (optional) is reached. You do not need to retype the entire block. You may add, delete or change a character already in the block.

To add to block number 30:
TYPE COMMAND:CH,30
BLOCK DISPLAY:N30 G0
TO ADD:M8
TYPE:M8
BLOCK CORRECTED:N30 G0 M8
BLOCK DISPLAY:N30 G0 M8

TO ADD:G90
TYPE:G0 G90 (if not the G0 is replaced by G90)
BLOCK CORRECTED:N30 G0 M8 G90

To delete from block number 30:
TYPE COMMAND:CH,30
BLOCK DISPLAY:N30 G0 M8 G90
TO DELETE:M8
TYPE:M;
BLOCK CORRECTED:G0 G90
BLOCK DISPLAY:N30 G1 X9.845
TO DELETE:45
TYPE:45;
BLOCK CORRECTED:N30 G1 X9.8

To change a character in block number 30:
TYPE COMMAND:CH,30
BLOCK DISPLAY:N30 G1 X10.986
TO CHANGE:X10.986 TO X10.988
TYPE:6;8
BLOCK CORRECTED:N30 G1 X10.988
BLOCK DISPLAY:N30 G1 X10.988
TO CHANGE:X10.988 TO 10.7
TYPE:988;7
BLOCK CORRECTED:N30 G1 X10.7

When using the through parameter, the computer prompts you with each block, starting with the first parameter and ending at the second parameter. You may press the ENTER key to advance to the next block whether or not you made any changes. At any time you want to abort this mode, push the MANUAL key.

Command Lock

CL   The Command Lock menu is a method of locking out specific commands that the user does not want other users to have access to. Commands that are set to "LOCKED", will only be available if the key lock has been disabled (Key lock switch is set to the vertical position). To edit any values, the user must move a selector cursor defined by a * symbol around the screen. This selector cursor can be moved up, down, left, or right by pressing the "backspace" or "U", "enter" or "D", "L", or "R" keys respectively. To change the status of any given command, move the selector cursor to that command's position, and press the

space bar to toggle that commands lock/unlock status. Press the "manual" key to save the current settings and exit from the command lock menu.

There are three commands that will not lock/unlock without the user entering a special password. These commands are the SURVEY MENU, DIAGNOSTICS, and the MACHINE CONFIGURATION options. If these three commands are in the locked position, they will remain locked regardless of the key lock switch position. These are the commands that should only be altered by a service person.

Figure 8-1 Command Lock Menu

Copy Program

CO, From, Through, To Just After This copies one or more blocks specified by "From, Through" parameters to just after the block specified by "To Just After" parameter. The original blocks are not deleted. The copied blocks are renumbered as necessary to fit between the block specified by the third parameter and the following block.

Using the following program, copy blocks from 1 through 3 to just after block 3. Type command CO,1,3,3.

Table 2: Copy Program

ORIGINAL PROGRAM	PROGRAM AFTER COPY
N1 G0 X1.	N1 G0 X1.
N2 G1 Z-2. F25.	N2 G1 Z-2. F25.
N3 G0 Z3.	N3 G0 Z3.
N4 X6.	N3.25 G0 X1.
	N3.50 G1 Z-2. F25.
	N3.75 G0 Z3.
	N4 X6.

Cold Start

CS On System 97 machines the operator does not have to manually cold start the machine. During the power on process the machine will automatically go through the cold start procedure.

This command reinitializes the absolute table location which is required after power on. This is Machine Zero (see Section 11, Machine Coordinate System). The procedure is as follows:

1) Jog each axis to its indicator (Machine Zero), within .050 of either side.

2) Press AUTO key.

3) Inspect the cold start indicator positions, making sure that each indicator is aligned.

After the Cold Start procedure has been initialized the CNC will prompt the operator to move to the last home position or the operator can go directly to the command mode.

Press AUTO or START to move to that position and establish the Tooling Coordinate System (see SETH command). Press MANUAL to return the CNC to the COMMAND mode.

Display Bucket #

DD   Displays the bucket number and tool number table, and identifies the bucket number located at the bucket ready position with an asterisk.

1) SWAP TOOLS- Option 1 within DD is SWAP TOOLS, which will exchange the tool in the spindle for the tool in the bucket ready position. The table will be updated.

2) SORT TOOLS- Option 2 within DD will sort the tools automatically until each tool number is located in the same bucket number. Upon completion, tool number 1 will be in the spindle.

Delete Blocks

DE, From, Through   This deletes specified blocks from the program. For example:

DE,10 will only delete block 10. DE,10,1000 will delete all blocks starting with 10 through and including block 1000.

Display Feed
Forward Parameters

DFF (optional)   This command is used to display advanced feed forward parameters for tools 1 through 30. The menu at the bottom of the display is a summary of keys used to page through the feed forward table, edit the 5 feed forward parameters, and exit the display. Only 1 of 3 pages is displayed at a time, showing the parameters of 12 tools. The ENTER key advances to the next page while the BACKSPACE key pages returns to the previous page. The #1 key will allow you

to change the parameters of a tool. The space bar will exit to the tool length offset menu.

NO.	GAIN	DECEL	ACCEL	DETAIL	FEED
1	100.0000	400.0000	10.0000	0.0100	125.0000
2	100.0000	400.0000	10.0000	0.0020	100.0000
3					
4					
5					
6					
7	100.0000	400.0000	10.0000	0.0002	80.0000
8					
9					
10					
11					
12					

FEED FORWARD TABLE..PRESS MANUAL TO ABORT--
1-NEW VALUE
ENTER-NEXT PAGE   BACKSPACE- PREVIOUS PAGE   SPACE- NEXT TABLE

Figure 8-2 Advanced Feed Forward Parameters

Display Fixture Offsets

DF   This displays the current table of the 48 fixture offsets. In the example below, offset 2 has a -1.0 value for the X, Y, Z, A, and B axes.

```
NO.            X        Y        Z        A        B
1           -1.0000  -1.0000  -1.0000  -1.0000  -1.0000
2
3
4
5
6
7
8
9
10
11
12
FIXTURE OFFSET TABLE-----------------PRESS MANUAL TO ABORT--
1- NEW VALUE        2- MODIFY VALUE           3-UTILITIES
ENTER- NEXT PAGE  BACKSPACE- PREVIOUS PAGE  SPACE- NEXT TABLE
```

Figure 8-3 Display Fixture Offsets

Diagnostics

DI   This command is used by trained maintenance personnel. The Emergency Stop history can be obtained by entering DI and pressing ENTER then entering DE and pressing ENTER.

Direct Numerical Control
DNC, Video Option, Error Option, Dry Run, Start Block Num.

This command causes the CNC to execute NC code as it is received from the RS-232 port (see Section 14, DNC).

A value of 1 for the Video Option will disable the video display. The video parameter is also used to perform Mid Tape Starts. Enter the line number to begin execution from. The control then processes the program from the beginning to this line. All modal codes are processed.

A value of 1 for the Error Option disables error checking. A value of 0 checks for syntax errors such as XX or Y—, and lines of code with only a comment.

Dry Run options are the same as the AU command Dry Run options.

The Start Block Num. is the block number to begin execution of the program. The control ignores all program code prior to this block. This is the same as a

direct block start in AU. This parameter may be used in conjunction with the Mid Tape Start.

XModem Direct Numerical Control
DNCX, Video Option, Error Option, Dry Run, Start Block Num.

This command operates the same as the DNC command. This command uses the Xmodem protocol instead of the XON/XOFF protocol. The Xmodem protocol allows for long term DNC operations at higher baud rates with longer communications cables. The Xmodem protocol sends data in packets of 128 data bytes. After sending the block of data, checksum is performed. The next packet is sent if no error is detected.

Draw
DR

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

A second menu will appear, allowing the user to choose from several options. All of these options can be selected while plotting is taking place.

```
VIEW AREA: X -20.0000 TO X  20.0000
           Y -12.8000 TO Y  12.8000

  A   AUTO DRAW (RESTART SCALED)
  C   CLEAR SCREEN
  F   FULL TABLE
  M   TOGGLE DISPLAY MODE
    (NC CODE, ABSOLUTE LOCATION & MODAL G-CODES)
  O   PLOTTING OPTIONS
  S   SINGLE STEP (START TO CANCEL)
  V   TOGGLE THE VIEW [TOP]
  X   EXIT
JOG ZOOM
ENTER SELECTION {
```

Figure 8-4 Graphics Menu

A = AUTO

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

C = CLEAR

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

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

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

O = OPTIONS PLOTTING   Pressing the O key displays an additional menu allowing the user to choose from:
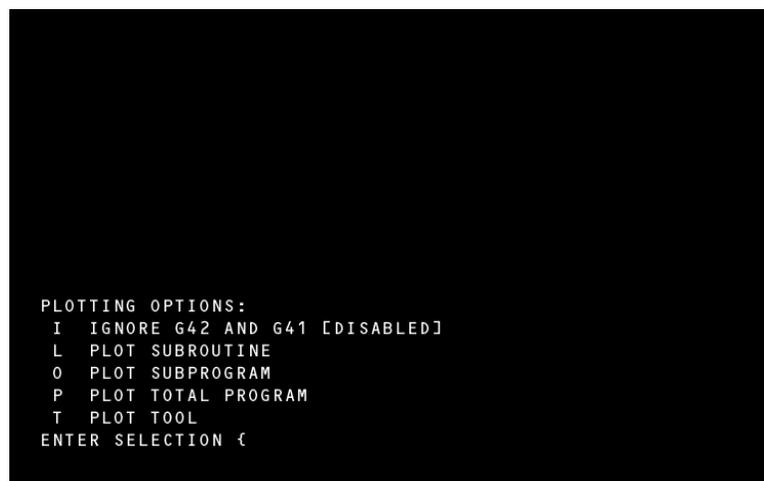

```
PLOTTING OPTIONS:
 I   IGNORE G42 AND G41 [DISABLED]
 L   PLOT SUBROUTINE
 O   PLOT SUBPROGRAM
 P   PLOT TOTAL PROGRAM
 T   PLOT TOOL
ENTER SELECTION {
```

Figure 8-5 Plotting Options Menu

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

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

V= VIEW TOP OR ISOMETRIC   The V Key can be pressed at any time during plotting to change the view from top to simple isometric view. The plotting will restart from the beginning of program. This view may not be rotated.

