LEGACY CNC 88

USER MANUAL

FADAL MACHINING CENTERS, LLC

Corporate Office..phone (818) 407-1400.............................. fax (818) 407-0020
Service / Parts...phone (818) 727-2100.............................. fax (818) 407-1004
Programming Support...............................phone (818) 727-2100.............................. fax (818) 407-0061

support@fadal.com
20701 Plummer Street, Chatsworth, California 91311 USA

TABLE OF CONTENTS

1.0 SUMMARY

1.1 NC WORD SUMMARY

Table 1-1: NC Word Summary

NC WORD SUMMARY	DEFINITION
A	A axis angular motion command (or optional Servo Coolant)
B	B axis angular motion command
C	C axis angular motion command
D	Tool diameter offset
E	Fixture offset
F	Feed rate, or spindle speed for tapping
G	Preparatory function
H	Tool length offset or Length and diameter offset for Format 1
I	X axis distance to arc center or Initial peck size for drilling (G73 G83) or X axis shift in boring cycle (G76) JY axis distance to arc center or Reducing value of the initial peck (G73, G83) or Y axis shift in boring cycle (G76)
J	Y axis distance to arc center or Reducing value of the initial peck (G73, G83) or Y axis shift in boring cycle (G76)
K	Z axis distance to arc center or Minimum peck size for drilling (G73, G83)
L	Subroutine definition or call or Subprogram repeat function (M98) or Programmable data input function (G10) or Line repeat function or Fixed cycle repetitions
M	Machine function code
N	Program sequence number
O	Program identification number
P	Dwell time in milliseconds (G04) or Percentage factor for retracting feed on tapping cycles or Fixture and tool offset number (G10) or Subprogram number (M98) or Value for R0-R9 (G10) or Sequence/ line number jump (M99) or Feed distance before peck (G73 G83) or P1 with G17 Q to use B axis during mapping or Angular tolerance for Feed Forward
Q	Peck size in drill cycles (G73, G83) or Thread lead in tapping cycles (G74, G75, G84) or Diameter for automatic tool diameter override (H99) or Scale factor for Flat Cam programming on the rotary table or Length tolerance to ignore Feed Forward
R	Subroutine parameter input R0 through R9 R0 Plane for fixed cycle or Radius designation (circular interpolation, G2 & G3) or Tool offset value amount (G10) Parametric Variables R0, R1 - R9
S	Spindle speed (RPM)
S.1	Set belt range to low
S.2	Set belt range to high
T	Tool number selector for turret
V	Variables in Macros (V1-V100)
X	X axis motion command
Y	Y axis motion command
Z	Z axis motion command

Table 1-2: Character Summary

CHARACTER	DEFINITION
0-9	Numerical digits
A-Z	Alphabetical characters
%	Program start or end, rewind to start
+	Plus, positive
-	Minus, negative
(	Comment start (standard NC program), or Engraving text start (L9201 Fixed Subroutine), or Mathematical operator (Macro Programming)
.	Decimal point
,	Comma
EOB	ENTER key, carriage return / line feed (ASCII 13,10)
*	Comment start
/	Optional block skip
:	Program identification number (Format 2)
#	Macro Line Identification

1.2 G CODES

1.2.1 PREPARATORY FUNCTIONS

Codes are divided into groups or families to distinguish which codes can function simultaneously in a program. Codes belonging to a similar group cannot function together. Codes from different families or groups can function together.

EXAMPLE: *N11 G90 G0 G1 X1. F40.*

The G0 and G1, from group A, cannot be programmed in the same line because they are both from the same group. The G90, from group F, can be with the G0 or the G1, if they were on separate lines, because it is from a different group.

EXCEPTION: A G90 and G91 can appear on the same line. Each will affect the motion words to the right of the G90 or G91 codes.

EXAMPLE: *N14 G90 X5.321 G91 Y.25 G90*

The X move will be made in absolute and the Y move will be made in incremental. The G90 at the end of the line places the machine back in absolute for the next line of the program.

1.2.2 MODAL & NON MODAL FUNCTIONS

MODAL: These codes remain in effect until modified or canceled by another modal code with the same group designation code letter.

NON MODAL: These codes only affect the line in which they appear and do not cancel modal codes.

Table 1-3: G Code Summary Table

CODE	GROUP DESIGNATION	MODAL	NON MODAL	DESCRIPTION
G0	A	Yes	-	Rapid Travel (Point-to-Point Positioning)
G1	A	Yes	* see note	Linear Interpolation
G2	A	Yes	* see note	Circular Interpolation Clockwise
G3	A	Yes	* see note	Circular Interpolation Counterclockwise
Note: G2 and G3 cancel G0 and remain active until canceled by each other. With G2 or G3 active, a move without I, J, K, or R is considered linear (G1).				
G4	B	-	Yes	Dwell
G5	A	-	Yes	Non Modal Rapid Travel
G8	D	Yes	-	Acceleration (No Feed Ramps)
G9	D	Yes	-	Deceleration (Feed Ramps & In-Position Check)
G10	I	-	Yes	Programmable Data Input
G15	C	Yes	-	YZ Circular plane with simultaneous A axis
G17	C	Yes	-	XY plane selection
G17.1	C*	Yes	-	AB word swap
G17.2	C	Yes	-	AB word swap cancel
G18	C	Yes	-	XZ plane selection
G19	C	Yes	-	YZ plane selection
G20	M	-	Yes	Check parameters for inches mode set in SETP
G21	M	-	Yes	Check parameters for metric mode set in SETP
G28	I	-	Yes	Return to current zero (set home) position
G28.1	I	-	Yes	Return from Jog Away
G29	I	-	Yes	Return from current zero (set home) position
G31	I	-	Yes	Probe touch function (Skip Function)
G31.1	I	-	Yes	Probe no touch function
G40	D	Yes	-	Cutter compensation canceled
G41	D	Yes	-	Cutter compensation left (climb)
G42	D	Yes	-	Cutter compensation right (conventional)
G43	J	Yes	-	Tool length compensation positive
G44	J	Yes	-	Tool length compensation negative
G45	I	-	Yes	Tool offset single expansion
G46	I	-	Yes	Tool offset single reduction
G47	I	-	Yes	Tool offset double expansion
G48	I	-	Yes	Tool offset double reduction
G49	J	Yes	-	Tool length offset cancel
G50	J	Yes	-	Ramp slope control cancel

Table 1-3: G Code Summary Table (Continued)

CODE	GROUP DESIGNATION	MODAL	NON MODAL	DESCRIPTION
G50.1	J	Yes	-	Mirror image cancel
G51	J	Yes	-	Ramp slope control
G51.1	J*	Yes	-	Mirror image
G51.2	J*	Yes	-	Tool Load Compensation
G51.3	J*	Yes	-	Axis Scaling
G52	I	Yes	-	Coordinate system shift
G53	I	-	Yes	Machine coordinate system
G54	O	Yes	-	Fixture offset 1 (E1)
G55	O	Yes	-	Fixture offset 2 (E2)
G56	O	Yes	-	Fixture offset 3 (E3)
G57	O	Yes	-	Fixture offset 4 (E4)
G58	O	Yes	-	Fixture offset 5 (E5)
G59	O	Yes	-	Fixture offset 6 (E6)
G66	C	Yes	-	Modal subroutine
G67	C	Yes	-	Modal subroutine cancel
G68	C	Yes	-	Rotation
G69	C	Yes	-	Rotation cancel
G70	M	Yes	-	Check parameters for inches mode set in SETP
G71	M	Yes	-	Check parameters for metric mode set in SETP
G73	E	Yes	-	Peck drill cycle
G74	E	Yes	-	Left hand tapping with compression holder
G74.1	E	Yes	-	Left hand Rigid tapping
G74.2	E	Yes	-	Prepare for Left hand Rigid tapping (optional)
G75	E	Yes	-	Tapping cycle with self-reversing head
G76	E	Yes	-	Fine bore cycle
G80	E	Yes	-	Fixed cycle cancel
G81	E	Yes	-	Spot drill cycle
G82	E	Yes	-	Counter bore cycle
G83	E	Yes	-	Deep hole drill cycle
G84	E	Yes	-	Right hand tapping with compression holder
G84.1	E	Yes	-	Right hand Rigid tapping
G84.2	E	Yes	-	Prepare for Right hand Rigid tapping (optional)
G85	E	Yes	-	Bore in, Bore out
G86	E	Yes	-	Bore in, Spindle off, Rapid out
G87	E	Yes	-	Bore in, Bore out
G88	E	Yes	-	Bore in, Dwell, Bore out
G89	E	Yes	-	Bore in, Dwell, Bore out

Table 1-3: G Code Summary Table (Continued)

CODE	GROUP DESIGNATION	MODAL	NON MODAL	DESCRIPTION
G90	F	Yes	-	Absolute programming
G91	F	Yes	-	Incremental programming
G91.1	P	Yes	-	High speed execution
G91.2	P	Yes	-	High speed execution cancel (Format 2 only)
G92	I	Yes	-	Programmed coordinate system preset
G93	K	Yes	-	Rotary axis 1/T feed rate specification
G94	K	Yes	-	Rotary axis DPM, IPM feed rate specification
G98	G	Yes	-	Return to initial plane after final Z
G99	G	Yes	-	Return to R0 plane after final Z

* Modal Code but not cancelled by similar group designation.

1.3 DEFAULT STATUS

The codes below are the default codes utilized by the control. They are in effect at power on, the beginning of program execution, when entering MDI (Manual Data Input), and after M2.

1.3.1 RESET

Format 1 will default to this status automatically. Format 2 will use this default status after the HO (Home all Axes) command is used. Use HO like a reset button when in the Format 2 mode. By typing the command HO then pressing the enter button, the control will go into the WAITING stage. At this point the control is reset. If it is desired to move to home, press the START button, if not, press the MANUAL button. The SU (Sum Program) command will reset and use the default status from the SETP parameters in both format 1 and 2.

Table 1-4: Default G Codes

G/M CODE	AT BEGINNING OF PROGRAM, UPON ENTERING MDI, AFTER M2	BY RESET ONLY
G0 - P	1	2
G1 - P	1	2
G8 Format 2	2 (Unless G9 is used in Auto - Then by reset)	
G9 Format 1	1	
G17 - P	1	2
G18 - P	1	2
G19 - P	1	2
G40	1 & 2	
G49	1	2
G50	1	2
G80	1	2
G67	1 & 2	
G69	1	2
G98	1	
M5	1 & 2	
M9	1 & 2	
M10	1 & 2	
M47	1	2
M48	1	2
M96 - P	1 & 2	
M97 - P	1 & 2	

NOTE
The 1 indicates the code is in effect in Format 1. The 2 indicates the code is in effect in Format 2. The P indicates that these codes may be established by the parameters established with the SETP (Set Parameter) command.

1.4 M FUNCTIONS

1.4.1 MODAL

These codes remain in effect until canceled by another modal code.

1.4.2 NON MODAL

These codes only affect the line in which they appear and do not cancel modal codes.

NOTE
Some M Functions start with motion commanded in a line. Some M Functions start after motion has been completed.

NOTE
For M60 through M64 only, the use of a minus sign before the number (M-60) will cause the function to occur after motion. This allows the rotary motion and brake application prior to any fixed cycle execution.

Table 1-5: M Function Summary Table

CODE	STARTS WITH MOTION	STARTS AFTER MOTION	MODAL	NON MODAL	DESCRIPTION
M0	-	Yes	-	Yes	Program stop
M1	-	Yes	-	Yes	Optional program stop
M2	-	Yes	-	Yes	End of program
M3	Yes	-	Yes	-	Spindle on clockwise
M3.1	Yes	-	Yes	-	Sub-Spindle on clockwise
M3.2	Yes	-	Yes	-	Return to Main Spindle
M4	Yes	-	Yes	-	Spindle on counterclockwise
M4.1	Yes	-	Yes	-	Sub-Spindle on counterclockwise
M4.2	Yes	-	Yes	-	Return to Main Spindle
M5	-	Yes	Yes	-	Spindle (and Sub-Spindle) stop
M6	-	Yes	-	Yes	Tool change
M7	Yes	-	Yes	-	Coolant 1 on
M7.1	Yes	-	Yes	-	Servo Coolant 1 on
M8	Yes	-	Yes	-	Coolant 2 on
M8.1	Yes	-	Yes	-	Servo Coolant 2 on

Table 1-5: M Function Summary Table (Continued)

CODE	STARTS WITH MOTION	STARTS AFTER MOTION	MODAL	NON MODAL	DESCRIPTION
M9	-	Yes	Yes	-	Coolant / Servo Coolant 1 & 2 off
M10	Yes	-	Yes	-	Reciprocation cancel
M11	Yes	-	Yes	-	Reciprocate X axis
M12	Yes	-	Yes	-	Reciprocate Y axis
M13	Yes	-	Yes	-	Reciprocate Z axis
M14	Yes	-	Yes	-	Reciprocate A axis
M15	Yes	-	Yes	-	Reciprocate B axis
M16	Yes	-	Yes	-	Reciprocate C axis (VMC45 only)
M17	-	-	-	Yes	End of subroutine (see M30)
M18	Yes	-	-	Yes	Cushman® or Erickson® indexer next step
M19	Yes	-	-	Yes	Spindle orient & lock
M20	Yes	-	-	Yes	General purpose indexer next step or Auto. Doors Close
M30	-	-	-	Yes	End of all subroutines (see M17) or End of program (Format 2)
M31	-	-	-	Yes	Exchange Pallets
M32	-	-	-	Yes	Store and Load Pallet A
M32.1	-	-	-	Yes	Store and Load Pallet A - Test
M33	-	-	-	Yes	Store and Load Pallet B
M33.1	-	-	-	Yes	Store and Load Pallet B - Test
M41	-	-	Yes	-	Low RPM range
M42	-	-	Yes	-	High RPM range Auto Hi/Low
M43	-	-	Yes	-	High RPM range Manual change
M45	-	-	-	Yes	Execute fixed cycle
M46	-	Yes	Yes	-	Positive approach activate
M47	-	Yes	Yes	-	Positive approach cancel
M48	Yes	-	Yes	-	Potentiometer control on
M48.1	Yes	-	Yes	-	Servo coolant override Pot on
M48.2	Yes	-	Yes	-	Pallet A Rotary override Pot on
M48.3	Yes	-	Yes	-	Pallet B Rotary override Pot on
M49	Yes	-	Yes	-	Potentiometer control off
M49.1	Yes	-	Yes	-	Servo coolant override Pot off
M49.2	Yes	-	Yes	-	Pallet A rotary override Pot off
M49.3	Yes	-	Yes	-	Pallet B rotary override Pot off
M60	-	Yes	-	Yes	A Axis Brake On
M61	-	Yes	Yes	-	A Axis Brake Off
M62	-	Yes	-	Yes	B Axis Brake On
M63	-	Yes	Yes	-	B Axis Brake Off

Table 1-5: M Function Summary Table (Continued)

CODE	STARTS WITH MOTION	STARTS AFTER MOTION	MODAL	NON MODAL	DESCRIPTION
M64	-	-	Yes	-	Activate MP8 Probe with M66 - Activate MP12 Probe with M67 - Activate Laser Probe
M65	-	-	Yes	-	Activate TS-20, TS-27 Probe
M66	-	-	Yes	-	User Attached Device
M67	-	-	Yes	-	User Attached Device
M68	-	-	Yes	-	User Attached Device
M69	-	-	Yes	-	User Attached Device
M80	-	-	-	Yes	Automatic Doors Open
M81	-	-	-	Yes	Automatic Doors Close (Optional)
M90	Yes	-	Yes	-	Default Gain Setting
M91	Yes	-	Yes	-	Normal Gain Setting
M92	Yes	-	Yes	-	Intermediate Gain Setting
M93	Yes	-	Yes	-	High Gain Setting
M94	Yes	-	Yes	-	Feed Forward Function
M94.1	Yes	-	Yes	-	Feed Rate Modification
M94.2	Yes	-	Yes	-	Advanced Feed Forward (Optional)
M95	Yes	-	-	Yes	Feed Forward Cancel
M95.1	Yes	-	-	Yes	Feed Rate Modification Cancel
M95.2	Yes	-	-	Yes	Advanced Feed Forward Cancel
M96	Yes	-	Yes	-	Intersectional CRC Cancel
M97	Yes	-	Yes	-	Intersectional CRC
M98	-	-	-	Yes	Execute subprogram
M99	-	-	-	Yes	End of subprogram or Line jump

1.5 PROGRAM TAPE INPUT

The following is an example of the input format the control reads from a paper tape or computer file:

%
N0.001 O100 (DRILL PROGRAM
N1 M6 T1
N2 (TOOL #1 CENTER DRILL
N3 G0 G90 S10000 M3 E1 X1. Y2.
N4 H1 M7 Z.1
N5 G73 G99 R0+.1 Z-.75 F25. Q.1 X1. Y2.
N6 X2.

N7 Y1.
N8 M5 M9 G80
N9 G90 G0 H0 Z0
N10 E0 X0 Y0
N11 M2
%

The first “%” character signals the start of data. The CNC data follows the first percent character. The second “%” character signals the end of the program.

Acceptable character code sets are:

1. EIA RS-358-B
2. EIA RS-244-B
3. ASCII

To send data to the VMC the procedure is as follows:

1. Use the Change Device (CD, __) command to establish the proper baud rate (see Baud Rate).
2. Enter the TA,1 (Tape Reader Input) command at the VMC.
3. Start reading the paper tape or send data from the computer.
4. Enter the BYE command to reset the COMM port.

To receive data from the VMC the procedure is as follows:

1. Prepare the device to receive the data.
2. Enter the Change Device (CD, ___) command at the VMC.
3. Enter the PU (Punch Program Tape) command at the VMC.

1.6 PROGRAM NUMBERS, PROTECTION & STORAGE

1.6.1 PROGRAM NUMBER

The program number is identified by the letter O and a numeric value from 1 to 9999. O1 - O9999 placed on the first line of program designates the program number.

It is not necessary to put an O word in the beginning of the current program in memory. However, a program must have an O word to be stored in the program library (see PR).

1.6.2 O WORD

An axis move or other words are not allowed to be coded on the line with the O word. The O word line may contain a comment.

EXAMPLE:

N1 O1 (PROGRAM 1(This is acceptable).
N1 O1 X3. *(*This is not acceptable).

EXAMPLE:

Format 1 or Format 2

N1 O1 (PROGRAM 1
N2 M6 T1
N3 (TOOL #1 1/2 END MILL
N4 G0 G90 S10000 M3 E1 X1. Y2.
N5 H1 M7 Z.1
N6 G1 Z-.1 F10.
N7 X1.F60.
N8 M5 M9
N9 G90 G0 H0 Z0
N10 M2

EXAMPLE:

Format 2 ONLY

In programming Format 2 a colon (:) can be used in place of an O word.

N1 :1 (PROGRAM 1
N2 M6 T1
N3 (TOOL #1 1/2 END MILL
N4 G0 G90 S10000 M3 E1 X1. Y2.
N5 H1 M7 Z.1
N6 G1 Z-.1 F10.
N7 X1.F60.
N8 M5 M9
N9 G90 G0 H0 Z0

N10 M30

1.7 PROGRAM PROTECTION

1.7.1 NOEDIT

EXAMPLE: *N1 O1 (NOEDIT or N1 O1 (P/N 1234 LEFT SIDE NOEDIT*

To delete a NOEDIT program from memory the NOEDIT program must not be the current program in memory. By choosing the option DELETE PROGRAM from the Program library menu (PR), the user can now delete the NOEDIT program. Once again, this is only true if the NOEDIT program is not the current program in memory.

> NOTE
> Keep a copy of the original program without NOEDIT.

A program with NOEDIT in the comment of the O word line, is a program that may never be edited at the CNC.

A NOEDIT program will not allow commands CH, DE, IN, NU, NE, CO, LE, PU or from PA: C, I, O, N, and R (see the PA command).

The only functions allowed to be used with the Page Editor and the NOEDIT programs are graphics, viewing the program, changing to another program, starting a new program, and running auto.

1.7.2 KEY LOCK

The KEY LOCK in the horizontal position locks out the availability to edit the program on the CNC. On a 32MP pendant the DOS side will also be locked out.

1.7.3 EMERGENCY STOP BUTTON

The EMERGENCY STOP BUTTON in the depressed position locks out the availability to edit. Release the button by turning it clockwise and then press the JOG button to reset the control.

1.7.4 PROGRAM STORAGE

Programs stored in memory can be managed by using the PR (Program Number) command (see section 8.0 COMMANDS, MAN-0131). The PR menu allows the operator to switch, display, start, copy and remove programs. Enter PR command to see the following menu:

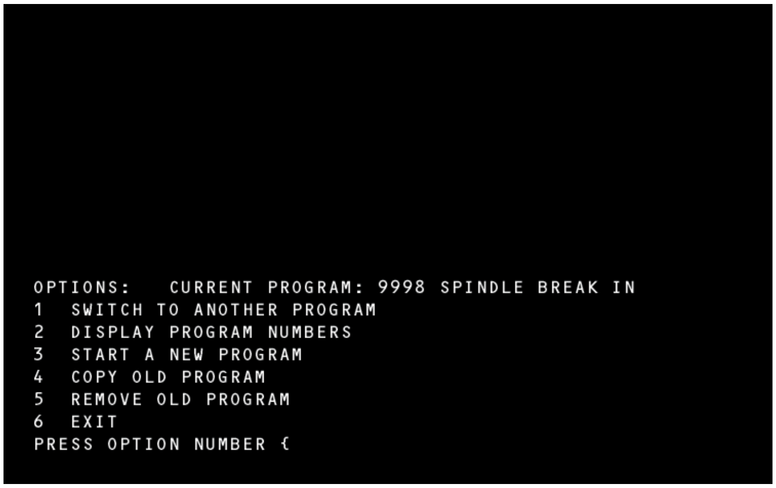


Figure 1-1: Program Storage Menu

EXAMPLE: Option #1

This option switches the current program to another program stored in memory.

EXAMPLE: Option #2

This option displays the programs in memory. The programs are listed in numerical order. If the address contains a comment, 16 characters of the comment are displayed as a program label.

EXAMPLE: Option #3

This option starts a new program. Active memory is cleared and a new block (N.001) is created containing the new program number. Program input is from the machine's keyboard.

EXAMPLE: Option #4

This option copies or duplicates a program stored in memory. The new program is assigned an unused number.

EXAMPLE: Option #5

This option deletes any program stored in memory. The program is removed from the machine's memory without any chance of recovery.

EXAMPLE: Option #6

This option returns to the command mode.

1.7.5 PROGRAM DATA INPUT

There are two procedures in which to save the current program in memory and input another program.

Input From The Keyboard:

1. Enter the PR command.
2. Select option #3 and enter the program number. The new program becomes active with the first block already containing the new O word.
3. Select option #6 to exit the menu to the command mode.
4. Enter the IN,1 (Insert Program) command to begin keyboard input after the line containing the program number. Alternatively, use the PA (Program Page Edit) command and use the insert I command to begin input from the keyboard.

Input From The RS-232 Port:

1. The first block of the active program should contain a program number.
2. Begin transmission to the CNC. Upon completion of receiving the program, the result is according to the following circumstances:
   a. No O word in the current program: the program sent to the machine becomes active; the old program is deleted.
   b. The program contains an O word: the old program is placed into memory; the program sent to the machine becomes active.
   c. The program contains a duplicate O word: the new program becomes active; the old program is deleted.

1.8 FORMAT CLASSIFICATION SHEET

Reference: Conforming to ANSI/EIA RS-274-D standard.

1.8.1 MACHINE

Vertical Machining Center (VMC).

1.8.2 FORMAT CLASSIFICATION SHORTHAND

D617.524.665

D variable block format contouring/positioning system

6 motion dimension words (X, Y, Z, A, B, C)

17 other words (E, D, O, N, M, F, G, S, R, H, L, P, Q, T, I, J, K)

5 absolute or incremental data, depending on mode of operation

2 digits to left of decimal point in longest axis (3 metric)

4 digits to the right of the decimal point in longest axis (3 metric)

6 motion control channels (X, Y, Z, A, B, C)

6 numerically controlled machine axes (X, Y, Z, A, B, C)

5 decimal point programming: if no decimal point, defaults assumed

1.8.3 FORMAT DETAIL

Inches Mode Increment System

N5.4
G2.1
X+3.4 Y+3.4 Z+3.4 I+3.4 J+3.4 K+3.4 B+3.4 R+3.4 Q+3.4
A+4.3
C+5.1
M2.1 H2 T2 D2
F4.2
S5.1, O4
L4 P4
MILLIMETERS MODE
N5.4
G2.1
X+3.3 Y+3.3 Z+3.3 I+3.3 J+3.3 K+3.3 B+3.3 R+3.3 Q+3.3
A+4.3
C+5.1
M2.1 H2 T2 D2
F4.2 S5.1 L4 P4 O4

1.8.4 G FUNCTION CODES

0, 1, 2, 3, 4, 5, 8, 9, 10, 15, 16, 17, 17.1, 17.2, 18, 19, 20, 21, 28, 28.1, 29, 31, 31.1, 40, 41, 42, 43, 44, 45-48, 49, 50, 50.1, 51, 51.1, 51.2, 51.3, 52, 52.1, 53, 54-59, 66-71, 73-76, 80-89, 90, 91.1, 91.2, 92-94, 98, 99

1.8.5 M FUNCTION CODES

0, 1, 2, 3, 3.1, 3.2, 4, 4.1, 4.2, 5, 6, 7, 7.1, 8, 8.1, 9-16, 17-20, 30, 31, 32, 32.1, 33, 33.1, 41-43, 45-47, 48, 48.1, 48.2, 48.3, 49, 49.1, 49.2, 49.3, 60-69, 80, 81, 90-93, 94, 94.1, 95, 95.1, 96, 97, 98, 99

- 2 digit BCD output (standard)
- 2 decades of relay output (optional)
- The use of a minus sign (M-60) will perform the function to be accomplished after motion. This usage applies to M60 through M69 only.