JOG = ZOOM   During the plotting process, or after the full plot, pressing the JOG button allows the user to ZOOM in or ZOOM out the display. The PULSE GENERATOR (the Jog Hand Wheel) now controls the position where the ZOOM BOX will be located on the screen (in this mode, JOG does not jog the machine). X and the Hand wheel moves the box left to right. Y and the Hand wheel moves the box

up and down. Z and the Hand wheel increases or decreases the size of the box. Locate the box and place it around the portion of the part path the user wants to see in a larger detail. Press the ENTER button and the part path contained in the ZOOM box will be redrawn larger. After each successive ZOOM, the pixel size representation is located to the right of the axis location of the displayed part path.

Display Tool Table

DT This command is used to display tool diameters and length offsets for tools 1 through 99. The menu at the bottom of the display is a summary of keys used to page through the tool table, edit tool data, and exit the display. Only 1 of 3 pages is displayed at a time. The ENTER key advances to the next page while the BACKSPACE key pages returns to the previous page. The #1,#2,#3, and #4 keys enable editing functions; #1 key replaces a value, #2 key increments the current value, #3 mass modifies the length incrementally, and #4 puts the display into the Utilities menu.

In program FORMAT 1, an H word applies the length factor in this table for tool length compensation and applies the diameter factor for cutter radius compensation. In program FORMAT 2, an H word applies the length factor for tool length compensation; the D word applies the diameter or radius (see the SETP command) factor for cutter radius compensation.

Exit the tool table display by pressing MANUAL.

Display Tool Time Table

DTT This command is used to display the Tool Time table. The menu at the bottom of the display is a summary of keys used to page through the tool time table, edit data, and exit the display.

⚠️ *WARNING:* Tool times become active only when appropriate parameter in SETP page has been turned on. **See SETP command**

The user may choose from the following DTT table options:

```
NO.    USED    TIME    NO.    USED    TIME    NO.    USED    TIME
1                      13                     25
2                      14                     26
3                      15                     27
4                      16                     28
5                      17                     29
6                      18                     30
7                      19                     31
8                      20                     32
9                      21                     33
10                     22                     34
11                     23                     35
12                     24                     36
TOOL TIME TABLE----------------------PRESS MANUAL TO ABORT--
1-SET USED   2-SET TIME   3-RESET ALL USED   4-RESET ALL TIME
ENTER- NEXT PAGE   BACKSPACE- PREVIOUS PAGE   SPACE- NEXT TABLE
```

Figure 8-6 DTT Table Options

1-SET USED This feature is for the expired time or USED time of the tool
2-SET TIME This feature is for the current used time or TIME the control counts
3-RESET ALL USED This feature clears expired time or USED time for all tools
4-RESET ALL TIME This feature clears current time or TIME for al tools

Depending on the SETP feature chosen, the tool times may be used to monitor USED time and or TIME. There are 3 pages to the Tool Time Table; one page is displayed at a time. The ENTER key advances to the next page while the BACKSPACE key returns to the previous page. Exit the tool table display by pressing MANUAL.

Following are the SETP options for TIMERS and a brief explanation of their use.

Timers

Figure 8-7 Timer Setup Menu

1) ALL TOOL TIMING OFF Do not check the tool time table; factory set to off.

2) DO NOT CHECK Tool timers will be active and count, will not check USED time.

3) END OF TOOL(AT M6) Tool timers active and will check USED after every M6. If US time exceeds TIME for tool specified, control will show a screen display: TOOL HAS EXPIRED!

4) AFTER EACH MOVE Tool timers active and will check USED after every move. If USED time exceeds TIME for tool specified, control will show a screen display: TOOL HAS EXPIRED!

5) AT END OF PROGRAM: TOOL HAS EXPIRED!

Select the desired option and set a value in the STT table for USED. The TIME value will be inserted by the control.

Display Variable Table Command

DV This command is used to display macro variables 1 through 100. Variables are accessible through a table display.

Fixture Offset
FO, Offset Number, X Value, Y Value, Z Value, A Value, B Value

This command enters the specified distance(s) in the fixture offset table. The offsets are relative to the Tool Coordinate System (Home). The first parameter selects one of the 48 offsets available.

EXAMPLE: *FO,2,-2.0,-2.0,2.0,100.0,205.7*
Enters for offset number 2 a value of X-2.0, Y-2.0, Z+2.0, A100.0, and B207.5

FO,2,,,-2.0
This command will not change the X,Y,A and B values. The Z parameter will be changed to a value of -2.0 (see Section 11, FIXTURE OFFSETS).

Home Axis

HO Automatic return to zero position of the Tooling Coordinate System. Note that this command operates the same as G28 in Format 1. The HO command acts as a reset button when in Format 2. R values are not reset with the HO command. This command is accomplished in one of two ways, according to the current position of the Z axis.

- If the current Z axis position is above (+) the Z0 position, the X and Y axes will move to zero first, then the Z axis will move in the negative direction to zero.
- If the current Z axis position is below (-) the Z0 position, the Z axis will move in a positive direction, to zero first, then the X and Y axes will move to zero.

After the moves are computed, the CNC enters the WAITING state. The operator can command the execution of the moves by pressing the START key or abort the moves by pressing the MANUAL key.

Insert Blocks
IN, From, Increment Insert blocks in the program. The "From" parameter specifies the starting sequence number. If "From" is not specified, 1 is assumed. The next sequence number will be determined by adding the "Increment" parameter to the present sequence number. If the "Increment" parameter is not specified, 1 is assumed. The smallest increment allowed is .001, thus allowing insertion without renumbering the entire program.

EXAMPLE: *IN*
Insert blocks starting with 1 and incremented by 1 thereafter.

EXAMPLE: *IN,2.5,.001*
Insert blocks starting with 2.5 and incremented by .001 thereafter.

EXAMPLE: *IN,10,10*
Insert blocks starting with 10 and incremented by 10 thereafter.
Each time the editor is ready to receive a block it prompts you by printing the next sequence number. Enter a block by typing the various words you desire in the block.

EXAMPLE: *N10 G2 X.707 Y.293 I.707 J-.707 F4.0*
The spaces in the above line are optional.
To exit the insert mode, press the ENTER key after the system has prompted you with a new line number.

Jog Axis
J(Axis ID) (Direction) This command places the CNC in JOG mode. The axis identification must be one of X, Y, Z, A or B. The direction is + or -. For example, to JOG Y in the negative direction you would type JY- and then press ENTER. The commas for parameter separation are not used with this command. Once in the JOG mode, the axis, direction, and feed range will be displayed. To exit the JOG mode, press the MANUAL key (see Section 7, Jog Key and the Hand Wheel).

Learn Mode
LE, First Block Number, Increment, Tool Number The primary use of this command is to enter blocks into the program from the jog mode. One example of use is the cleaning out of an irregular pocket. The first parameter is the starting block number, the second parameter is the increment of numbering (the first and second parameters are used the same as the insert command), the third parameter is the tool length offset being used. Once in the learn mode the CNC will prompt you to "PRESS JOG TO CONTINUE OR MANUAL TO EXIT". Steps for using the learn mode are as follows:

1) Enter the command LE with desired block number, increment and tool being used (a length must have been specified in the tool table or the CNC will use the total Z length from the zero position).

2) Once in the learn mode press the jog key.

3) Jog the machine to the desired position and then press the manual key. After the manual key has been pressed, the CNC displays the move to be inserted into memory. The move may be edited by typing the desired data at the line number prompt, or accepted by pressing the ENTER key.

4) Edit the move (if necessary), then press the ENTER key. The prompt PRESS JOG TO CONTINUE OR MANUAL TO EXIT is displayed.

5) To continue, press JOG and repeat steps 2-4, to exit press the MANUAL key.

List Program

LI, From, Through   Command used to list program on the CRT display.

EXAMPLE: *LI Lists the entire program*
LI,10 Lists from 10 to the end of the program
LI,20,90 Lists from 20 through 90

The speed of the display may be altered by pressing the number keys 0 through 9 while the display is in process. Each of these keys sets a different speed. "0" halts the display. Keys 1 (slowest) - 9 (fastest) will restart the display at various speeds. To exit the List mode, press the MANUAL key (see PA command for an alternate).

Macro

MA   This command is used to set the Debug and Run modes for macros. This may be used to read variable data in memory.

EXAMPLE: *SET DEBUG*

Manual Data Input

MD   This command allows the operator to enter NC data blocks that are to be executed immediately without affecting the current program in memory. Upon entering MDI, the CNC displays the current mode, tool and format (see SETP command). After entering the first data block the CNC enters the WAITING state until one of the following is pressed:

- • AUTO or START key (to execute the data)

- or -

- • MANUAL key (to abort and return to the command mode)

Every block entered thereafter is executed immediately upon pressing the ENTER key:

1) Type MD then press ENTER to put the control in the MDI mode.

2) Now type your CNC block and then press ENTER.

EXAMPLE: *G1 G91 Z-2. F100.*
This causes a Z- move of 2.0 inches at a feed rate of 100 IPM

3) At completion of each block, the VMC waits for another block of code.

EXAMPLE: *G0 G90 Z0 Returns the Z axis to the zero position*
Press the MANUAL key to return to the command mode.

Note: The MDI mode can also be entered by pressing the MANUAL key while in the command mode.

Memory

ME   This command will display the percentage of free memory in the control.

Menu

MU (when used in the command mode)   This command is used to access the menu of commands that are used in the command mode. This allows you to find a command that you do not know. Upon entering the MU command, a directory of commands and the page number on which they appear is displayed. Type the page number on which the desired command resides and then press the ENTER key. The ENTER and BACKSPACE keys are used to page forward and backward through the menu. To exit the Menu mode press MANUAL.

New Program

NE   This command is used to **remove the active program** (see PR command). The program in current memory is deleted from the control. Before removing the active program the CNC will compress memory then verify your decision by prompting you for a Y (yes) or N (no) response.

Renumber Program

NU, Increment   Renumbers the current program. The value supplied as "Increment" is used as the first block number and then is used as the step between blocks for the rest of the program. If the "Increment" parameter is left blank, the control assumes 1.

Program Page Edit

PA   This will list the currently active program. Other functions, such as word search, program editing and program execution are allowed.