1.8.6 F FUNCTION RANGE

The F word is used to define the feed rate. It is modal and remains in effect for G1, G2, and G3 moves until another F word is used in the program or in the MDI mode. See G93 and G94 in the index for more information.

- 1 to 150 percent feed rate override
- .01 to 375 inches per minute
- 1 to 3810 millimeters per minute
- .6 to 9000 degrees per minute (72 to 1)
- .6 to 7992 degrees per minute (90 to 1)
- .6 to 3960 degrees per minute (180 to 1)
- .6 to 1980 degrees per minute (360 to 1)

1.8.7 S FUNCTION

The S word represents the PRM to be used when the spindle is turned on with the M3, M4, or SPINDLE ON/OFF with the shift button combination. The lower belt range RPM amounts can be used from the upper belt range by using a .2 at the end of the integer. For example, S1000.2 would result in 1000 would result in a belt range to the lower range.

> WARNING
> The S word is modal and will remain in effect until another S word is used in auto or the MDI mode.

VMC 7.5 HP (Manual Belt)

75 to 3750 Top belt range

75 to 7500 Bottom range

VMC 15 HP

40 to 2500 Top belt range

150 to 10000 Bottom range

EXAMPLE: VMC 15 HP (Auto High/Low)

75 to 2500 Top belt range, S.1 used to override belt to Top belt range

2501 to 10000 Bottom range, S.2 used to override belt to Bottom belt range

VMC High Torque (Auto Hi/low)

40 to 2500 Top belt range, S.1 used to override belt to Top belt range

2501 to 10000 Bottom range, S.2 used to override belt to Bottom belt range

VMC High Speed Head (Single Range)

300 to 15000 Single range

1.8.8 T FUNCTION CODE

The T word specifies turret location selection. The number will range from 1 through 30 depending on the available turret locations in the tool changer. The T word is usually used in conjunction with the M6 tool change M function. It would appear as an M6T# on a line by itself (See M6 for details). However the T word is modal and can be used on any line prior to the M6 code.

NOTE

The use of a minus sign with the T word (T-5) will rotate the turret until the pocket is located directly opposite from the spindle. This might be used to rotate long tools in the turret to some location to avoid hitting a part during program execution. At the next tool change the turret will rotate automatically back to its original position.

NOTE

Do not use the T-# with an M6.

1.8.9 D FUNCTION CODE

The D word specifies which diameter or radius offset to use from the tool table for cutter radius compensation. It ranges from 1 through 99. This code is not necessary in Format 1, but may be used for cutter diameter override.

1.8.10 H FUNCTION CODE

Programming Format 1:

The H word will pick up the diameter, and tool length offset from the tool table. It ranges from 1 through 99. It is also used for Tool timers selection.

H99 Q Value

H99 is used for automatic tool diameter override with CRC (see CRC).

H0 cancels the current length offset (see G49).

Programming Format 2:

In Format 2 the H word will only pick up the tool length offset. It is also used for Tool timers selection.

H0 cancels the current length offset (see G49).

1.8.11 MAXIMUM WORKING DIMENSIONS

VMC 5, 10, 15

X=20 inches, Y=16 inches, Z=20 inches

Table size= 16" x 29.5"

Maximum clearance under spindle is 24"

Minimum clearance under spindle is 4"

EXAMPLE:

VMC 15XT

X=30 inches, Y=16 inches, Z=20 inches

Table size= 16" x 29.5"

Maximum clearance under spindle is 24"

Minimum clearance under spindle is 4"

VMC 2016L

X=20 inches, Y=16 inches, Z=20 inches (optional 28")

Table size= 16" x 29.5"

Maximum clearance under spindle is 24"

Minimum clearance under spindle is 4"

EXAMPLE:

VMC 3016L

X=30 inches, Y=16 inches, Z=20 inches (optional 28")

Table size= 16" x 38"

Maximum clearance under spindle is 24"

Minimum clearance under spindle is 4"

EXAMPLE:

VMC 3016

X=30 inches, Y=16 inches, Z=20 inches (optional 28")

Table size= 16" x 39"

Maximum clearance under spindle is 24" (optional 32")

Minimum clearance under spindle is 4"

EXAMPLE: VMC 3020

X=30 inches, Y=20 inches, Z= 24 inches (optional 32")

Table size= 40.5" x 20"

Maximum clearance under spindle is 28" (optional 36")

Minimum clearance under spindle is 4"

VMC 2216

X=22 inches, Y=16 inches, Z=20 inches

Table size= 16" x 39.5"

Maximum clearance under spindle is 24"

Minimum clearance under spindle is 4"

EXAMPLE: VMC 4020

X=40 inches, Y=20 inches, Z=20 inches (optional 28")

Table size= 20" x 47.9"

Maximum clearance under spindle is 24" (optional 32")

Minimum clearance under spindle is 4"

EXAMPLE: VMC 4020A

X=40 inches, Y=20 inches, Z=20 inches (optional 28")

Table size= 48" x 20"

Maximum clearance under spindle is 24" (optional 32")

Minimum clearance under spindle is 4"

EXAMPLE: VMC 5020A

X=50 inches, Y=20 inches, Z=20 inches (optional 28")

Table size= 20" x 47.9"

Maximum clearance under spindle is 24" (optional 32")

Minimum clearance under spindle is 4"

EXAMPLE: VMC 6030

X=60 inches, Y=30 inches, Z=30 inches

Table size= 30" x 62.5"

Maximum clearance under spindle is 35.5"

Minimum clearance under spindle is 5.5"

VMC 8030

X=80 inches, Y=30 inches, Z=30 inches

Table size= 30" x 82.5"

Maximum clearance under spindle is 35.5"

Minimum clearance under spindle is 5.5"

1.8.12 GEOMETRIC RELATIONSHIP

X, Y, Z, C per RS-267-A

A, B need not be parallel to any particular axis.

2.0 M FUNCTIONS

2.1 M0 PROGRAM STOP

M0 temporarily suspends program execution and cancels the spindle and coolant functions. The CNC enters the WAITING state until the operator pushes one of the following buttons:

- START or AUTO button (to continue program execution). The coolant and spindle will not turn on unless coded.
- MANUAL (aborts program execution).
- JOG (initiates the jog away feature).

Using the jog away feature, any axis can be moved away from its current position without disturbing the CNC's memory of that position. This allows an optional automatic return of the jogged axes to that position to complete the machining cycle (see section 7.0 SLIDE HOLD, MAN-0131).

EXAMPLE:

G80
M5 M9
G53 Z0
M6 T3 (TOOL #3, .5 2FL HSS EM .01 C'RADIUS
M0 (OPERATOR CHANGE CLAMPS FROM POSITION 1 TO POSITION 2
(The program will stop at this line, and will not continue until the START or AUTO button is pressed).

G90 G0 S7500 M3 E1 X1.43 Y-2.7

NOTE

The look ahead processing does not continue past an M0. Look ahead processing begins after the START or AUTO button is pressed while in the WAITING state.

NOTE

Another way to accomplish a program stop is to use G4 P66000 on a line in the program. This will cause the control to enter the WAITING state. In this case the spindle and coolant **will not** be turned off and the look ahead processing **will** continue beyond the G4 code (see section 3.0 G CODES, MAN-0131).

2.2 M1 OPTIONAL PROGRAM STOP

M1 is similar to M0 with the exception that the program will stop only when the optional stop switch is in the ON position. This code could be included in a program for the convenience of the operator to allow the program to stop at certain points. For machine pendants without an optional stop toggle switch, type MU (Menu) after the AUTO button is pressed or any time while the program is being executed, and select Option 2 from the Run Time Menu.

EXAMPLE:

Z.1 G0 G40

M5 M9
G53 Z0
M6 T5 (TOOL #5, .375 (3/8) 4FL CARBIDE EM, TIN
M1
(The program will stop here only if either the optional stop switch is on or Option 2 has been selected from the Run Time Menu).

G90 G0 S10000 M3 E1 X-.3 Y-.2

2.3 M2 END OF PROGRAM (FORMAT 1)

M2 indicates the end of the main program, and will cause the following events:

- Cancel the current tool length offset.
- Cancel the fixture offset and move to the current position at E0.
- Return all axes to the home position and move to the SETH position.
- Cancel the spindle and coolant function.
- Reset preparatory functions (See section 1.0 SUMMARY, MAN-0131).

The CNC then enters the WAITING state ready to run the program again.

> NOTE
> The CNC continuously processes data, filling the look ahead buffer. The WAITING state appears when the buffer is full. If the AUTO, MANUAL, or START button is pressed while processing, the CNC will immediately enter the WAITING state, allowing the operator to press START or AUTO to begin execution.

2.4 M2 END OF PROGRAM (FORMAT 2)

In Format 2, M2 acts as an M30 to mark the end of the main program and will only:

- cancel the current tool length offset.
- cancel the spindle and coolant functions.

EXAMPLE:

Formats 1 & 2

M5 M9
G53 Z0
M6 T1
E0 X0 Y0
M2 (This is the end of the main program).

2.5 M3 SPINDLE CW

M3 is used to start spindle rotation in a clockwise direction.

EXAMPLE: *M6 T7 (TOOL #7, #1 C'DRILL*
M1
G90 G0 S300 M3 E1 X0 Y0 (The spindle will turn on CW at this line).

2.6 M3.1 SUB-SPINDLE ON, IGNORE MAGNET

✓ SPECIAL FEATURE

M3.1 is used to start a sub-spindle. Normally an M3 would be used to start the main spindle. When this occurs, the control is looking for the magnet on the pulley. If the control does not see the magnet turning, a "Motor Overload" message will appear on the screen. The M3.1 will ignore the pulley magnet and direct the sub-spindle to start.

An S word on the line or before the M3.1 will establish an RPM to use when the spindle starts. An S word after the M3.1 will direct the spindle to rotate at the new RPM. An M function board needs to be installed to redirect the Spindle-On command to the drive system of the sub-spindle. Use an M5 to stop the sub-spindle from rotating. See also M3.2 to use the main spindle after the sub-spindle is used.

EXAMPLE: *M6 T7 (TOOL#7,BLANK TOOL FOR SUB-SPINDLE*
M1
G90 G0 S300 M3.1 E1 X0 Y0 (The sub-spindle will start at this line).

2.7 M3.2 ACKNOWLEDGE SPINDLE MAGNET

✓ SPECIAL FEATURE

Use an M3.2 to acknowledge the spindle magnet of the main spindle after an M3.1 was used. If this is not used after an M3.1, the M3 would start the sub-spindle again.

2.8 M4 SPINDLE CCW

This function is used to start the spindle rotation in a counterclockwise direction.

EXAMPLE:

X3.674 Y1.5
M19
Z-1.2
M4 S300 (The spindle will turn on CCW at this line).
Z-1.18 G1 F4.

2.9 M4.1 SUB-SPINDLE ON, IGNORE MAGNET

SPECIAL FEATURE

M4.1 is used to start a sub-spindle. Normally, an M4 would be used to start the main spindle. When this occurs the control is looking for the magnet on the pulley. If the control does not see the magnet turning, a “Motor Overload” message will appear on the screen. The M4.1 will ignore the pulley magnet and direct the sub-spindle to start.

An S word on the line or before the M4.1 will establish an RPM to use when the spindle starts. An S word after the M4.1 will direct the spindle to rotate at the new RPM.

An M function board needs to be installed to redirect the Spindle-On command to the drive system of the sub-spindle. Use an M5 to stop the sub-spindle from rotating. See also M4.2 to use the main spindle after the sub-spindle is used.

EXAMPLE:

M6 T7 (TOOL#7,BLANK TOOL FOR SUB-SPINDLE CLEARANCE
M1
G90 G0 S300 M4.1 E1 X0 Y0 (The sub-spindle will start at this line).

2.10 M4.2 ACKNOWLEDGE SPINDLE MAGNET

SPECIAL FEATURE

Use an M4.2 to acknowledge the spindle magnet of the main spindle after an M4.1 was used. If this is not used after an M4.1, the M4 would start the sub-spindle again.

2.11 M5 SPINDLE OFF

TIME SAVER

M5 stops the spindle and the coolant. The spindle **will neither** orient **nor** lock. Using M5 on a line prior to an M6 will reduce the time of a tool change by stopping the spindle before the head returns to the tool change position.