The cursor is to the left of the listing and is controlled by one of the six following keys.

```
<N1 09998 (SPINDLE BREAK IN
 N2 G70 X0 G90 Y0
 N3 G91 M49
 N4 M8 S1000
 N6 S2000 (OPERATOR LOOK
 N7 X1.
 N8 S3000
 N9 X-1.
 N10 S4000
 N11 X1.
 N12 S5000
 N13 X-1.
EDIT-------------PRESS H FOR HELP, SPACE BAR FOR NEXT MENU--
U-UP         F-FUNCTIONS   C-CHANGE    S-SEARCH     N-NUMBER
D-DOWN       G-GRAPHICS    I-INSERT    R-REPLACE    O-COPY
ENTER-PAGE DOWN   BACKSPACE-PAGE UP    DEL-DELETE   P-PROGRAM
```

Figure 8-8 Program Page Edit Menu

Position the cursor to a line to execute one of the following functions by pressing the corresponding key:

C key   Change line
I key   Insert line after cursor line
DEL key   Delete cursor line or multiple lines
S key   Search for character or characters
R key   Replace program words
A key   Run cursor block only
H key   Help menu
O key   Copy lines

P key Program selection
N key Number lines
F key Function (Function) menu: move (cursor to position first)
G key Graphics menu (see Draw command)
AUTO key Begin program from beginning, from cursor line or search   models
and begin from cursor

Editing is addressed in the same manner as the CH command. Inserting new data blocks is addressed by the I key and it functions in the same manner as the IN command (see CH and IN commands). Press MANUAL to exit the listing.

Function Menu

Using the Function Menus

The Function menus are accessed through the Page Editor by pressing the F key. The screen will display 9 different function titles and function numbers. This menu consists of many independent functions that solve various geometric problems. Each is designed to help the user calculate items such as ANGLE, LINES, INTERSECTIONS, TANGENT, BLEND RADIUS, CIRCLE, and TRIANGLE. It is also designed for creating TOOL CALL or END OF PROGRAM coding and for defining FIXED CYCLES or SUBROUTINES.

Cursor Movement

Once in the Function menus, move the cursor up or down in the menu and describe the items by filling the values in. To move the cursor down press the ENTER button. To move the cursor UP press the **U** key.

If the value has been entered incorrectly, move the cursor to where the error is. Then press the backspace key until it has the incorrect data is removed. When all the data has been entered, press the **C** key to compute the geometry.

Getting Started

The user should always be aware of what position in the current program the cursor is. The user should place the cursor on a line of the current program before entering the Function menu. This line should be above the area where the calculated information needs to be inserted. When the Function menu inserts information into the Page editor, a comment is also inserted to indicate which function was used.

The Menus

Once in the function title listing, select the number of the function titles until you arrive at the individual Function menu. The cursor is located at a specific geometric question. Fill in the blank, and then press the ENTER button to move the cursor down to the next question. If the data has been entered incorrectly press the **U** key to move the cursor upward to the data and use the Backspace key to back over the information. Retype the data.

When all data has been successfully entered, press the **C** key for compute. The geometry will automatically be computed and displayed at the bottom portion of the screen.

By pressing the **D** key graphics will enlarge to cover the entire screen. To ZOOM in, press the - key; to reduce the view, press the + key. If the solution is not what the user wants, Press the **S** key for same function and retype the information until the desired solution is found. When the solution is accepted, the data may be inputted and saved to the current program after the current cursor location. Pressing the I key will insert data into the editor. This will also return the display to the Page editor. The current program will contain new code with appropriate comments from the Function menu.

An entire G code program can be written by choosing from the other functions available on the menu. Repeat the above instruction until the program is complete. Be sure to insert the appropriate feeds and speed and Z milling values. View program on the Graphics Menu before machining. Dry run program before cutting the part.

Graphics Menu   see Draw command.

Background Editing
SPACE BAR or MU   To use background editing the programmer must have the control in AUTO and
(when used in the   press the **SPACE BAR** at the keyboard. Pressing the space bar changes the
AUTO mode)   screen and the background editing menu will appear.

While in the BACKGROUND EDIT menu the programmer now has several options 1-DRYRUN OPTIONS, 2 - OFFSETS, 3-HELP.

Dry Run Options

Number	Dry Run Summary
1	Block Skip Switch Toggle: Toggles the Block Skip Switch on and off. Status of the switch is displayed on the auto mode screen as BLK when on. A block of NC code is ignored by the CNC when the block is preceded by a forward slash (/) and the Block Skip Switch is toggled ON.
2	Optional Stop Switch Toggle: Toggles the optional stop switch on and off. Functions the same as the mechanical switch on the control panel (see Section 7, Optional Stop Switch). Status of the switch is displayed on the auto mode screen as OPT when on.

Number	Dry Run Summary
3	Reset CNC Modal Values: Resets modal codes to the default values that are selected via the SETP command.
9	Dry Run Option: Program execution in a dry run mode. Interpolation moves (G1-G3) are made at the programmed feed rates and rapid moves are at 150 IPM.
10	Dry Run Option: Program execution in a dry run mode. Interpolation moves (G1-G3) are made at 150 IPM and rapid moves are at 150 IPM.
11	Dry Run Option: Program execution in a dry run mode. Interpolation moves (G1-G3) are at 75 IPM and rapid moves are at 300 IPM.
12	M,S,T Function Lockout: Program execution disabling all M, S, T functions; spindle on, coolant on etc. will be ignored.
13	Z Axis and M6 Lockout: Program execution disabling Z axis moves and tool changes. This option will reduce the control look ahead. After cancellation of this option, the Z axis will move on the next line with a Z axis programmed move.
14	No Look Ahead (For Dry Run): Normally the CNC look ahead is 90 user defined data blocks; this function reduces the look ahead to 2 blocks.
15	Display Clocks: Displays all real time clocks for power on, running, last part, current part and current time.
19	Cancel All Dry Run Modes: Restores program execution as programmed. This cancels options 9 through 14.

Offsets

Four tables can be edited: tool offset, fixture offset, tool time and macro variables tables (See DF, DT, DTT and FO commands).

To exit the menus press the MANUAL key.

Help

The Functions Menu is used for editing of the active program, a program in memory, or writing a new program, similar to PAGE EDIT (PA). The currently active program that is running in AUTO will be displayed to the screen. At the bottom of the screen the editing features are displayed as follows:

```
X    1.0000    0
Y           0   0
Z           0   0  C
 X    0        S10000    T1   G0 G17 G70 G91   H0 =0
 Y    0        F3   /          G8 G40 G80 G98  D0 =0
 Z    0 C  0   M9            E0                   00:00:13
                    09998 SPINDLE BREAK IN
 N22 M5
 N23 X0.1
<N24 M3
 N25 X0.8
 N26 M2
FUNCTIONS------------------------PRESS MANUAL TO ABORT--
U-UP     T-TOP    C-CHANGE   S-SEARCH   P-PROGRAM
D-DOWN  B-BOTTOM I-INSERT    R-REPLACE       JOG
ENTER-PAGE DOWN   BACKSPACE-PAGE UP  DEL-DELETE
```

Figure 8-9 Editing Features

U KEY moves the cursor up
D KEY moves the cursor down
T KEY moves the cursor to the top of the program
B KEY moves the cursor to the bottom of the program
C KEY changes line or edit the line on which the cursor sits
I KEY inserts below the cursor line
S KEY searches for a specified word
R KEY searches and replace a specified word
JOG KEY JOGs away from the current position

⚠ *WARNING:* Be extremely careful when making changes to the current program in auto!

**Program
Maintenance Library
PR, Program #**

This command displays the Program Maintenance Library menu. This menu is the only means to display the list of programs in memory, or copy an old program. The menu options are as follows:
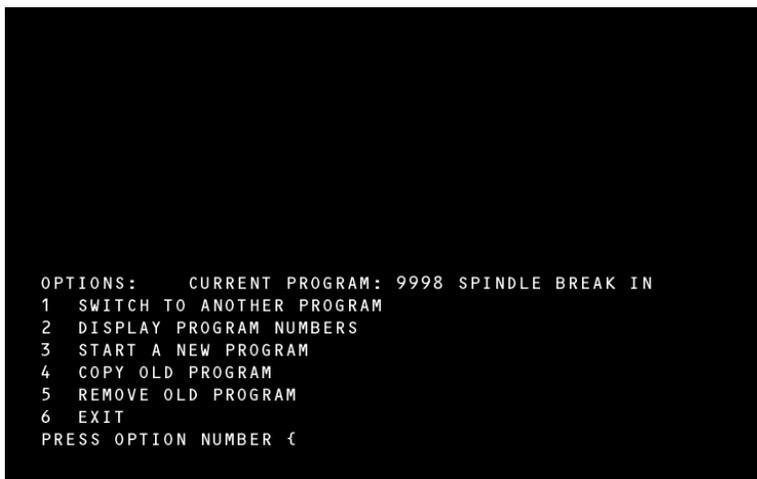
```
OPTIONS:      CURRENT PROGRAM: 9998 SPINDLE BREAK IN
1   SWITCH TO ANOTHER PROGRAM
2   DISPLAY PROGRAM NUMBERS
3   START A NEW PROGRAM
4   COPY OLD PROGRAM
5   REMOVE OLD PROGRAM
6   EXIT
PRESS OPTION NUMBER {
```

Figure 8-10 Program Maintenance Library Menu

The selection of option 5 will perform a memory compression prior to requesting confirmation of the deletion. The "Program #" parameter is used only in switching to another program stored in the memory.

EXAMPLE: *PR,22*

The above command causes program #22 to be the active program. An O word must be inserted in the active program before the Program Maintenance Library menu is displayed (see Section One, Multiple Part Program).