2.12 M6 TOOL CHANGE

M6 changes tools in the spindle. The M6 usually appears in a line with a T# code. The T# will specify which tool to pick up next. The M6 can be used from any position on the table.

M6 will cause the following events:

- The current tool length offset will be canceled, and the Z axis will move to the cold start position.
- The spindle will stop and orient, and the coolant will stop.
- The tool changer will move out to the tool in the spindle, and the head will rise above the tool.
- The turret will rotate to the position specified by the T word, and then the head will move down over the tool, installing that tool into the spindle.
- After exchanging the tools, the Z axis will move to the SETZ position if the M6 is used in a position other than the Z axis CS position.

> **NOTE**
> Execution of this code will cancel the Z portion of a fixture offset. This Z axis amount will be applied with the next H or Z words.

2.12.1 SETP PARAMETER

The SETP option SPINDLE ON AFTER M6 can affect the M6. The factory recommends the NO setting.

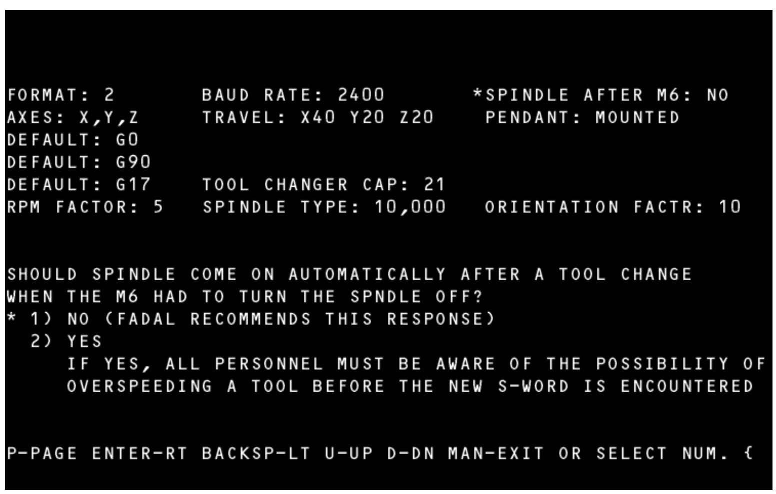


Figure 2-1: SETP Parameter Menu

WARNING
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 over-speed of the next tool. It is recommended that this parameter is set to NO.

2.12.2 T-# MOVE TOOL CHANGER

✓ SPECIAL FEATURE

A T-# in a line by itself (with no M6) will rotate the turret so that the turret # specified is opposite the spindle. See Example 2 below.

EXAMPLE:

M5 M9
G90 G0 G53 Z0
M6 T21 (TOOL #21, MP-8 PROBE (The tool is changed to tool #21 at this line).

EXAMPLE:

M5 M9
G90 G0 G53 Z0
X-10. Y5. (MOVE TO CLEARANCE POSITION
M6 T2 (TOOL #2, 10 INCH 1.25 SPADE DRILL
(Tool #2 is changed into the spindle).
T-10 (The turret will rotate so that tool #10 is opposite the spindle).

2.13 M7 COOLANT ONE ON

The M7 function activates the power supply either to the electrical receptacle labeled “FLOOD” or to the electrical receptacle labeled “MIST”. Both receptacles are located on the back of the control box, and each is 110 VAC / 6 amps and may be used for various electrical devices. The receptacle that is activated by M7 depends on the SETP option selected (see the Coolant Options table below).

2.13.1 SETP PARAMETER

The M7 code will activate the Coolant One (flood) receptacle only when the SETP parameter is set for M7 as flood. If M7 is set to mist, then M7 will activate the Coolant Two (mist) receptacle and M8 will activate the Coolant One (flood)

EXAMPLE:

With M7 set to FLOOD in the SETP pages
M6 T1 (TOOL #1, 1 INCH 3FL HOG EM
G90 G0 S5000 M3 E1 X.9 Y.6
H1 D1 Z-.98 M7 (Coolant One is turned on at this line).

Table 2-1: Coolant Options Codes M7, M7.1, M8, M8.1, M9

SETP OPTION SELECTED	FLOOD RECEPTACLE ON	MIST RECEPTACLE ON	OPTIONAL SERVO COOLANT ON
M7=FLOOD; M8=MIST Option 1	M7	M8	M7.1
M8=FLOOD; M7=MIST Option 2 (Default)	M8	M7	M8.1

M9 cancels all Coolant functions

2.14 M7.1 PROGRAMMABLE COOLANT ON

The M7.1 activates the optional Servo Coolant and the electrical receptacle labeled “FLOOD” (Coolant One) only if M7 is selected as the Flood Coolant in the SETP parameter page (see the Coolant Options table above). For more details on operation and programming of the Servo Coolant system see section 19.0 MISCELLANEOUS, MAN-0131.

2.14.1 SETP PARAMETER

The M7.1 code will activate the optional Servo Coolant only when the SETP parameter is set for M7 as flood. If M7 is set to mist, then M8.1 must be used to activate the optional Servo Coolant.

2.15 M8 COOLANT TWO ON

✓ OPTIONAL FEATURE

The M8 function activates the power supply either to the electrical receptacle labeled “MIST” or to the electrical receptacle labeled “FLOOD”. Both receptacles are located on the back of the control box, and each is 110 VAC / 6 amps and may be used for various electrical devices. The receptacle that is activated by M8 depends on the SETP option selected (see the Coolant Options table above).

2.15.1 SETP PARAMETER

The M8 code will activate the Coolant One (flood) receptacle only when the SETP parameter is set for M8 as flood. If M8 is set to mist, then M8 will activate the Coolant Two (mist) receptacle and M7 will activate the Coolant One (flood) receptacle.

EXAMPLE:

With M8 set to MIST in the SETP pages
M6 T1 (TOOL #1, 1 INCH 3FL HOG EM
G90 G0 S5000 M3 E1 X.9 Y.6
H1 D1 Z-.98 M8 (Coolant Two is turned on at this line).

2.16 M8.1 PROGRAMMABLE COOLANT ON

✓ OPTIONAL FEATURE

The M8.1 activates the optional Servo Coolant and the electrical receptacle labeled "FLOOD" (Coolant One) only if M8 is selected as the Flood Coolant in the SETP parameter page (see the Coolant Options table above). For more details on operation and programming of the Servo Coolant system see section 9.0 MISCELLANEOUS, MAN-0131).

The M8.1 code will activate the optional Servo Coolant only when the SETP parameter is set for M8 as flood. If M8 is set to mist, then M7.1 must be used to activate the optional Servo Coolant.

2.17 M9 COOLANT OFF

The M9 code will cancel both Coolant One and Coolant Two (M7, M8, M7.1 and M8.1).

EXAMPLE:

E0 X0 Y0
M5 M9 (The coolant will be canceled at this line).

2.18 M10 CANCEL RECIPROCATION

✓ SPECIAL FEATURE

The M10 code cancels reciprocation. The reciprocated move will stop and complete its motion at the end point of the original move.

2.19 M11 X AXIS RECIPROCATION

✓ SPECIAL FEATURE

The M11 code will reciprocate the last X axis move made in the program. As the X axis is moving back and forth from the beginning point to the end point of the move, all other axes can be moved while the X move is reciprocating.

The feed rate of the reciprocated move is separate from the advancing moves. The F word before the M11 will apply to the reciprocated move. The F word after the M11 will apply to the advancing moves. The feed rate for the advancing moves is usually much lower than the feed rate for the first or reciprocated move.

EXAMPLE:

(See Figure 2-2).
G0 G90 S2000 M3 X0 Y0 (This X position is the beginning point of the reciprocated move).
H1 M7 Z.1
G1 X5.0 F50. (The F50. here applies to the reciprocated move only. X5.0 is the end point).
M11 (Reciprocate the last X move until the M10 code is used).
G1 Y-2. Z-.25 F1. (The F1. applies to the advancing moves only).

Y-2.25
G19
Y-2.5 Z0 K.25 G2
M10 (The reciprocation is canceled and the X axis moves to the end point at X5.0 Y-2.5).
G4 P2000 (A dwell is used to allow the reciprocated move to get to the end point)
.

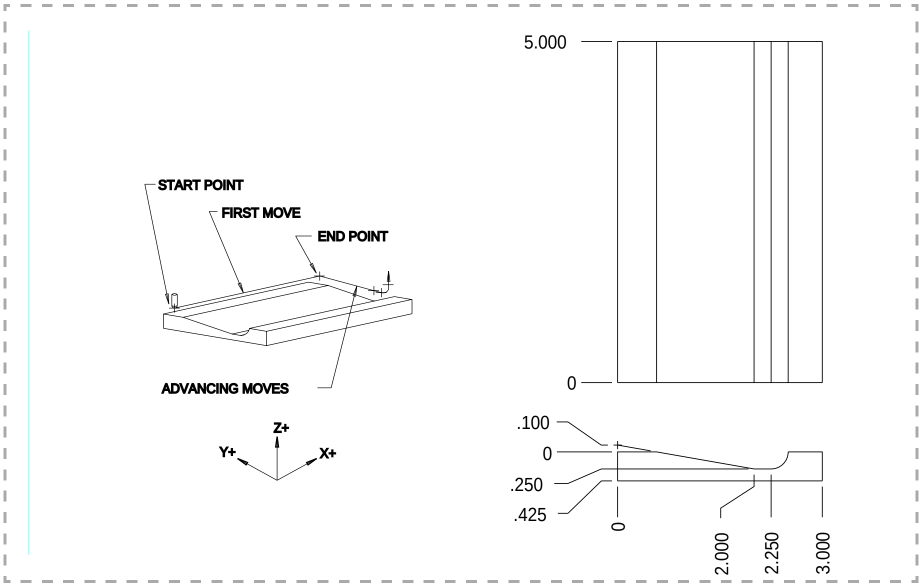


Figure 2-2: M11 X Axis Reciprocation

2.20 M12-M16 RECIPROCATION FOR Y, Z, B, A

These codes are similar to M11, however, they apply to the Y, Z, B, and A axes.

2.21 M17 END OF SUBROUTINE

✓ SPECIAL FEATURE

The M17 code is used to mark the end of a subroutine. No other coding is allowed on the same line with the M17. (See section 6.0 FIXED SUBROUTINE, MAN-0131)

EXAMPLE:

L100 (SUB FOR X+ MOVE
X.5

Optional:

M17 (This marks the end of subroutine #1).
L200 (SUB FOR X- MOVE
X-.5

Optional:

M17 (This marks the end of subroutine #2).
L300 (SUB FOR ALL HOLES
G91
L120
Y-.5
L220

Required:

M17 (This marks the end of subroutine #3).
M30
(End of subroutine section).
(MAIN PROGRAM
(Program execution begins after the M30 code).

2.22 M18 AIR RATCHETING INDEXER

This code is used with the Cushman® (PCB-0022) and Erickson® (PCB-0023 or PCB-0024) style rotary indexer interface boards from FADAL. The M18 sends a signal to index, then the CNC will wait for a cycle-complete signal from the indexer before continuing with execution of the program. If the M18 is used with a fixed cycle, the M45 code must be used to execute the cycle because the control does not recognize the M18 as a move (see M45).

EXAMPLE:

G82 G99 R0.1 Z-.25 F30. P18 M45
M18 (A90.) (Index to next step and wait for cycle-complete signal).
M45
M18 (A180.)
(Index to next step and wait for cycle-complete signal).
M45

2.23 M19 SPINDLE STOP AND ORIENT

This is used to stop and orient the spindle. This orientation lock is released by using an M3 or M4 or by pressing the SPINDLE ON/OFF button.

EXAMPLE:

L100 (SUB FOR BROACHING KEY WAY, ONE STROKE
G91 G1 F50. Z-.3 G9
Y-.01 G9
Z.3 G5 G9
Y.012 G9
L200 (SUB FOR ALL BROACHING STROKES, .300 TOTAL KEY STROKES
M19 (Orient spindle for broaching tool).
L199
L151
M17

2.24 M20 GENERAL PURPOSE INDEXER

✓ OPTIONAL FEATURE

This is used to send a start signal to a device attached to the indexer interface board (PCB 0008 or PCB 0007) from FADAL. After the start signal has been sent, the CNC will wait for a cycle-complete signal from the device before the CNC will continue execution of the program. If the M20 is used with a fixed cycle, the M45 code must be used to execute the cycle because the control does not recognize the M20 as a move.

EXAMPLE:

G82 G99 R0.1 Z-.17 F45. P18 M45
M20 (A90.) (Indexer moves to next position and waits for a cycle-complete signal).
M45 (M45 is used because M20 is not considered a move by the control).

2.25 M30 END OF ALL SUBROUTINES (FORMATS 1&2)

This code is used for two reasons: to mark the end of the subroutine section of a program, and to end a main program.

End of subroutine section marker: This is a Format 1 feature, however, it can be used in a Format 2 style program because features from the two styles can be intermixed. If the M30 is used for an end of subroutine marker, use M2 as the end of the main program (see section 5.0 SUBROUTINES & SUBPROGRAMS, MAN-0131). This code must be the only code on the line.

When the auto button is pressed or the AU command is used, the control will recognize the L100 as a subroutine and then search for other subs and then the M30 code. Program execution will start from the line **just after** the M30 line.

EXAMPLE:

O54 (P/N 543-W23
L100 (SUB FOR HOLES (Subroutine Section).
X2.965 Y-1.1107
X4.1

M17
M30
(This is the end of the subroutine section of this program).
(MAIN PROGRAM
(Program execution begins after the M30 code).

2.26 M30 END OF PROGRAM (FORMAT 2)

This is a format 2 feature. The M2 and the M30 both function as end of program codes. If this code is used in Format 1 it will act like the M2 code acts in Format 1.

If M30 or M2 are used in Format 2, they will:

- Cancel the current tool length offset.
- Cancel the spindle and coolant functions.

EXAMPLE:

G0 G90 G53 Z0
E0 X0 Y0
M6 T1
M30
(This is the end of the main program).

2.27 M31 EXCHANGE PALLETS

✓ OPTIONAL FEATURE

M31 performs a pallet exchange. The pallet changer will store the current pallet and load the other pallet. No other machine movements will be made. This is the only code allowed on the program line.

NOTE
All fixture and machine offsets MUST be canceled prior to attempting a pallet exchange (see Section 17.0 PALLET CHANGER, MAN-0131).

NOTE
The table move to the pallet position may be stopped with the SLIDE HOLD button. The pallet movement to and from the stored position may also be stopped with the SLIDE HOLD button.

See also G17.1 and G17.2

2.28 M32 LOAD AND STORE PALLET A

OPTIONAL FEATURE

When Pallet B is in the load position, M32 will store Pallet B and the pallet arm will move to Pallet A. The machine will be placed in the WAITING state and the pallet door will remain open until the START button is pressed. When the START button is pressed Pallet A will be returned to the load position.

When Pallet A is in the load position, M32 will store Pallet A. The machine will be placed in the WAITING state and the pallet door will remain open until the START button is pressed. When the START button is pressed Pallet A will be returned to the load position.

> NOTE
> All fixture and machine offsets MUST be canceled prior to attempting a pallet change (see section 17.0 PALLET CHANGER, MAN-0131).
>
> NOTE
> The table move to the pallet position may be stopped with the SLIDE HOLD button. The pallet movement to and from the stored position may also be stopped with the SLIDE HOLD button.

2.29 M32.1 LOAD PALLET A & VERIFY PALLET A HAS BEEN LOADED

OPTIONAL FEATURE

When Pallet B is in the load position, M32.1 will store Pallet B and load Pallet A.

If Pallet A is at the load position M32.1 will verify Pallet A is at the load position. No movement will occur (see section 17.0 PALLET CHANGER, MAN-0131).

> NOTE
> The table move to the pallet position may be stopped with the SLIDE HOLD button. The pallet movement to and from the stored position may also be stopped with the SLIDE HOLD button.

2.30 M33 STORE AND LOAD PALLET B

OPTIONAL FEATURE

When Pallet A is in the load position, M33 will store Pallet A and the pallet arm will move to Pallet B. The machine will be placed in the WAITING state and the pallet door will remain open until the START button is pressed. When the START button is pressed Pallet B will be returned to the load position.

When Pallet B is in the load position, M33 will store Pallet B. The machine will be placed in the WAITING state and the pallet door will remain open until the START button is pressed. When the START button is pressed Pallet B will be returned to the load position.

NOTE

All fixture and machine offsets MUST be canceled prior to attempting a pallet change (see section 17.0 PALLET CHANGER, MAN-0131).

NOTE

The table move to the pallet position may be stopped with the SLIDE HOLD button. The pallet movement to and from the stored position may also be stopped with the SLIDE HOLD button.

2.31 M 33.1 LOAD PALLET B & VERIFY PALLET B HAS BEEN LOADED

OPTIONAL FEATURE

When Pallet A is in the load position, M33.1 will store Pallet A and load Pallet B.

If Pallet B is in the load position, M33.1 will verify that Pallet B is at the load position. No movement will occur (see section 17.0 PALLET CHANGER, MAN-0131).

NOTE

The table move to the pallet position may be stopped with the SLIDE HOLD button. The pallet movement to and from the stored position may also be stopped with the SLIDE HOLD button.

2.32 M41-M43 BELT DRIVE RANGE

Newer machines have automatic high/low belt changes.The M41 - M43 codes are used on older machines that require a manual belt range change. This code could be used in MDI to indicate which belt range was selected:

M41 is 150 to 2700 RPM (top drive pulley)
M42 is 150 to 5200 RPM (middle drive pulley)
M43 is 300 to 10000 RPM (bottom drive pulley)

2.33 M45 EXECUTE FIXED CYCLE

This code is used to execute a fixed cycle or a modal subroutine. Execution will occur at the current location. Normally, motion to a new position causes a cycle to execute. Use the M45 when execution of a cycle is desired without making a move.

M45 is generally used with an indexer. The motion of the indexer is independent of the control. Even though the operator sees the rotation of the indexer, this is not counted as motion to the control, so the cycle is not executed with indexer motion. The M45 after each M20 or M18 will execute the cycle after each rotation.

EXAMPLE:

G82 G99 R0.1 Z-.17 F45. P18 M45
M20 (A90.) (Indexer moves to next position and waits for a cycle-complete signal).

M45 (M45 is used because M20 is not considered a move by the control).

2.33.1 M45 USED WITH FIXTURE OFFSETS

M45 is also used when motion to the next fixture offset is at the same point as the current fixture. For example E1 X1. and E2 X1. would appear to the machine as no motion, even if the operator can see the motion from one fixture to the next. When the move is to the same position on the next fixture this is called a “null move” and is recognized as not having moved. The M45 is then used to execute the cycle after this null move.

EXAMPLE:

E1 X1. Y-1. (Move to first position).
G82 G99 R0.1 Z-.17 F45. P18 M45 (Start cycle and execute it).
E2 X1. Y-1. (Move to second position, first null move).
M45 (M45 is used because of the null move).
E3 X1. Y-1.
M45

2.34 M46 POSITIVE APPROACH

SPECIAL FEATURE

This code causes the machine to move the X and Y axes in a negative, then positive, direction before the execution of a fixed cycle (G73-G89).

The X and Y axis move .015 in the negative direction at 25 inches per minute, then move .015 in the positive direction at 3 inches per minute. This returns the machine back to the original location and then the fixed cycle will execute.

A Q WORD on the line with an M46 will define the amount to move for the positive approach if more or less than .015 is desired.

M46 can be coded on any block of the program except for blocks containing any words that need to be on a line by themselves. Only the X and Y positions that follow a fixed cycle are affected by this code. This function is modal and will remain in effect until canceled by an M47.

The feed and speed pots are disabled during the positive approach moves, and are enabled after the moves are complete.

EXAMPLE:

M46 Q.025 (Start positive approach mode).
G82 G99 R0+.1 Z-.18 F45. P18 X3.45 Y.78 (Move, take positive approach, drill).
X3. Y.89 (Move, take positive approach, drill).
M47 (Cancel positive approach).

2.35 M47 CANCEL POSITIVE APPROACH

This code is used to cancel the positive approach mode.

2.36 M48 POTENTIOMETER CONTROLS IN

This code enables the operator to override the programmed feed rate and spindle RPM by use of the potentiometers located on the pendant just above the jog selection switches. This code would only be required when an M49 code is used in the program.

EXAMPLE:

M49 (Cancel the operator's ability to override the feed rate and RPM).
G85 G99 R0+.3 Z-.7 F100. X3.78 Y1.
X3. Y5.M48 (Enable the operator to alter the feed rate and RPM).

2.37 M49 POTENTIOMETER CONTROLS OUT

This code disables the potentiometers located on the pendant just above the jog selection switches. See the M48 example above.

2.38 M48.1 & M49.1 SERVO COOLANT POTENTIOMETER CONTROLS IN/OUT

✓ OPTIONAL FEATURE

✓ SPECIAL FEATURE

M48.1 enables and M49.1 disables the override pot. This is to allow and disallow manual movement of the nozzle from the override pot for the servo coolant nozzle.

2.39 M48.2 & M49.2 PALLET A ROTARY TABLE OVERRIDE POTENTIOMETER

✓ OPTIONAL FEATURE

M48.2 enables and M49.2 disables the Pallet A rotary table axis override pot. This potentiometer is available on VMCs with the pallet changer/rotary table combination. When Pallet A is in the store or in the working position, this potentiometer allows the operator to rotate the rotary table on Pallet A for loading and unloading parts (see section 17.0 PALLET CHANGER, MAN-0131).

M49.2 will automatically return the rotary table, A axis, to the original position before rotation with the pot and must always be used when the M48.2 is used. Occasionally a G4 P# will have to be used after the M49.2 to insure that it rotates to its original position.

EXAMPLE:

N300 E0 X0Y0 A0
N301 M33.1 (LOAD PALLET B, STORE PALLET A
N302 M48.2 (PALLET A ROTARY OVERRIDE ENABLE
...
N506 M49.2 (PALLET A ROTARY OVERRIDE DISABLE
N507 E0 X0 Y0 A0
N508 M32.1 (LOAD PALLET A
N509 G4 L99

> NOTE
> When using the M48.2 or M48.3 with the A or B axes, insert a new line after the M48.2 or M48.3 and place a G4 P99 on this line. The G4 P99 is a code used to dwell or wait a millisecond amount of P. The dwell in this case is used to wait 99 milliseconds until the A or B axis has returned to a set position before continuing the program. (99 milliseconds is a suggested amount for this situation. The P can be increased or decreased.) If the wait (P amount) is not long enough, the control will time out waiting for the A or B axes to return to position.

2.40 M48.3 & M49.3 PALLET B ROTARY TABLE OVERRIDE POTENTIOMETER

OPTIONAL FEATURE

SPECIAL FEATURE

M48.3 enables and M49.3 disables the Pallet B rotary table axis override pot. This potentiometer is available on VMCs with the pallet changer / rotary table combination. When Pallet B is in the store or in the working position, this potentiometer allows the operator to rotate the rotary table on Pallet B for loading and unloading parts. (See section 17.0 PALLET CHANGER, MAN-0131)

M49.3 will automatically return the rotary table, B axis, to the original position before rotation with the pot. This must always be used when the M48.3 is used. Occasionally, a G4 P# will have to used after the M49.3 to insure that it rotates to its original position.

EXAMPLE:

N507 E0 X0Y0 A0
N508 M32.1 (LOAD PALLET A, STORE PALLET B
N509 M48.3 (PALLET B ROTARY OVERRIDE ENABLE
...
N530 M49.3 (PALLET B ROTARY OVERRIDE DISABLE
N531 E0 X0 Y0 A0
N532 M33.1 (LOAD PALLET B

> NOTE
> When using the M48.2 or M48.3 with the A or B axes, insert a new line after the M48.2 or M48.3 and place a G4 P99 on this line. The G4 P99 is a code used to dwell or wait a millisecond amount of P. The dwell in this case is used to wait 99 milliseconds until the A or B axis has returned to a set position before continuing the program. (99 milliseconds is a suggested amount for this situation. The P can be increased or decreased.) If the wait (P amount) is not long enough, the control will time out waiting for the A or B axes to return to position.

2.41 M60 - M69 USER ATTACHED DEVICES

Relay sockets are provided for the attachment of special devices such as brakes and clamps. Refer to the VMC Maintenance Manual for information regarding user attached devices.

If the VMC is equipped with Renishaw probe options, M64 is used to activate the MP8, MP11, and MP12 probe. M65 is used to activate the TS-20 or TS-27 tool setter. For the MP11 or the MP12, M66 is required along with the M64. For the Laser Probe, M67 is required along with the M64.

M60A Axis Brake On
M61A Axis Brake Off
M62B Axis Brake On
M63B Axis Brake Off
M64Activate MP8 or MP11 Probe
M64 M66Activate MP12 Probe
M64 M67Activate Laser Probe
M65Activate TS-20 or TS-27 Touch Probe

2.41.1 M-60 & M-62 FOR FIXED CYCLES

The use of a **minus sign** (M-60 or M-62) will cause execution of these functions after machine positioning and before execution of a fixed cycle.

EXAMPLE:

G81 G99 R0+.1 Z-.5 F20.
A90. M-60Move, Clamp, Drill
A180. M-60Unclamp, Move, Clamp, Drill

2.42 M80 AUTOMATIC DOORS OPEN

✓ OPTIONAL FEATURE

This function is used to open the automatic doors.