**Punch Program Tape
PU, Data Option,
Code Option, TTY
Option**

After selecting the desired baud rate, the PU command is used to transmit the desired data in the required format. The PU command will not punch a program that is using the no edit function. See the CD (change device) command for the communications options. The data is output in the standard left justified format. The tool offsets are output in the format of the TO command followed by the fixture offsets output in the format of the FO command. The first parameter, "Data Option", selects one of four possible formats as follows:

0 = program, tool and fixture data

1 = tool and fixture data only

2 = program data only

3 = all programs in library

4 = parameters and backlash

5 = all axis survey

The second parameter, "Code Option", selects the desired code as follows:

0 = ASCII code

1 = EIA code

The third parameter, "TTY Option", selects whether or not tape leader and nulls are sent to the receiving device. The options are as follows:
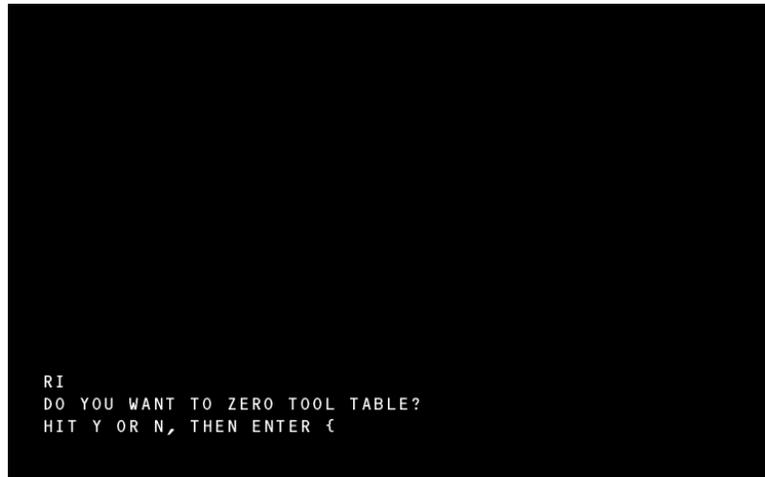
0 = computer (no leader and nulls)

3 = leaders and nulls (for teletype or paper tape punch)

Further information is covered in the Communications Section.

Reinitialize

RI This command is used to reinitialize the memory of the CNC. Three options are given as follows:


```
RI
DO YOU WANT TO ZERO TOOL TABLE?
HIT Y OR N, THEN ENTER {
```

Figure 8-11 Reinitialize Options

- DO YOU WANT TO ZERO TOOL TABLE?
- DO YOU WANT TO ZERO FIXTURE OFFSETS?
- DO YOU WANT TO REINITIALIZE MEMORY?

Enter the RI command. The CNC requires a Y (Yes) or N (No) response for each of the 3 options. The memory is cleared for each Y response. A Y response for option C requires you to Cold Start the machine (see **CS** command) and **reset tool order** (see **SETTO** command). A memory compression is accomplished by the control whether the answers to the options are Y or N.

Set Cold Start

SETCS This command is used to return the machine to the Cold Start position for power off. After entering the SETCS command the HO command must be entered. The positional display on the screen is the absolute position from the Cold Start position. If the Auto key is pressed, all axes are returned to the Cold Start position.

Set Home Position
Of All Axes

SETH The current absolute locations of all axes relative to machine zero are taken as their home positions. If command HO is issued, all axes are moved to this zero

position. When executing a CNC program, a G28 returns the axes to this position.

Set Home Position For One Axis

SET(axis)   This command is used to set home locations for individual axes.

SETX Set current absolute location of the X axis as its home position.

SETY Set current absolute location of the Y axis as its home position.

SETZ Set current absolute location of the Z axis as its home position.

SETA Set current absolute location of the A axis as its home position.

SETB Set current absolute location of the B axis as its home position.

Metric Programming

SETME   This command is used to switch from the Inch mode to the Metric mode. All input data will be processed as Millimeters. In this mode all data (tool and fixture offsets, feed rate, etc.) is to be in Metric units.

Note: This command is to be entered only when machine is at the Cold Start position.

Inch Programming

SETIN   This command is used to switch from the Metric mode to the Inch mode. All input data will be processed as inches. In this mode all data (tool and fixture offsets, feed rate, etc.) is to be in Inch units.

Note: This command is to be entered only when machine is at the Cold Start position.

Set System Parameters

SETP   This command is used to access the machine's system parameters. System parameters configure the software for the model of your machine for such things as axis travel, axis configuration, spindle adjustment, spindle drive type, tool changer capacity and pendant style. Generally these parameter settings will not change. Other parameters are for selecting modes for RS-232 communications, modal code defaults and programming formats to suit the user's preference.

The factory settings for your machine are listed on the inside of the pendant door. Update this listing any time you make a change.

The parameter settings and their values are displayed as a menu with the individual parameter with the "*" displayed at the bottom of the screen. The cursor,"*", is moved with the Enter key, Backspace key, D key, and U key. When the cursor is moved the parameter is displayed at the bottom of the screen. Change the value by typing the number corresponding to the desired setting, and then press ENTER.

All parameter settings are initialized when the machine is powered on and the Cold Start procedure is executed. The default values for modal codes are initialized when entering MDI, the AUTO mode, and in Format 1, when an M2 (end of program) is detected.

Pallet Programming
SETPA and SETPB

These commands set which pallet is currently loaded in the machine. Use SETPA (Set Pallet A) if the A pallet is loaded and SETPB (Set pallet B) if the B pallet is loaded. The software will prompt the operator when to enter these commands during the start-up procedure.

Programming Formats
Formats

There are two programming formats that are selectable by parameter settings. These formats determine the style in which a program is formatted and executed.

For the most part, Format 1 and Format 2 are identical with minor differences. Format 2 maximizes compatibility with the 6MB, 10M or 11M controls. Therefore, existing programs for these controls can be used in the CNC 88 and CNC 88 HS.

The following are the screen displays for the various formats and processors. These examples may not apply to your specific machine. The displays depicted on the following pages are typical of the screens that you will see. The specific data displayed is dependent on the processor in your machine and which parameter you have selected with the cursor.

Format 1


```
*FORMAT: 1        BAUD RATE: 2400      *SPINDLE AFTER M6: NO
 AXES: X,Y,Z      TRAVEL: X40 Y20 Z20   PENDANT: MOUNTED
 DEFAULT: G0
 DEFAULT: G90
 DEFAULT: G17     TOOL CHANGER CAP: 21   IMM. FIXED CYCLE: NO
 RPM FACTOR: 5    SPINDLE TYPE: 10,000   ORIENTATION FACTR: 10


THERE ARE TWO PROGRAM OPERATION FORMATS AVAILABLE
DO YOU WANT MAXIMUM 6MB/10M/11M COMPATIBILITY?
* 1) NO, ORIGINAL FADAL STYLE — FORMAT 1
  2) YES, FORMAT 2


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

Figure 8-12 Format 1 Screen Display


```
*DEFAULT: INCH     M8=FLOOD   M7=MIST     3 PHASE > 5% LOW: NO
 PU FORMAT: FILE
 CRC MODE: M97     BINARY BUFFERS: 255   HI TORQUE/RIGID: NO
 SPINDLE OFF: NO   TURRET FACTOR: 1      CMD MENU: SPACE
 PALLET: NO        GAIN: 100             RAMP: 100
 ASPECT: 65       *TIMERS: OFF           OVERLOAD: 2



ENTER THE DEFAULT VALUE
*1) INCH
 2) METRIC

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

Figure 8-13 Format 1 Screen Display (continued)

Figure 8-14 Format 1 Screen Display (continued)

Format 2

Figure 8-15 Format 2 Screen Display

Figure 8-16 Format 2 Screen Display (continued)

Figure 8-17 Format 2 Screen Display (continued)

Note: Depending on the parameter that the cursor is selecting, not all parameters are displayed.

Operation Formats   THERE ARE TWO PROGRAM OPERATION FORMATS AVAILABLE

Figure 8-18 Operation Formats

This parameter allows the user to select 6MB/10M/11M compatibility. The operational difference between the two formats depends upon the coding used.

AXES:X,Y,Z   ENTER THE AXIS CONFIGURATION

Figure 8-19 Axis Configuration

When selecting the A or B axes, the machine should be powered off. When the power is returned the axes will be active.

DEFAULT: G0   ENTER THE DEFAULT VALUE

Figure 8-20 Default Value G0

The code selected is active at power on and when entering the MDI mode.

DEFAULT: G90   ENTER THE DEFAULT VALUE

Figure 8-21 Default Value G90

The code selected is active at power on and when entering the MDI mode.

DEFAULT: G17   ENTER THE DEFAULT VALUE


```
FORMAT: 2        BAUD RATE: 2400        SPINDLE AFTER M6: NO
AXES: X,Y,Z      TRAVEL: X40 Y20 Z20    PENDANT: MOUNTED
DEFAULT: G0
DEFAULT: G90
*DEFAULT: G17     TOOL CHANGER CAP: 21
RPM FACTOR: 5     SPINDLE TYPE: 10,000   ORIENTATION FACTR: 10




ENTER THE DEFAULT VALUE
*1) G17
 2) G18
 3) G19

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

Figure 8-22 Default Value G17

This parameter is used to select the default machine plane.

RPM FACTOR   ENTER THE SPINDLE RPM ADJUSTMENT FACTOR


```
FORMAT: 2        BAUD RATE: 2400        SPINDLE AFTER M6: NO
AXES: X,Y,Z      TRAVEL: X40 Y20 Z20    PENDANT: MOUNTED
DEFAULT: G0
DEFAULT: G90
DEFAULT: G17      TOOL CHANGER CAP: 21
*RPM FACTOR: 5     SPINDLE TYPE: 10,000   ORIENTATION FACTR: 10




ENTER THE SPINDLE RPM ADJUSTMENT FACTOR
THE FACTOR MUST BE BETWEEN 0 AND 31


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

Figure 8-23 RPM Factor

This parameter should only be adjusted by trained maintenance personnel.

BAUD RATE: 2400   ENTER THE DEFAULT BAUD RATE (THE RATE AFTER POWER-ON)


```
FORMAT: 2        *BAUD RATE: 2400        SPINDLE AFTER M6: NO
AXES: X,Y,Z       TRAVEL: X40 Y20 Z20    PENDANT: MOUNTED
DEFAULT: G0
DEFAULT: G90
DEFAULT: G17     TOOL CHANGER CAP: 21
RPM FACTOR: 5     SPINDLE TYPE: 10,000    ORIENTATION FACTR: 10



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


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

Figure 8-24 Enter Default Baud Rate

The operator may select the desired communications baud rate.

Note: The 57,600 and 115,200 baud rates are available but not listed. These baud rates must be established from the Command Mode.

TRAVEL   ENTER X, Y, Z TRAVEL.