2.43 M81 AUTOMATIC DOORS CLOSE

✓ OPTIONAL FEATURE

This function is used to close the automatic doors.

2.44 M90-M93 GAIN SETTING

These codes are used to determine the way that the axes will respond during contouring moves. When using feed rates lower than F50. ipm, the M91 code should be used. M91 is the factory setting in the SV menu. When using high feed rates for contouring (above F50. imp), the M92 code would allow the machine to track closer. The M93 code is only used internally for the rigid tapping cycle.

2.44.1 M90 CHANGE AXIS GAIN: P WORD

This is the gain for each axis where a value of 100 is normal and 120 is 20% more than normal. Gain has a multiplying effect on the response of a servo system. Gain controls how hard the motors are driven. Larger numbers make the motors more responsive to velocity changes, but a gain that is too large will cause the system to be unstable and cause overshooting (as the feed rates increase, the gain should increase as well.) With the appropriate value of gain chosen, the axes will meet detail at higher feed rates. At around 150 IPM, a gain of 125 should be used. Below 40 IPM, a gain of 100 is appropriate.

M90 - DEFAULT (Determined by the SV command).
M91 - NORMAL
M92 - INTERMEDIATE
M93 - HIGH

The default (at power on) is determined by the setting on the axis controller card. The SV command is used to write the settings to the controller card.

2.45 M94 FEED FORWARD FUNCTION

✓ SPECIAL FEATURE

The M94 code is used for linear moves only, to increase accuracy during high speed surfacing where radical changes in direction occur. These moves are generally at a feed rate of 50 ipm or higher. CNC programs for 3D surfaces use many small linear moves (G1) to form surfaces and the G8 code is used to eliminate hesitation between each move. In most cases this is desirable, except where radical changes in direction occur.

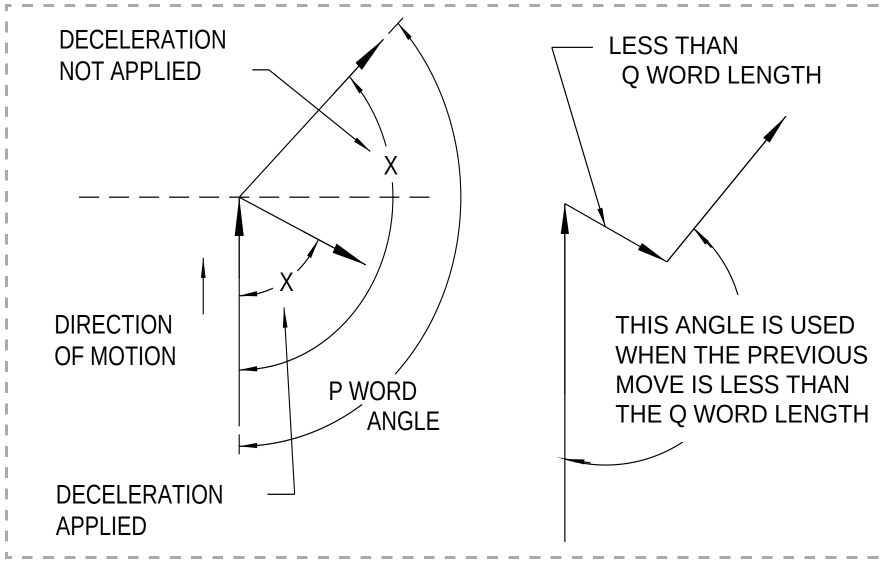


Figure 2-3: Feed Forward Function

The Feed Forward function affects the way the control accomplishes the acceleration and deceleration at the beginning and end of each move. When a move falls into the range assigned by the M94 parameters, the control will monitor axis servo feedback to determine how to move, instead of using reprocessed moves as it normally would.

The M94 and its parameters must be on a line with NO other codes. It is modal and is canceled by an M95. It only operates with the -3 processors or higher, and is not operational in the G91.1 mode.

Deceleration occurs when the angle between moves is smaller than the P word and the move length is greater than or equal to the Q word. The Q word is not a required parameter, and is used to filter out extremely small moves.

M94 P91 Q.003 (This sets the decel/accel for any move that is both .003 or longer and smaller than 91 degrees from the last move).

2.45.1 P WORD

The P word sets the angular tolerance for the feed forward mode, and must be less than 180 degrees. When the angle between the current direction and the next programmed direction is less than the P word, the machine initiates a high speed deceleration to increase the accuracy of the directional change. Acceleration then begins regardless of the length of the following moves. The acceleration may occur over an unlimited number of program blocks. Full acceleration is accomplished over a distance of two hundred thousandths when no further deceleration is required.

2.45.2 Q WORD

The Q word is used to set the length tolerance for the feed forward mode. When the length of the next programmed move is equal to or longer than the Q word, the control checks the angle between the current move and the next move. If this angle is less than the P word value, deceleration occurs.

> NOTE
> M94 and M94.1 can both be in effect at the same time.

2.46 M94.1 FEED FORWARD BY FEED RATE MODIFICATION

TIME SAVER

The M94.1 code is another type of feed forward mode used for high speed surfacing. The feed rate will be modified if the angle of the next move falls in the range established by the parameters of the M94.1 code line.

No other codes can be in the same line as the M94.1 and its parameters. The feed rate must be specified before the M94.1 line. No other F Word may be specified after M94.1 until an M95.1 is used. G0 moves can be used after M94.1 but will not be modified by the M94.1 coding. The M94.1 is modal and is canceled with an M95.1 code. It is available on -3 or higher controls. This is not compatible with the G91.1 code.

EXAMPLE:

M94.1 P170 Q10. R0+50. R1+1. R2+15.

SPECIAL FEATURE

The second move is 135 degrees from the first move, therefore the feed will be modified because the move is less than 170 degrees (set with the P word). Because the second move is less than one inch (set by the R1+1 word), the feed will be modified. The angular difference between the P word angle and the second move is 35 degrees. Every 15 degrees of angular difference (R2+15.), the feed will be modified by 10 percent (Q10). In this case the feed will be modified by 20 percent. In order for a move to be modified, it has to be less than the P value and less than the R1 value.

2.46.1 P WORD

The P word represents an angle. If the angle between the current move and the next move is less than the P word angle, the feed rate will be modified.

2.46.2 Q WORD

The Q word represents a percentage. This will be the amount that the feed rate will change each time it is modified (see R2 below for frequency of the modification).

2.46.3 R0+#

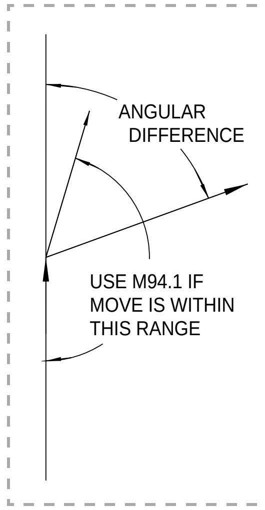


Figure 2-4: R0+#

The R0+# represents a percentage. This states that the modified feed rate should reduce no more than this percentage of the programmed feed rate.

2.46.4 R1+#

The R1+# represents a length. This states that if the next move is longer than this amount, then use the programmed feed rate for that move.

2.46.5 R2+#

The R2+# represents angular degrees. With the Q word modification percentage, this will be used to determine how the feed will be modified. This will modify the feed rate (by the percentage assigned to the “Q” word) every R2+# degrees for the current difference in angular moves by the percentage assigned to the “Q” word.

EXAMPLE:

N15 F100. G1
N16 M94.1 P170 Q10. R0+50. R1+1. R2+15.

The modified feed rate would be determined by this formula:
Fmodified = Fprogrammed - (Fprogrammed • Q word • Angular Difference / R2+#)

With an angular difference of 60 degrees and a programmed feed rate of 100. ipm, the modified feed would be 60 ipm:
*Fmodified = 100. - (100. * .1 * 60. / 15.) = 60.*

> NOTE
> M94.1 and M94 can both be in effect at the same time.
> The feed rate to be modified must be specified before the M94.1. No other F Word may be specified after M94.1 until an M95.1 is used. A new feed rate may be specified and then the M94.1 can be used again.

2.47 M94.2 ADVANCED FEED FORWARD (AFF)

OPTIONAL FEATURE

The advanced feed forward option is designed to satisfy the needs of high speed machining. Normally the gain, acceleration rate, deceleration rate, and detail factor on a machine tool is established to satisfy a large range of customer needs. Until now this did not directly target the specific needs of high speed contouring on surfaces. AFF allows the user to tune the machine to specific needs.

Production rate is important! AFF allows the machine to cut loose and fast for roughing cuts, tighter for semi-finish cuts, and very close tracking for finish cuts.

One method for controlling surface integrity is feed rate. Other controls will use what is termed “look ahead” to analyze angular change in a series of moves. The more dramatic a change, the lower the feed rate. This results in lower cycle times. AFF differs in that the feed rate is constant resulting in faster cycle times.

AFF allows five factors to be altered:

- Gain
- Deceleration
- Acceleration
- Detail
- Feed Rate

These factors can be altered on-the-fly, can be hard coded in the program, or the parameters, can be used from and stored in a parameter page.Use the DFF command to access the parameter page.Use the background edit menu to alter the parameters and change them on-the-fly.Use parametric variables to hard code the parameters in the program.

2.47.1 R1

DECELERATION: The time to decelerate the axes from programmed feed rate to a full stop measured in milliseconds. Deceleration is important to slow the tool down for smooth transitions into or around corners. The deceleration will improve the ability of higher feed rates to be used. At higher feed rates, a larger deceleration may be necessary to provide a smoother transition into the corners and to meet the specified detail. There is a point when the deceleration will not improve the quality of the part but will adversely affect the total part time. Pick a deceleration value that gives good part times while meeting the desired tolerances of the part. Values of 20 to 80 are appropriate for most feed rates. The deceleration ranges from 5 to 250 milliseconds.

2.47.2 R2

To pick up values from the DFF table, set the value of R2 to the corresponding tool number in the table. The parameters will then be used from the table.

2.47.3 P WORD

ACCELERATION: The time to accelerate the axes from a full stop to the programmed feed rate measured in milliseconds. The tool is accelerated out of corners or part details to the programmed feed rate. This is the approximate total time for the acceleration curve to bring the tool up to full speed. Values of 10 to 40 are appropriate for most feed rates. The acceleration ranges from 5 to 250 milliseconds.

2.47.4 Q WORD

DETAIL: The minimum detail acceptable is measured in inches. The detail parameter will hold the X, Y, and Z axes to a specified detail amount. This detail will dynamically change for each axis depending on the contour, but will always meet the programmed detail value. The ability of the axes to meet their detail is directly affected by the other AFF parameters. A larger gain will help the axis be “driven” to meet the detail specified. The deceleration will help the axes to softly move from the programmed feed rate down to zero speed and to the detail desired. The acceleration will not directly help the detail but will help when using faster feed rates. It will improve the transition from zero speed to the programmed feed rate. The detail value ranges from .0002" to .0250". The appropriate value depends on the part and tool. If it is a roughing tool, a larger detail should be used.

2.48 M95 FEED FORWARD CANCEL

This code is used to cancel the M94 mode. It is non modal and must be the only code on the line.

This code can also be used as a non modal form of the G9 code. If the program is in the G8 mode, an M95 on a line by itself will affect the next line in the program so that it will decelerate and accelerate. After the move is complete, the G8 mode will continue.

EXAMPLE:

N15 F100. G1
N16 M94.1 P170 Q10. R0+50. R1+1. R2+15.
...
N10350 X.001Y-.04
N10351 M95 (Cancel Feed Forward).

2.49 M95.1 FEED FORWARD BY FEED RATE MODIFICATION CANCEL

This code is used to cancel the M94.1 mode. It is non modal and must be the only code on the line.

2.50 M95.2 AFF CANCEL

This code is used to cancel the optional Advanced Feed Forward mode.

2.51 M96 INTERSECTIONAL CUTTER COMPENSATION CANCELED (ROLL CRC)

This code is used to cancel intersectional cutter radius compensation. The M96 code is modal and will remain in effect until the M97 code is used (see section 9.0 CUTTER RADIUS COMPENSATION, MAN-0131).

EXAMPLE:

N22 G1 F10. Z-.25
N23 M96
N24 G1 G4 1X0 F40.
N25 Y1.
N26 X2.

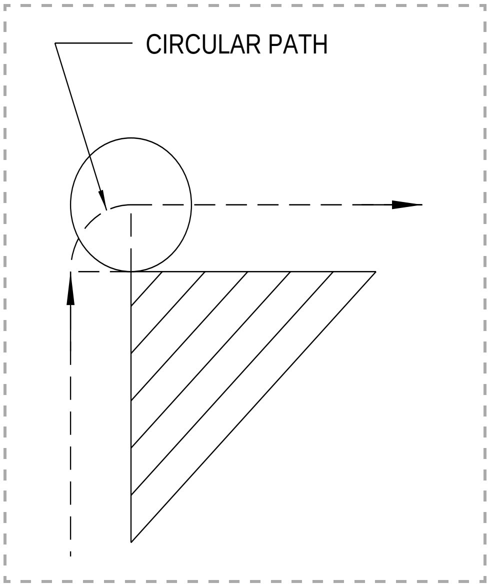


Figure 2-5: Circular Path

2.52 M97 INTERSECTIONAL CUTTER COMPENSATION

This code is used to start the intersectional cutter radius compensation mode. The M97 code is modal and will remain in effect until the M96 code is used (see section 9.0 CUTTER RADIUS COMPENSATION, MAN-0131).

EXAMPLE:

N22 G1 F10. Z-.25
N23 M97
N24 G1 G4 1X0 F40.
N25 Y1.
N26 X2.

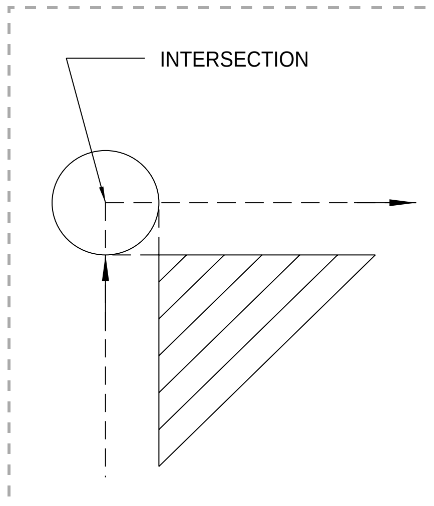


Figure 2-6: Intersection

2.53 M98 SUBPROGRAM (FORMATS 1 & 2)

This code is used to call a subprogram (see section 5.0 SUBROUTINES & SUBPROGRAMS, MAN-0131).

M98 P# L#

2.53.1 P WORD

Identifies the number of the subprogram to be called.

2.53.2 L WORD

Specifies the number of times to execute the subprogram.

EXAMPLE:

N22 X1.0
N23 M98 P3 L2 (Execute subprogram 3 and repeat 2 times).

2.54 M99 END OF SUBPROGRAM (FORMATS 1 & 2)

This code can be used to mark the end of a subprogram. If this is at the end of a program, this is the code that identifies a program as a subprogram. No other codes or parameters should be on the line with the end of subprogram marker (see section 5.0 SUBROUTINES & SUBPROGRAMS, MAN-0131).

EXAMPLE:

N1O3 (SUB PROGRAM 3
N2 M6T1
...
N47 M99 (Return to main program).

2.55 M99 LINE JUMP (FORMATS 1 & 2)

This code can be used for a line jump. With the P word parameter on the line with the M99, it becomes a line jump.

M99 P#

2.55.1 P WORD

Identifies the line number to continue execution from.

EXAMPLE:

N44 M0 (OPERATOR ADJUST BORING HEAD IF NEEDED
N45 S800 M3 M7
N46 Z.2 X0 Y2. G0
N47 G85 G99 R0+.2 Z-.356 F1.6 X0 Y0 M46
N48 G80 M5 M9
N49 X0 Y2. G0
N50 M0 (OPERATOR CHECK THE BORE SIZE
N51 (IF BORE IS GOOD THEN SWITCH BLOCK SKIP ON
/N52 M99 P44 (If the block skip switch is off, jump to line 44 for another pass).
N53 X1.5 Y2.

3.0 G CODES

3.1 G0 RAPID TRAVEL

G0 moves from one point to another point at the maximum traverse rate of the machine. G0 is generally used when cutting will not take place when moving from one location to another.

Multiple axis moves begin by all axes moving together at the same rate until each axis move is completed. This gives the appearance of a forty-five degree move at the beginning of the move. For the remaining distances, each axis will continue to move to the end point.

When using MDI, a rapid Z axis move will move independent of the X, Y, A & B axes. When the Z axis is to move in the positive direction, it moves prior to X, Y, A, & B axis motion. When the Z axis is to move in the negative direction, it moves after X, Y, A, & B axis motion.

G0 is modal and will remain in effect until it is canceled by the G1, G2 or G3 codes. G0 will not cancel any feed rates used by the interpolation modes. An F word can appear on the same line with a G0 code, however, the F word will only be used when an interpolation code is used.

G0 can appear at any point on a line to make all moves on the line rapid.

The rapid travel switch on the pendant can be used to alter the rapid travel rate.

The feed rate potentiometer will only affect the rapid rate during single step, just after a slide hold, and while in any of the dry run modes. See also G5 Non Modal Rapid.

EXAMPLE:

F30. (This F word is modal).
G0 G90 Z.1 (This line will be in rapid travel).
X1.3 Y2.7 (This line will be in rapid travel).
G1 Z-.245 (The G1 will cancel the G0 and use the F30. from above).
G91 X.5 (This will be at F30.0).
G90 Z.1 G0 (This line will be in rapid travel).

3.2 G1 LINEAR INTERPOLATION

This code is used for linear interpolation. Linear moves can be made by one, or any combination of, all the active axes. (See Linear Interpolation, section 13.0 INTERPOLATION, MAN-0131. Also, Rotary Axis Interpolation details, section 12.0 ROTARY AXES, MAN-0131.)

> NOTE
> Max program feedrate at 100% is 400 IPM.

3.3 G2 CIRCULAR INTERPOLATION ARC CLOCKWISE

G2 is used for CW circular interpolation and helical moves. (See section 13.0 INTERPOLATION, MAN-0131.)

G2 X1. Y1. I-1. J0

> NOTE
> Max program feedrate at 100% is 400 IPM.

3.4 G3 CIRCULAR INTERPOLATION ARC COUNTER-CLOCKWISE

G3 is used for CCW circular interpolation and helical moves. (See section 13.0 INTERPOLATION, MAN-0131.)

G3 X1. Y1. I-1. J0

> NOTE
> Max program feedrate at 100% is 400 IPM.

3.5 G4 DWELL

Whenever a pause in the program is required, use the G4 code.

A pause may be used to allow the spindle and coolant to fully turn on after using the M3 and M7 or M8 codes. This often happens with a tall part or fixture, where the tool gets to the top surface before the spindle is up to speed.

EXAMPLE:

G90 G0 S10000 M3 E1 X-.45 Y-.2
H1 Z-.3 M8
G4 P1000 (This one second dwell allows the spindle to come up to speed).
X3. G1 F80.

- A P word represents time. The time is given in milliseconds.
- P1 = 1/1000 second (or one millisecond)
- P500 = 500 milliseconds or 1/2 second
- P60000 = 1 minute

The G4 would also be used in a situation where the tool needs to dwell to allow for spindle rotation, such as a spot face or counter bore situation. A general rule to follow is to dwell for at least three revolutions. To calculate elapsed time during three revolutions divide 180,000 by the RPM used. The 180,000 represents time in milliseconds for three minutes.

EXAMPLE: For 5000 RPM:180,000 / 5000 = 36

G1 F10. Z-.25
G4 P36 (Dwell for 36 milliseconds; 3 revolutions at 5000 RPM).
Z0 G0

3.5.1 G4 AS AN IN-POSITION CHECK

The use of a G4 without the P word will perform an in-position check. This would be non-modal and would only affect the line in which it existed. See also G9.

X1.0 G4 (An in-position check is forced here).
X2.0
X3.0

3.5.2 G4 AS A PROGRAM STOP

SPECIAL FEATURE

The use of a G4 with P66000 forces an endless dwell or a program stop, placing the machine in the waiting state. When in the WAITING state the spindle and coolant will remain on, as opposed to M0 and M1 which turn them off. To continue program execution press the Start or Auto button.

3.6 G5 NON MODAL RAPID

SPECIAL FEATURE

The G5 code is used for non modal rapid moves. It exhibits the same motion as G0, however, this code will only affect the line in which it exists.

EXAMPLE:

X2.5 G1 F20.
G5 Z.1 (Rapid movement of this line only).
X3.0 Y-2.5 (The G1 is still in effect from above).

3.7 G8 ACCELERATION (NO FEED RAMPS)

TIME SAVER

This code is used when **no hesitation** is desired between moves. If the tool hesitates the tool pressure lessens and the tool will leave a tool mark on the contour. The G8 code would be used to eliminate the tool marks.

The hesitation is called a feed ramp or acceleration-deceleration. Ramping is used to help the tool move to the desired position.

- The G8 code is often used in combination with the M92 code.
- This code is modal and will remain in effect until the G9 code is used.
- The G8 code is a default code for format two.
- The G8 code is incompatible with a G41 or G42 coded on the same line.
- The G9 code is used to cancel the G8 code.

EXAMPLE:

G0 G8 G90 (Ramping is off at this line).
G2 I.5 G91 Z.02 L7
X-.5 G41
X.55 Y-.55 I.55 G3

- The M95 code is used as a non modal form of the G9 code. It is generally used when G8 is in effect. See M95 for more details.

3.8 G9 DECELERATION (FEED RAMPS)

This code is used when hesitation is desired between moves. When the tool hesitates the tool pressure lessens and the tool will leave a tool mark on the contour. The G9 would be used to help the tool move from place to place when inertia may be a problem. The use of the G9 code as opposed to using the G8 code will help insure contouring accuracy.

If an axis is faulting at a certain move, the G9 could be used to help the machine to get through the move by decelerating at the end of the move and then accelerating again at the beginning of the next move.

The deceleration will only slow the tool down at the end of the move. (It will not come to a complete stop).

- This code is modal and will remain in effect until the G8 code is used.
- This code is default for format one.

EXAMPLE:

X1.0 G9
X2.0
X3.0

3.8.1 G9 AS AN IN-POSITION CHECK

To stop the tool completely at the end of each move, an in-position check must be used. The G9 code, used in succession on two or more lines, causes an in- position check. Because of the look ahead processing, the line with the first G9 in successive order will use the in-position check. See also G4 and M95 for other forms of in-position check.

EXAMPLE:

X1.0 G9 (Because of the look ahead, the first G9 will be an in-position check).

X2.0 G9 (In-position check).
X3.0 G9 (In-position check).

3.9 G10 PROGRAMMABLE DATA INPUT

This code is used to replace, alter, or read the values of fixture offsets, tool offsets, and parameters R0 through R9.

- When G10 is used in the absolute mode (G90), the current value is replaced by the value identified by X, Y, Z, R0, or P.
- In the incremental mode (G91), the current value is altered by the positive or negative value of X, Y, Z, R0, or P.
- L# identifies the operation.
- P# selects the offset # or identifies the value for parameter R0 - R9.
- X#, Y#, Z# identifies the axis and the value to be changed.
- R0# identifies the value.

3.9.1 L2

Used to replace or alter fixture offsets: L2 P# (0, 1- 48) X# Y# Z# A# B#.

WITH G90

G10 L2 P5 X1.356 Y2.63 Z-.01 (This replaces the current value of fixture offset #5 with X1.356 Y2.63 Z-.01).

WITH G91

G10 L2 P5 X-1. (This subtracts one inch from the current X value of fixture offset #5).

Used to offset the part home position with a coordinate system shift (see G52).

WITH G90 OR G91

G10 L2 P0 X1. Y1. (P0 functions as a G52 X1. Y1.).

3.9.2 L10	Used to replace or alter tool length offsets: L10 P1-99 R0# With G90 *G10 L10 P12 R0 -5.467 (This replaces the current value of TLO #12 with -5.467).* With G91 *G10 L10 P12 R0+1. (This adds one inch to the current value of TLO #12).*
3.9.3 L12	Used to replace or alter a tool diameter: L12 P1-99 R0# WITH G90 *G10 L12 P1 R0+.52 (This replaces the current value of dia. offset #1 with .52).* WITH G91 *G10 L12 P1 R0-.02 (This subtracts .02 from the current value of dia. offset #1).*
3.9.4 L13	Used to read the values of a fixture offset: L13 P0,1-48 WITH G90 OR G91 *G10 L13 P2 (The R0 variable = the Z amount, the R1 = X, and the R2 = Y).*
3.9.5 L14	Used to read the value of a tool length offset: L14 P1-99 WITH G90 OR G91 *G10 L14 P2 (The R0 variable = the offset amount of TLO #2).*
3.9.6 L15	Used to read the value of a diameter offset: L15 P1-99 WITH G90 OR G91 *G10 L15 P2 (The R0 variable = the offset amount of diameter #2).*

3.9.7 L100 - L109

Used to replace or alter the value of a variable: L100 P#

WITH G90

G10 L102 P-6.5 (This replaces the current value of the R2 variable with -6.5).

WITH G91

G10 L102 P+.543 (This adds .543 to the current value of the R2 variable).

3.9.8 T

Used to facilitate turret tests.