```
FORMAT: 2         BAUD RATE: 2400        SPINDLE AFTER M6: NO
AXES: X,Y,Z      *TRAVEL: X40 Y20 Z20    PENDANT: MOUNTED
DEFAULT: G0
DEFAULT: G90
DEFAULT: G17     TOOL CHANGER CAP: 21
RPM FACTOR: 5    SPINDLE TYPE: 10,000    ORIENTATION FACTR: 10

ENTER X,Y,Z TRAVEL
     1) X22 Y16 Z20    *7) X40 Y20 Z20     13) X30 Y16 Z20
     2) X20 Y12 Z20     8) X40 Y20 Z28     14) X30 Y16 Z28
     3) X20 Y13.5 Z20   9) X60 Y30 Z30     15) X50 Y20 Z20
     4) X22 Y13.5 Z20  10) X22 Y16 Z28     16) X50 Y20 Z28
     5) X22 Y13 Z20    11) X20 Y16 Z20     17) X80 Y30 Z30
     6) X20 Y13 Z20    12) X20 Y16 Z28


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

Figure 8-25 Travel

The machine travel is selected with this parameter.

Tool Changer Cap   ENTER THE TOOL CHANGER CAPACITY

```
FORMAT: 2       BAUD RATE: 2400       SPINDLE AFTER M6: NO
AXES: X,Y,Z     TRAVEL: X40 Y20 Z20   PENDANT: MOUNTED
DEFAULT: G0
DEFAULT: G90
DEFAULT: G17   *TOOL CHANGER CAP: 21
RPM FACTOR: 5    SPINDLE TYPE: 10,000   ORIENTATION FACTR: 10




ENTER THE TOOL CHANGER CAPACITY
 1) 16 TOOL ATC
*2) 21 TOOL ATC
 3) 30 TOOL ATC


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

Figure 8-26 Tool Changer Capacity

Select the appropriate tool changer capacity.

Timers   SELECT THE AUTOMATIC TOOL TIMER MODE

```
DEFAULT: INCH     M8=FLOOD  M7=MIST    3 PHASE > 5% LOW: NO
PU FORMAT: FILE   N-WORDS ORDERED: YES TOOL TABLE: DIAMETER
CRC MODE: M97     BINARY BUFFERS: 255  HI TORQUE/RIGID: NO
SPINDLE OFF: NO   TURRET FACTOR: 1     CMD MENU: SPACE
PALLET: NO        GAIN: 100            RAMP: 100
ASPECT: 65       *TIMERS: OFF          OVERLOAD: 2


SELECT THE AUTOMATIC TOOL TIMER MODE?
*1) ALL TOOL TIMING OFF
 2) DO NOT CHECK
 3) END OF TOOL (AT M6)
 4) AFTER EACH MOVE
 5) AT END OF PROGRAM

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

Figure 8-27 Automatic Toll Timer Mode

Select the desired option and set a value in the DTT table for USED. The TIME value will be inserted by the control. See DTT command.

Spindle Type   ENTER THE SPINDLE DRIVE TYPE & RPM

```
FORMAT: 2          BAUD RATE: 2400       SPINDLE AFTER M6: NO
AXES: X,Y,Z        TRAVEL: X40 Y20 Z20   PENDANT: MOUNTED
DEFAULT: G0
DEFAULT: G90
DEFAULT: G17       TOOL CHANGER CAP: 21
RPM FACTOR: 5   *SPINDLE TYPE: 10,000    ORIENTATION FACTR: 10




ENTER THE SPINDLE DRIVE TYPE & RPM
    *1) 10000 HI/LOW    4) 5000 HI/LOW    7) 7500 WYE/DEL.
     2) 10000 3 STEP    5) 6500 HI/LOW    8) 5000 DIRECT
     3) 15000 DIRECT    6) 3750 WYE/DEL. 9) 15K WYE/DEL.


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

Figure 8-28 Spindle Drive Type

Select the correct spindle type for the machine. This parameter is set at the factory.SPINDLE AFTER M6:

Spindle After M6   SHOULD SPINDLE COME ON AUTOMATICALLY AFTER A TOOL CHANGE WHEN THE M6 HAD TO TURN THE SPINDLE OFF?

```
FORMAT: 2          BAUD RATE: 2400       *SPINDLE AFTER M6: NO
AXES: X,Y,Z        TRAVEL: X40 Y20 Z20    PENDANT: MOUNTED
DEFAULT: G0
DEFAULT: G90
DEFAULT: G17       TOOL CHANGER CAP: 21
RPM FACTOR: 5      SPINDLE TYPE: 10,000   ORIENTATION FACTR: 10



SHOULD THE SPINDLE COME ON AUTOMATICALLY AFTER A TOOL CHANGE
WHEN THE M6 HAD TO TURN THE SPINDLE OFF?
*1) NO (FADAL RECOMMENDS THIS RESPONSE)
 2) YES
    IF YES, ALL PERSONNEL MUST BE AWARE OF THE POSSIBILITY OF
    OVERSPEEDING A TOOL BEFORE THE NEW S-WORD IS ENCOUNTERED


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

Figure 8-29 Spindle After M6

When this parameter is selected as YES, the spindle automatically turns on after the tool change. The spindle comes on at the last programmed spindle speed. This may cause an overspeed of the next tool. **It is recommended that this parameter is set to number 1.**

Pendant   ENTER THE PENDANT STYLE

```
FORMAT: 2        BAUD RATE: 2400        SPINDLE AFTER M6: NO
AXES: X,Y,Z      TRAVEL: X40 Y20 Z20    *PENDANT: MOUNTED
DEFAULT: G0
DEFAULT: G90
DEFAULT: G17     TOOL CHANGER CAP: 21
RPM FACTOR: 5    SPINDLE TYPE: 10,000   ORIENTATION FACTR: 10



ENTER THE PENDANT STYLE
 1) KEYBOARD BESIDE THE VIDEO TUBE
 2) KEYBOARD BELOW THE VIDEO TUBE
*3) SAME AS 2 BUT MOUNTED ON FULL-ENCLOSURE CHIP GUARDS


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

Figure 8-30 Pendant

Select the appropriate pendant location for the machine. When option two is selected the table may make a Y axis positive move before a tool change. This occurs only when the Y axis is five inches or more, in the negative direction, from the cold start position.

Imm. Fixed Cycle   SHOULD A FIXED CYCLE EXECUTE IMMEDIATELY?

```
FORMAT: 1        BAUD RATE: 2400        SPINDLE AFTER M6: NO
AXES: X,Y,Z      TRAVEL: X40 Y20 Z20    PENDANT: MOUNTED
DEFAULT: G0
DEFAULT: G90
DEFAULT: G17     TOOL CHANGER CAP: 21  *IMM. FIXED CYCLE: NO
RPM FACTOR: 5    SPINDLE TYPE: 10,000   ORIENTATION FACTR: 10



SHOULD A FIXED CYCLE EXECUTE IMMEDIATELY?
 1) YES
*2) NO, ONLY IF X OR Y DIMENSION WORDS ARE
        IN THE DEFINITION LINE


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

Figure 8-31 Imm. Fixed Cycle

A YES response causes a fixed cycle to be executed immediately upon definition at the current axis location. A NO response requires axis motion to activate the fixed cycle.

Orientation Factor   ENTER THE SPINDLE RPM ADJUSTMENT FACTOR

THE FACTOR MUST BE BETWEEN 0 AND 31

```
FORMAT: 1          BAUD RATE: 2400        SPINDLE AFTER M6: NO
AXES: X,Y,Z        TRAVEL: X40 Y20 Z20    PENDANT: MOUNTED
DEFAULT: G0
DEFAULT: G90
DEFAULT: G17       TOOL CHANGER CAP: 21   IMM. FIXED CYCLE: NO
RPM FACTOR: 5      SPINDLE TYPE: 10,000   *ORIENTATION FACTR: 10




ENTER THE SPINDLE ORIENTATION RPM ADJUSTMENT FACTOR
THE FACTOR MUST BE BETWEEN 0 AND 31

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

Figure 8-32 Orientation Factor

This parameter should only be adjusted by trained maintenance personnel.

DEFAULT: INCH   ENTER THE DEFAULT VALUE

```
*DEFAULT: INCH     M8=FLOOD  M7=MIST     3 PHASE > 5% LOW: NO
 PU FORMAT: FILE
 CRC MODE: M97     BINARY BUFFERS: 255   HI TORQUE/RIGID: NO
 SPINDLE OFF: NO   TURRET FACTOR: 1      CMD MENU: SPACE
 PALLET: NO        GAIN: 100             RAMP: 100
 ASPECT: 65        TIMERS: OFF           OVERLOAD: 2




ENTER THE DEFAULT VALUE
*1) INCH
 2) METRIC

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

Figure 8-33 Default Value: Inch

The operator must select the inch or metric mode for the machine. The G70, G71, G20, and G21 check this setting to verify the operational mode.

PU FORMAT   SELECT PUNCH OUTPUT FORMAT


```
 DEFAULT: INCH     M8=FLOOD  M7=MIST     3 PHASE > 5% LOW: NO
*PU FORMAT: FILE
 CRC MODE: M97     BINARY BUFFERS: 255  HI TORQUE/RIGID: NO
 SPINDLE OFF: NO   TURRET FACTOR: 1     CMD MENU: SPACE
 PALLET: NO        GAIN: 100            RAMP: 100
 ASPECT: 65        TIMERS: OFF          OVERLOAD: 2




 SELECT PUNCH OUTPUT FORMAT
  1) PUNCHED TAPE FORMAT (TELETYPE STYLE)
 *2) COMPUTER FILE FORMAT (NO NULLS)

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

Figure 8-34 PU Format

This parameter is set to file for computer use. The punch tape format is used when a tape reader is employed.

CRC Mode   ENTER THE DEFAULT OUTSIDE CORNER MOVEMENT


```
 DEFAULT: INCH     M8=FLOOD  M7=MIST     3 PHASE > 5% LOW: NO
 PU FORMAT: FILE
*CRC MODE: M97     BINARY BUFFERS: 255  HI TORQUE/RIGID: NO
 SPINDLE OFF: NO   TURRET FACTOR: 1     CMD MENU: SPACE
 PALLET: NO        GAIN: 100            RAMP: 100
 ASPECT: 65        TIMERS: OFF          OVERLOAD: 2




 ENTER THE DEFAULT OUTSIDE CORNER MOVEMENT
  1) M96 - ROLL
 *2) M97 - INTERSECTIONAL

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

Figure 8-35 CRC Mode

This parameter selects the default mode for intersectional cutter radius compensation.

Pallet   DO YOU HAVE A PALLET CHANGER?

```
 DEFAULT: INCH     M8=FLOOD   M7=MIST     3 PHASE > 5% LOW: NO
 PU FORMAT: FILE
 CRC MODE: M97    BINARY BUFFERS: 255   HI TORQUE/RIGID: NO
 SPINDLE OFF: NO  TURRET FACTOR: 1     CMD MENU: SPACE
*PALLET: NO        GAIN: 100            RAMP: 100
 ASPECT: 65        TIMERS: OFF          OVERLOAD: 2




 DO YOU HAVE A PALLET CHANGER
*1) NO
 2) YES

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

Figure 8-36 Pallet

Select the option appropriate for the machine.

M7-FLOOD, M8-MIST   ENTER M7, M8 PREFERENCE

```
 DEFAULT: INCH    *M8=FLOOD   M7=MIST     3 PHASE > 5% LOW: NO
 PU FORMAT: FILE
 CRC MODE: M97    BINARY BUFFERS: 255   HI TORQUE/RIGID: NO
 SPINDLE OFF: NO  TURRET FACTOR: 1     CMD MENU: SPACE
 PALLET: NO        GAIN: 100            RAMP: 100
 ASPECT: 65        TIMERS: OFF          OVERLOAD: 2




 ENTER M7,M8 PREFERENCE
 1) M7 IS FLOOD COOLANT, M8 IS MIST
*2) M8 IS FLOOD COOLANT, M7 IS MIST

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

Figure 8-37 M7-Flood, M8-Mist

The operator may select either M7 or M8 as the flood coolant code.

Binary Buffers 255   SELECT THE NUMBER OF BINARY BUFFERS FOR CNC LOOK-AHEAD.

```
DEFAULT: INCH     M8=FLOOD  M7=MIST    3 PHASE > 5% LOW: NO
PU FORMAT: FILE
CRC MODE: M97    *BINARY BUFFERS: 255  HI TORQUE/RIGID: NO
SPINDLE OFF: NO  TURRET FACTOR: 1      CMD MENU: SPACE
PALLET: NO          GAIN: 100          RAMP: 100
ASPECT: 65          TIMERS: OFF        OVERLOAD: 2




ENTER THE NUMBER OF BINARY BUFFERS FOR CNC LOOK-AHEAD
 1) 15           3) 50          *5) 255
 2) 30           4) 120

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

Figure 8-38 Binary Buffers: 255

The BINARY BUFFERS parameter can be changed to increase or decrease the control look ahead. A binary buffer is a block of memory that has been processed by the control and is waiting for execution. One line of code may produce numerous binary blocks. A simple drill code generates three binary blocks: the XY position, the Z down, and Z up. The number of binary blocks can be set at 15, 30, 50, 100, or 255. The factory sets the buffers at 255. This is most effective for programs with many small moves that must be executed rapidly. This parameter helps the Run Time Menu to be used more effectively. The smaller the buffers the more quickly the Run Time Menu changes will take effect in the program.

Turret Factor   ENTER THE ENGAGEMENT FACTOR FOR THE TOOL TURRET GENEVA GEAR

THE FACTOR MUST BE BETWEEN 1 AND 50


```
DEFAULT: INCH     M8=FLOOD  M7=MIST    3 PHASE > 5% LOW: NO
PU FORMAT: FILE
CRC MODE: M97     BINARY BUFFERS: 255  HI TORQUE/RIGID: NO
SPINDLE OFF: NO  *TURRET FACTOR: 1     CMD MENU: SPACE
PALLET: NO        GAIN: 100            RAMP: 100
ASPECT: 65        TIMERS: OFF          OVERLOAD: 2




ENTER THE ENGAGEMENT FACTOR FOR THE TOOL TURRET GENEVA GEAR
THE FACTOR MUST BE BETWEEN 1 AND 50 (1 FOR SERVO TURRET)

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

Figure 8-39 Turret Factor

This parameter is set at the factory. For VMCs equipped with the Servo-Turret, this factor MUST always be 1.

Gain   ENTER THE GAIN FACTOR FOR RIGID TAPPING

                           THE FACTOR MUST BE BETWEEN 0 AND 255

Figure 8-40 Gain

This parameter affects the spindle response during rigid tapping. The higher the number the faster the spindle turns in relation to the feed rate. When the speed is too fast the thread may be too loose.

3 Phase 5% Low: No   IS YOUR 3 PHASE POWER MORE THAN 5% LOW?

Figure 8-41 3 Phase 5% Low

The selection chosen is based upon the building power supply.

High Torque/Rigid Tap

DO YOU HAVE THE HIGH TORQUE OR RIGID TAP OPTION?

```
DEFAULT: INCH     M8=FLOOD  M7=MIST    3 PHASE > 5% LOW: NO
PU FORMAT: FILE
CRC MODE: M97     BINARY BUFFERS: 255 *HI TORQUE/RIGID: NO
SPINDLE OFF: NO   TURRET FACTOR: 1     CMD MENU: SPACE
PALLET: NO        GAIN: 100            RAMP: 100
ASPECT: 65        TIMERS: OFF          OVERLOAD: 2




DO YOU HAVE THE HIGH TORQUE OR RIGID TAP OPTION?
*1) NO
 2) YES


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

Figure 8-42 HighTorque/Rigid Tap

This parameter is set at the factory.

CMD Menu

TURN COMMAND MENUS:

```
DEFAULT: INCH     M8=FLOOD   M7=MIST     3 PHASE > 5% LOW: NO
PU FORMAT: FILE
CRC MODE: M97     BINARY BUFFERS: 255  HI TORQUE/RIGID: NO
SPINDLE OFF: NO   TURRET FACTOR: 1    *CMD MENU: SPACE
PALLET: NO        GAIN: 100            RAMP: 100
ASPECT: 65        TIMERS: OFF          OVERLOAD: 2




TURN COMMAND MENUS:
 1) OFF
 2) ON
*3) TOGGLE ON WITH SPACE BAR


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

Figure 8-43 CMD Menu

The operator may select the command menu structure.

Ramp   ENTER THE RAMP FACTOR FOR RIGID TAPPING

THE FACTOR MUST BE BETWEEN 0 AND 255

```
DEFAULT: INCH      M8=FLOOD   M7=MIST      3 PHASE > 5% LOW: NO
PU FORMAT: FILE
CRC MODE: M97      BINARY BUFFERS: 255   HI TORQUE/RIGID: NO
SPINDLE OFF: NO   TURRET FACTOR: 1      CMD MENU: SPACE
PALLET: NO         GAIN: 100            *RAMP: 100
ASPECT: 65         TIMERS: OFF          OVERLOAD: 2




ENTER THE RAMP FACTOR FOR RIGID TAPPING
THE FACTOR MUST BE BETWEEN 0 AND 255


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

Figure 8-44 Ramp

This parameter sets the speed at which the spindle accelerates during rigid tapping.

A-Axis Ratio   ENTER A-AXIS RATIO

```
FORMAT: 2          BAUD RATE: 2400        SPINDLE AFTER M6: NO
AXES: X,Y,Z,A,B TRAVEL: X40 Y20 Z20   PENDANT: MOUNTED
DEFAULT: G0      *A-AXIS-RATIO: 90:1     M60/A-AXIS BRAKE: YES
DEFAULT: G90     B-AXIS RATIO: 90:1      M62/B-AXIS BRAKE: YES
DEFAULT: G17     TOOL CHANGER CAP: 21
RPM FACTOR: 5    SPINDLE TYPE: 10,000    ORIENTATION FACTR: 10



ENTER A-AXIS RATIO
    *1) 90:1          4) 120:1          7) 60:1
     2) 180:1         5) 72:1
     3) 360:1         6) COOLANT


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

Figure 8-45 A-Axis Ratio

Select the appropriate option for the rotary table being used.

B-Axis Ratio   ENTER B-AXIS RATIO

Figure 8-46 B-Axis Ratio

Select the appropriate option for the rotary table being used.

M60/A-Axis Brake   DOES M60 TURN ON THE A-AXIS BRAKE?

Figure 8-47 M60/A-Axis Brake

Select the option desired to activate or de-activate the air brake for the axis.

M62/B-Axis Brake   DOES M62 TURN ON THE B-AXIS BRAKE?

```
FORMAT: 2        BAUD RATE: 2400       SPINDLE AFTER M6: NO
AXES: X,Y,Z,A,B  TRAVEL: X40 Y20 Z20   PENDANT: MOUNTED
DEFAULT: G0      A-AXIS-RATIO: 90:1    M60/A-AXIS BRAKE: YES
DEFAULT: G90     B-AXIS RATIO: 90:1    *M62/B-AXIS BRAKE: YES
DEFAULT: G17     TOOL CHANGER CAP: 21
RPM FACTOR: 5    SPINDLE TYPE: 10,000  ORIENTATION FACTR: 10




DOES M62 TURN ON THE B-AXIS BRAKE?
*1) YES
 2) NO


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

Figure 8-48 M62/B-Axis Brake

Select the option desired to activate or de-activate the air brake for the axis.

N-Words Ordered   ENTER THE N-WORD SEQUENCE CONFIGURATION

```
DEFAULT: INCH     M8=FLOOD  M7=MIST    3 PHASE > 5% LOW: NO
PU FORMAT: FILE *N-WORDS ORDERED: YES TOOL TABLE: DIAMETER
CRC MODE: M97     BINARY BUFFERS: 255  HI TORQUE/RIGID: NO
SPINDLE OFF: NO   TURRET FACTOR: 1     CMD MENU: SPACE
PALLET: NO        GAIN: 100            RAMP: 100
ASPECT: 65        TIMERS: OFF          OVERLOAD: 2




ENTER THE N-WORD SEQUENCE CCONFIGURATION
*1) THE N-WORDS ARE IN ASCENDING NUMERICAL ORDER OR
    THE PROGRAM WILL BE RENUMBERED AFTER INPUT.
 2) THE N-WORDS ARE NOT IN ORDER.


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

Figure 8-49 N-Words Ordered

The CNC 88 requires each block of NC code to have sequence numbers in numerical order. Since the 6MB/10M/11M controls do not require block numbers in numerical order, select option number 2. Upon tape input the CNC will add sequence numbers for reference. Otherwise, after tape input the program must be renumbered if the sequence numbers are not in numerical order.

Tool Table   WILL THE TOOL COMPENSATION TABLE HAVE THE RADIUS OR DIAMETER?

```
DEFAULT: INCH     M8=FLOOD   M7=MIST      3 PHASE > 5% LOW: NO
PU FORMAT: FILE   N-WORDS ORDERED: YES *TOOL TABLE: DIAMETER
CRC MODE: M97     BINARY BUFFERS: 255   HI TORQUE/RIGID: NO
SPINDLE OFF: NO   TURRET FACTOR: 1      CMD MENU: SPACE
PALLET: NO        GAIN: 100             RAMP: 100
ASPECT: 65        TIMERS: OFF           OVERLOAD: 2




WILL THE TOOL COMPENSATION TABLE HAVE THE RADIUS OR DIAMETER?
*1) DIAMETER
 2) RADIUS

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

Figure 8-50 Tool Table

The cutter offset specification in the tool compensation table may be defined as a diameter or radius. The SETP mode is exited by pressing the MANUAL key. If new values were selected, the CNC requires that you perform the Cold Start procedure (see CS command).

Set Turret Order

SETTO   This command is used to set the tool turret location. The current turret location is established as number 1. The remaining locations are numbered sequentially in a clockwise order, looking from the bottom of the turret.

SETTO without a number parameter following resets all of the tool numbers to that of the bucket numbers, regardless of where the tools are located, and sets bucket 1 at t he bucket ready position, and tool 1 in the spindle.

1) Using Turret CW or Turret CCW, rotate bucket 1 to the bucket ready position.

2) From the <ENTER NEXT COMMAND> line, type SETTO.

3) All of the tool numbers will be reset to that of the bucket numbers. Tool number 1 is in the spindle.

4) Check the table in DD.

5) If Turret rotates in the incorrect direction, the Turret Motor may need to be rephased.

SETTO,# SETTO,# is used to reset the Turret locations by specifying that "#" is the number of the bucket (not the tool number) located at the bucket ready position and ready to be exchanged. The remaining bucket and tool numbers are recovered as the sequence is retained.

1) Rotate the Turret using Turret CW or Turret CCW at ;east one position until the desired bucket number (not tool number) is at the bucket ready position.

2) If Turret rotates in the incorrect direction, the Turret Motoe may need to be rephased.

3) From the <ENTER NEXT COMMAND> line, type SETTO,# where # is the bucket number of the bucket now at the bucket ready position, and ready to exchange tools.

4) The sequence of the remaining tools in the Turret is not changed, and the new bucket numbers are updated in the DD table. The asterisk identifies the bucket in the bucket ready position.

5) The SETTO,# procedure may be repeated as many times as needed.

Set Tool Length Offset

SL, Tool Number, Optional Change Value This commands automatic entry of tool length compensations. The procedure is as follows:

1) Set the home position using the SET(parameter) command.

2) Install tool in the spindle.

3) Press the MANUAL key to enter the command mode.

4) Press the JOG key to enter the jog mode.

5) Jog the Z axis until the tool is in the proper location.

6) Press the MANUAL key to enter the command mode.

7) Type SL comma and the tool number.

8) Press the ENTER key to insert this location in the tool table.

If the second parameter is a number other than zero, the current location of the Z axis is ignored and the current value of the length offset in the tool table is incremented by the value specified by the second parameter. For Example; If

tool #1 originally has a length offset value of -10.000 and the command SL,1,-.025 is entered, the NEW offset will be -10.025.

Sum Program

SU, Display From, Through, CRC Option, Display Option

Sum the X, Y, Z, A, and B moves in the current program and display the final location, relative to the programmed home position as: X= Y= Z= A= B=.

During the SUM process, the moves that the computer is processing are displayed if the fourth parameter is 1, 2, or 3 (see the following examples). This information can be outputted to a computer or paper tape punch (see CD command).

The speed at which the processed program is displayed may be altered by pressing the keys 0 through 9. Each of these keys sets a different speed. "0" halts the display, while keys 1 (slowest) - 9 (fastest) restart the display at various scroll speeds. The "Display From" parameter indicates the first line to display after beginning the processing from the start of the program. The "Through" parameter indicates the last line to process.

The "CRC Option" parameter indicates whether to process CRC. A parameter value of 1 will ignore CRC. A parameter value of 0 processes CRC.

The "Display Option" parameter indicates the display mode to use.

A 1 displays the incremental move only.
A 2 displays incremental moves and absolute locations.
A 3 displays incremental moves, absolute locations, and active modal G codes.

To abort the Sum process, press the MANUAL key.

SU,0,0,0,1

Sum the entire program, check the CRC generated moves. Display only the incremental moves on the left side of the screen. Display the absolute location of the end of the program as X= Y= Z= A= B=.

SU,10,0,1,1

Sum the entire program, ignoring CRC generated moves. Start displaying from block number 10. Display only the incremental moves on the left side of the screen. Display the absolute location of the end of the program as X= Y= Z= A= B=.

SU,10,50,0,2

Sum from beginning, until block number 50, displaying all CRC generated moves. Start displaying from block 10. Display the incremental moves on the

left side of the screen and the absolute locations of the moves on the right side of the screen. Display the absolute location of the end of the program as

X= Y= Z= A= B=.

Survey

SV   The survey command SV is a utility that manages the axis compensation data. Each axis controller stores the screw compensation, zero offset for the scales, and the servo gain settings. The survey menu automatically loads the survey into memory, starting with the default X Axis survey. If no survey exists, a new empty survey will be created. The survey is then automatically displayed on the screen in groups of 40 values at a time. If more than 40 values in any given survey exist, there will be an additional option located at the bottom of the first column called "survey values". This option will allow the user to toggle between Page 1 (the first 40 values) and Page 2 (the remaining values), of the survey. The zero offset and gain options are also displayed on the bottom of the first column.

A survey may be saved by either exiting out of the survey menu by pressing the Manual key, or by selecting another survey to edit. On exiting, the CNC will automatically cold start to enable the changes. If a survey has not been altered before exiting, the CNC will not cold start. This new feature allows the user to go into the survey menu just to look at the current settings, without having to wait for a cold start when exiting.

If a mistake has been made and a user wants to reload an axis without saving the changes, move the selector cursor to display the "Enter Axis ID" prompt. At this prompt, re-enter the axis that is currently being edited. A "Do You Want To

Save Survey Before Re-Loading (Y/N)" message will appear. Press "N" to reload the survey without saving the changes.


```
* [ X ] AXIS CURRENT SURVEY SETTINGS.

    [-9] -1      [ 1 ]  2
    [-8]  2      [ 2 ]  1
    [-7] -3      [ 3 ]  0
    [-6] -2      [ 4 ] -1
    [-5] -1      [ 5 ] -1
    [-4]  0      [ 6 ] -2
    [-3]  0      [ 7 ] -2
    [-2]  1      [ 8 ] -1
    [-1]  2      [ 9 ] -1
    [ 0]  3      [ 10]  0
    [ ZERO OFFSET ] 0
    [ GAIN OPTION ] NORMAL

ENTER AXIS ID (X,Y,Z,A OR B)
```

Figure 8-51 Survey Settings

WARNING: This command should ONLY be used by trained MAINTENANCE PERSONNEL.

Tape (Program) Input

TA, Device Option, Error Option, Add at the End Option

The TA command first clears the current program and prepares to receive program data blocks, tool offsets, or fixture offsets. If the current program has an O word, it is placed into machine memory. If there is no O word in the program, it is deleted from the machine memory.

The first parameter determines whether the data is from the tape reader of the machine or from the RS-232-C port.

    0 = Input from Tape Reader.
    1 = Input through RS-232 port.
    2 = Input maintenance programs from machine memory.
    3 = Input probe programs from machine memory.

The second parameter selects the three possible error options.

> 1 = Indicates that the program HAS parity errors.
> 2 = Ignores errors on input and gives an error count after input.

Note: A block of code containing an error is ignored, an error count upon completion of input displays the number of blocks having errors.

> 3 = Allows the control to accept programs from another
> CNC control.

Enter a value of 1 for the third parameter if the input is to be added at the end of the current program. After this input, the NU command is required before editing or execution.

Note: The control will automatically delete a program from the library when the file number of the program being received is same as the one in the library.

Tool Changer Open

TC,1 This command is used to open the tool changer to manually insert tools. If the "Option" parameter is a 1, the tool changer moves to position under the spindle. The spindle rises to release the tool (if applicable) and waits. Press MANUAL to return the spindle and retract the turret. If the parameter is a 2, the machine does NOT wait with the spindle raised. The spindle and turret immediately return.

Tool Parameter Definition

TO, Number, Diameter, Length Offset This command is used to manually enter data in the tool compensation table. The table contains tool diameter and length offsets for 99 tools.

EXAMPLE: *TO,6,.75,-2.75*
Enters data for tool 6; a diameter of .75; a -2.75 length offset

EXAMPLE: *TO,6,,-2.75*
Enters data for tool 6; the current diameter/radius of tool #6 is unchanged; a -2.75 length offset

In programming Format 1, the CNC summons these values by use of an H word programmed in a block of NC code. The length offset value is applied

immediately when the H word is detected during program execution or in MDI. The diameter offset value is applied when a G41 or G42 is detected, compensating the value of the last designated H word.

In programming Format 2, the length offset is applied immediately when the H word is detected. The D word is used to apply the diameter/radius value for cutter compensation when a G41 or G42 is coded.

Enter the command DT to examine all tool data.

Tool Loading Procedure

1. From MDI mode, (MANUAL DATA INPUT), type M19, press ENTER and START to orient Spindle.

2. Press MANUAL to switch to the <ENTER NEXT COMMAND> mode.

3. Rotate the Turret using Turret CW or Turret CCW keys until bucket 1 is in the bucket ready position.

4. Type SETTO to reset the bucket numbers with bucket 1 at bucket ready position.

5. Return to MDI by pressing MANUAL.

6. Load the first tool into the Spindle by pressing TOOL IN/OUT and insert into the Spindle. Notice which of the two keyslots in the Tool Holder is deeper, or has a protruding setscrew. Align the tool so that the deeper keyslot faces forward and does not have any protruding setscrew to interfere with the alignment key on the arm of the ATC.

7. Type M6T2, and the DATC will place the first tool in bucket 2, and wait for the second tool.

8. Similarly, load the second tool into the Spindle.

9. Type M6T2, and the second tool will move to bucket 3.

10. Similarly, load the third tool into the Spindle.

11. Repeat as necessary until all of the tools have been loaded.

Utility

UT, Tool Number This command has six basic functions, tool setting cycle, fixture offset setting, TS-20 test, and MP 8 test. The tool setting cycle may be used to input diameter

and length offsets for multiple tools. The fixture offset setting may be used to set fixture offset locations into the fixture table. The TS-20 test is used to test the operation of the TS-Series touch probes. The MP 8 test is used to test the operation of the MP-Series probes. See the VMC Training manual for the specific operation of this command.

The tool number parameter is utilized to retrieve a specific tool. UT,5 would perform a tool change and place tool number five in the spindle.

When the UT command is entered without the tool number parameter, the utility menu is displayed.

Offset Utility Options:


```
X           O    O
Y           O    O
Z           O    O C




UTILITY OPTIONS:
1   TOOL SETTING CYCLE
2   FIXTURE OFFSET SETTING
3   TEST TS-20 PROBE
4   TEST MP PROBE
5   PALLET CHANGER
6   CLOCKS
7   EXIT
ENTER OPTION NUMBER {
```

Figure 8-52 Offset Utility Options

Offset Utility Option 1

Tool Setting Cycle

This option is used to set tool length offsets. See the Touch Probe section of this manual for the operation of this option.

Offset Utility Option 2

Fixture Offset Setting

This option is used to set fixture offset locations.

FIXTURE OFFSET Menu Items:

Item 1

Select Number/Locator

This option displays the currently selected fixture data. The operator is then prompted to select another fixture number. Enter the new fixture number or press ENTER to use the same number. The fixture offset options are displayed.

```
X              0    0
Y              0    0
Z              0    0  C


FIXTURE OFFSET = 1
 X= 0                         B= 0
 Y= 0                         A= 0
 Z= 0

FIXTURE OFFSET OPTIONS:
1   SELECT NUMBER/LOCATOR     6  FIND CORNER
2   JOG TO LOCATE             7  FIND 90 DEGREE CORNER
3   STORE LOCATION            8  MOVE TO FIXTURE OFFSET
4   FIND CENTER OF CIRCLE     9  EXIT
5   FIND MIDPOINT
PRESS OPTION NUMBER {
```

Figure 8-53 Select Number/Locator

After selecting this item, the user is prompted for the fixture number.

ENTER FIXTURE OFFSET NUMBER (1-48)

Enter the number of the fixture to be set. The user is then prompted for the locator diameter.

ENTER LOCATOR DIAMETER

When using an edge finder, enter the edge finding diameter. When using a dial indicator, press ENTER to continue. The user is prompted for a spindle speed, if a locator diameter is entered. Enter the desired RPM for the edge finder. The RPM is set; however, the spindle is not started. The spindle MUST be started manually when the operator is ready to find an edge. The fixture data and offset options are displayed after the RPM is entered.

Note: The RPM is input without the letter S. The control returns to the Command mode when the letter S is entered.

Item 2   Jog to Locate

This option prompts the user to enter the Jog mode. The operator may enter the Jog mode and find the part edge. The operator may return to the offset option display at any time, by pressing the MANUAL button.

```
X      1.5400     0
Y     -1.9225     0
Z          0      0  C


FIXTURE OFFSET = 5
 X=  -0.1000                    B= 0
 Y= 0                           A= 0
 Z= 0




PRESS JOG AND MOVE TO THE FIXTURE OR PRESS MANUAL TO EXIT
{
```

Figure 8-54 Jog to Locate

Item 3 **Store Location**

This option is used to record the fixture offset location to the fixture offset table.

```
X      1.5400     0
Y     -1.9225     0
Z           0     0  C


FIXTURE OFFSET = 5
 X=  -0.1000                    B= 0
 Y= 0                           A= 0
 Z= 0



LOCATOR COMPENSATION =  0.1000


STORE LOCATION
PRESS X, Y, Z, A OR B TO SELECT AXIS, ANY OTHER KEY TO EXIT
```

Figure 8-55 Store Location

Locator Diameter

When using a locator diameter, this option is used to adjust for that diameter. When this option is selected, the compensation amount for the locator is displayed with the prompt for the axis to set.

Press the letter of the axis to be entered into the fixture table.

The locator compensation options are then displayed.

```
X      1.5400     0
Y     -1.9225     0
Z           0     0  C


FIXTURE OFFSET = 5
 X=  -0.1000                    B= 0
 Y= 0                           A= 0
 Z= 0



PRESS ONE OF THE FOLLOWING KEYS:
 + KEY, LOCATOR ON POSITIVE SIDE
 - KEY, LOCATOR ON NEGATIVE SIDE
 0 KEY, STORE CURRENT POSITION {
```

Figure 8-56 Locator Diameter

Press the plus key if the locator touched the part on the axes positive side. The control subtracts the locator compensation amount to the current location, and stores that value in the fixture table. Press the minus key if the locator touched the axes negative side of the part. The control adds the locator compensation amount to the current location, and stores that value in the fixture table. Press 0 if the current location is desired. The current axis location is stored in the fixture table. The operator is returned to the axis selection display to select the next axis to set. Press any other key to return to the offset options. Repeat this process for each axis to set.

Dial Indicator

The procedure for the dial indicator is the same; however, the compensation options are not displayed. When the axis is selected the current location is entered into the fixture offset table.

Item 4   **Find Center of Circle**

```
LOCATE CENTER

FIRST POINT   X={
              Y=
SECOND POINT  X=
              Y=
THIRD POINT   X=                 C-COMPUTE
              Y=                 P-PICKUP XY
Z OFFSET AMOUNT=                 U-UP
                                 X-EXIT
                                 Z-PICKUP Z
                                 JOG
```

Figure 8-57 Find Center of Circle

Using a edge finder in the jog mode, the center of a circle can be determined. Enter the Jog mode by pressing the JOG button, touch the edge finder to the circumference, then press the MANUAL button to enter the coordinates for the first point. Repeat this procedure touching at 2 other points on the

circumference of the circle. The center will be calculated using these coordinates by pressing the C button.


```
LOCATE CENTER

FIRST POINT  X= 4.86
             Y=-4.4225
SECOND POINT X= 8.57
             Y= 2.2375
THIRD POINT  X=                     C-CLEAR
             Y=                     D-DRAW
Z OFFSET AMOUNT=                    I-INSERT FO
                                    S-SAME FUNC
                                    X-EXIT{


THE CENTER OF THE CIRCLE IS AT X  4.5470 Y  0.1152
```

Figure 8-58 Edge Finder

The data can then be inserted into the fixture offset table by pressing the I button.


```
X      8.5700     0
Y      2.2375     0
Z          0      0 C


FIXTURE OFFSET = 1
 X=   4.5470                   B= 0
 Y=   0.1152                   A= 0
 Z= 0

FIXTURE OFFSET OPTIONS:
1   SELECT NUMBER/LOCATOR       6   FIND CORNER
2   JOG TO LOCATE               7   FIND 90 DEGREE CORNER
3   STORE LOCATION              8   MOVE TO FIXTURE OFFSET
4   FIND CENTER OF CIRCLE       9   EXIT
5   FIND MIDPOINT
PRESS OPTION NUMBER {
```

Figure 8-59 Fixture Offset

To exit this option press the X button.

Item 5   Find Midpoint

Using the edge finder as described in the preceding paragraph, the mid point between two planes or edges can be found and entered in the fixture offset table.

Item 6 **Find Corner**

Using the edge finder as described in the proceeding paragraph, the corner between two intersecting planes or edges can be found and entered in the fixture offset table.

Item 7 **Find 90° Corner**

Using the edge finder as described in the proceeding paragraph, a 90° corner between two points can be found and entered in the fixture offset table.

Item 8 **Move to Fixture Offset**

The fixture offset calculated in Options 4-7 can be activated with this option. This will move the machine to that fixture offset's coordinates.

Item 9 **Exit**

Selecting this option exits to the Utilities menu.

Offset Utility Option 3 **Test TS-20 Probe**

This option is used to test the TS- touch probe. See the Touch Probe section of this manual.

Offset Utility Option 4 **Test MP Probe**

This option is used to test the MP probe. See the Touch Probe section of this manual.

Offset Utility Option 5 **Pallet Changer**

This option displays the pallet changer utility menu. See the Pallet Changer section of this manual.

Offset Utility Option 6   **Clocks**

This option is used to display the clock service utility.

```
X      8.5700     O
Y      2.2375     O
Z          O      O  C




CLOCK SERVICE UTILITY:
1   DISPLAY CLOCKS
2   SET TIME
3   RESET CLOCKS
4   EXIT
ENTER OPTION NUMBER
```

Figure 8-60 Clock Service Utility

Item 1   **Display Clocks**

Select this option to display all current clock settings.

```
X      8.5700     O
Y      2.2375     O
Z          O      O  C


TIME 06:10:38 PM

POWER ON        01:04:40
RUNNING         00:00:00
CURRENT TOOL    00:00:00
LAST PART       00:00:00
CURRENT PART    00:00:00

(MANUAL TO EXIT) {
```

Figure 8-61 Display Clocks

The current time is displayed at the top. The power on time is the amount of time since the last reset. Time is accumulative from each power on.

Running time is the total accumulative time that the machine has been in the AUTO mode. The time is suspended when the machine is in the WAITING state. This is time is accumulative from the last reset.

The last part time is running time of the last program run. When the AUTO mode is entered the clock stops. The time stops when the M2 or M30 is performed. Running time only is used.

The current part time is the current running time of the program in execution.

Item 2   Set Time

This option allows the user to set the current time. The previous time is displayed with the prompt to enter the new time. Press ENTER to retain the current time setting. Enter new times using a twelve hour clock. The AM or PM MUST be entered.

Note: This setting cannot be changed with the key lock on.

Item 3   Reset Clocks

When option three is selected, all clocks, except current time and tool time, are reset to zero.

To reset tool time go to the DTT table.

Note: This option is not functional when the key lock is on.

Item 4   Exit

Select this option to return to the Offset Utility Options menu.

Tape Verification

VT, Device Option   This command reads a paper tape that is punched by the VMC. Using a check sum routine, the control verifies the punched tape. The CNC displays the message TAPE IS GOOD indicating a successful punch.

The "Device Option" parameter of 1 indicates use of the RS-232 port to read the tape. This parameter is a 0 if the control is to use the machine tape reader. The procedure is as follows:

1) Type the command: VT,1 then press the ENTER key.

2) Start the tape reader.

3) Stop the tape reader when done.