WITH G90 OR G91

```
O5805 (WORK TOOL CHANGER
L100
G10 T1
M6.1 T1
T-2
M17
M30
L199 (PUT A TOOL IN SPINDLE
M0 (FINISHED TEST
```

3.10 G15 YZ CIRCULAR INTERPOLATION WITH THE A AXIS

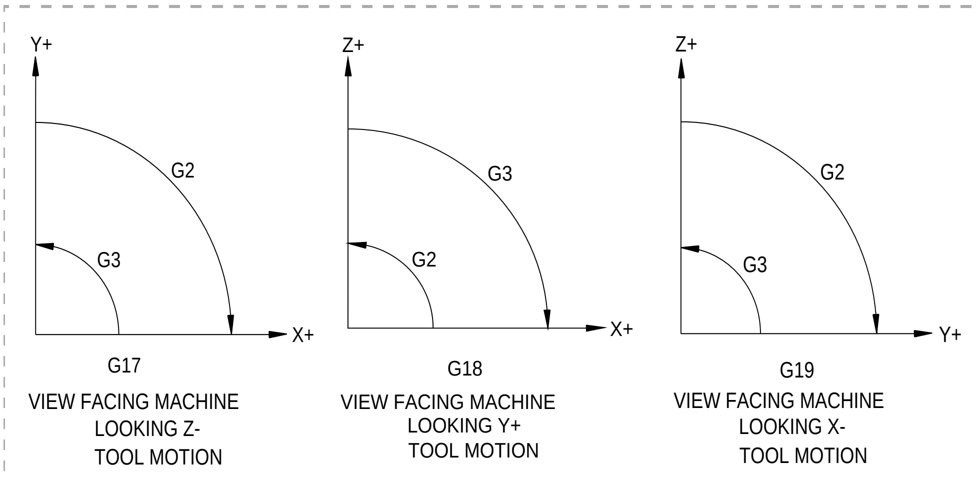
SPECIAL FEATURE

This code is used when the bottom of a cutter is required to cut an arc with Y, Z, and A axis motion. (See Section 12.0 ROTARY AXES, MAN-0131.)

3.11 G17-G19 PLANE SELECTION

Plane selection codes are used to identify the plane for such functions as:

- Circular Interpolation (G2, G3)
- Cutter Compensation (G40-G42)
- Coordinate Rotation (G68, G69)
- Flat Cam, XY Plane Conversion to XA/XB Motion (See section 12.0 ROTARY AXES, MAN-0131)




```
                      G2          R...
G17 - XY PLANE        G3   X...   Y...    I...    J...

                      G2          R...
G18 - ZX PLANE        G3   Z...   X...    K...    I...

                      G2          R...
G19 - YZ PLANE        G3   Y...   Z...    J...    K...
```

3.12 G17.1-G17.2 A/B WORD SWAP

SPECIAL FEATURE

The G17.1 word activates B-axis command substitution for the A-axis command. This allows the use of the A axis rotary moves in one program for use with rotary heads on both pallets. If the program calls for an A axis move and the rotary device is connected to the B axis controller, the G17.1 code will swap the A word for a B word. Existing programs written for the dual 4th axis setups that contain both A and B words are allowed. The G17.1 will automatically swap the B words to A words. The G17.2 word cancels A/B axis command swap mode.

3.13 G20 INCH PROGRAMMING

This code is used to verify that the operator has set the CNC to the INCH mode. This code **does not** place the machine in the inches mode. The inch mode is set with the SETIN or SETP command. (See SETIN and SETP command section 8.0 COMMANDS, MAN-0131.) Also, see the G70 code.

3.14 G21 METRIC PROGRAMMING

This code is used to verify that the operator has set the CNC to the METRIC mode. This code does not place the machine in the metric mode. The metric mode is set with the SETME or SETP command. (See SETME and SETP commands section 8.0 COMMANDS, MAN-0131.) Also, see the G71 code.

3.15 G28 RETURN TO ZERO

This code is used to return all axes (Format 1) or a specific axis (Format 2) to the current Tooling Coordinate System (Home Position). The G28 code acts the same in absolute and incremental mode.

3.15.1 FORMAT 1

Typically it is used to move home after a G92 or G52 code is used. The G28 will ignore the G92 preset position or a G52 shift and move to the positions established by the SET(X,Y, Z, A, B or H) commands or by a fixture offset. The G92 and G52 codes will remain in effect after the G28 move.

If a fixture offset is in effect, the G28 will return all axes to the fixture home position, unless a motion word appears with the G28 code. The G28 will not cancel the fixture offset. If a Z offset is in effect that is larger than +4.0 inches and the Z axis SET position is at the cold start position, the Z axis will over travel on the G28 line. It would be best to not use a G28 in this case. Instead, use an M6 to cancel the Z offset, then use an E0 X0 Y0.

- This code will cancel an H word if it is in effect.
- Motion words to a position in the same block with a G28 will be executed before the G28 and the position will be retained as the initial position during execution of a G29 code.

EXAMPLE: *M5 M9*
G28 All axes will move to the current home position
M0 (OPERATOR CHANGE CLAMPSL

EXAMPLE: *M5 M9*
G28 Y6. The Y axis will move to Y6.0 (the initial position) then all axes move home

3.15.2 FORMAT 2

Format 2 programming requires the axis to be specified in the block with the G28 for it to move to that position.

EXAMPLE: *G28 Z0 Only the Z axis will move to the zero position*
G28 X0 Only the X axis will move to the zero position

G28 will cancel the current fixture offset and move to the location established with the motion word on the same line as the G28 code, unlike Format 1 where G28 moves to the location in reference to the last called out fixture. The G28 moves are relative to the CS position or the SETX, SETY, SETZ, SETA, SETB, or SETH commands, whichever was used last.

3.16 G28.1 CANCEL JOG AWAY

This code is used to cancel the jog away amount and return specified axes to the current programmed position. This code is only intended to be used with option number two from the jog away return selection menu. The G28.1 code acts the same in absolute and incremental mode.

EXAMPLE:

Table 3-1: Cancel Jog Away

PROGRAMMED POSITION	MACHINE POSITION	JOG AWAY OFFSET
X3.0 Y3.0	X3.1000 Y3.1000	X0.1000 Y0.1000
G28.1 X0	X3.0000 Y3.1000	X0.0000 Y0.1000

> NOTE
> The value in the axis word with the G28.1 is irrelevant and is only used to determine which axis to cancel jog away.

3.17 G29 RETURN FROM ZERO

This code is used to return all axes to the initial position established with the last G28 code used in the program. Motion words in the same block as the G29 will be executed after the axes are returned to the initial position.

EXAMPLE: *G29 X5.0*

This only returns the X axis to the INITIAL position before moving incrementally the programmed amount. All other axes remain at their current location.

3.18 G31 PROBE TOUCH FUNCTION

The G31 is only used in conjunction with a probe. This code causes the machine to stop motion when the probe is touched and then execution continues at the next line in the program. The G31 can be used with table or spindle probes (see also G31.1).

- The motion can be defined in absolute or incremental terms.
- The positions can be stored with a P word, a macro V variable, and output through the RS232 port.
- All G31 moves must be G1 linear moves; no G0, G2, or G3 moves are allowed.
- Rotation can be in effect when the G31 is used.
- CRC should not be in effect when G31 is used.
- Mirrored axes should be canceled before using the G31 code.
- Fixed cycles need to be canceled before using the probe.

> NOTE
> Program a move that would normally be excessive. For example if a one inch move is required to get the probe up to a wall, use a two inch move in the program. The probe will stop the motion, and whatever motion is remaining for that line will be discarded and the control will continue execution of the program at the next line.

Expect some over travel, if the feed rate used with the move is high, and also if the stylus in the probe is long. A method for accuracy would be to use the G31 and the G31.1 codes together. Use the G31 at a high feed rate to get up to the wall. With the high feed rate the stylus is angled and over the edge because of the time required to read the probe and stop the motion. Then reverse the motion to move away from the wall with the G31.1 code in the line. Do this at a slow feed rate. At F1. the motion is slow enough that it will usually stop within one tenth. The G31.1 will stop motion when the probe is not touching. This means that the stylus will be perpendicular to the table and directly at the edge of the wall when the probe is not touching.

If the stylus is not running true, or a chip is in the spindle, the probe will not give a true position reading. For consistency, use an M19 to orient and lock the spindle at the same position each time the probe inserted in the spindle. If an operator is to place the probe in the spindle by hand, orient the spindle prior to inserting it in the spindle. Sometimes the stylus will work itself loose, confirm that it is tightly screwed in before using the probe.

3.19 STORING PROBED POSITIONS

3.19.1 SAVING POSITIONS THROUGH THE RS-232 PORT

1. Any software designed to save data from the port will be sufficient to retain the data.
2. When a touch is made, the motion will stop and the current position will be output through the port.

G1 G31 X1. F50. This line sends just the X axis location to the port
G1 G31 X2. Y5. F50. This line sends the X and Y locations to the port
G1 G31 X3. Y-4. Z-2. F50. This line sends the X, Y and Z locations to the port

3. Macro SPRINT statements can be used just before the probe line to identify the information being saved.

#SPRINT "PROBE TOUCH #1:"
G1 X1. Y1. G31

3.19.2 SAVING POSITIONS TO P WORDS

1. P1, P2, and P3 are used to save the touch positions when the fixed probe subroutines are going to be used in the program. (See Section 15.0 TOUCH PROBE, MAN-0131)

G1 X3. Y-6. G31 P1 The first touch position is saved to P1
X0 Y0 G5
G1 X0 Y6. G31 P2 The second touch position is saved to P2
X0 Y0 G5
G1 X-3. Y-6. G31 P3 The third touch position is saved to P3
L9101 R1+2. Use probe fixed subroutine function #2 to find center

2. P1, P2, and P3 can be used with the macro PX1-3, PY1-3, PZ1-3, PA1-3, and PB1-3 variables. When a probe touch (G31) or probe no-touch (G31.1) is used on a line with a P1, P2, or P3 each axis position is stored regardless of the axis that moved to get the touch point.

G90 G0 X3. Y-6.
Z1. H21
G1 F30. G31 Z-2.
F1. Z0 G31.1 P1 P1 has stored the XYZAB position at this line

#V1=PZ1
#PRINT "THE TOUCH POINT IS: X",PX1,", Y",PY1,", AND Z",PZ1

3.19.3 SAVING THE POSITION AS A V VARIABLE

The current position can be saved to a V variable by using a macro AX, AY, AZ, AA, or AB command.

G90 G1 X4. Y4. G31 F50.
X0 Y0 G31.1 F1.
#V1=AX This saves the X position to V1
#V2=AY This saves the Y position to V2

3.20 G31.1 PROBE NO TOUCH FUNCTION

This code causes the machine to stop motion when the probe is not touching and then execution continues at the next line in the program. The G31.1 can be used with table or spindle probes. This code functions exactly like the G31 code. See also G31.

3.21 G40 CUTTER COMPENSATION CANCEL

This code is used to cancel cutter radius compensation. (See section 9.0 CUTTER RADIUS COMPENSATION, MAN-0131.)

3.22 G41 CUTTER COMPENSATION LEFT

This code is used to activate cutter radius compensation to the left. (See section 9.0 CUTTER RADIUS COMPENSATION, MAN-0131.)

3.23 G42 CUTTER COMPENSATION RIGHT

This code is used to activate cutter radius compensation to the right. (See section 9.0 CUTTER RADIUS COMPENSATION, MAN-0131.)

3.24 G43 TOOL LENGTH COMPENSATION POSITIVE

This code is used to apply tool length compensation to the Z axis in the positive direction. This code is not required in programs primarily running on the FADAL control. This code is supported for compatibility with other controls. The direction of motion is determined by a combination of the sign of the offset value and the programmed G code (G43, or G44). See the chart below with G44.

3.25 G44 TOOL LENGTH COMPENSATION NEGATIVE

This code is used to apply tool length compensation to the Z axis in the negative direction. See the chart below. This code is not required in programs primarily running on the FADAL control. This code is supported for compatibility with other controls. The direction of motion is determined by a combination of the sign of the offset value and the programmed G code (G43, G44).

Table 3-2: Table Offset

CODE	IF THE OFFSET IN THE TABLE IS +	IF THE OFFSET IN THE TABLE IS -
G43	Tool moves in plus direction	Tool moves in minus direction
G44	Tool moves in minus direction	Tool moves in plus direction

3.26 G45 TOOL OFFSET SINGLE EXPANSION

This code is used for extending the programmed axis move by a value stored in the tool offset table. The value is determined by an H word. Program the H word in the same block with the G45 code and an axis move. **Only** the block containing the G45 code is extended.

> NOTE
> The G45, G46, G47, or G48 codes may **only** be used in X only, Y only, or quarter arc moves. No angular movements or full circles are allowed.

These codes were commonly used **before** CNC equipment had cutter radius compensation and fixture offsets.

EXAMPLE:

G0 G91
G45 X1.0 H1

The above example extends the 1.0 axis move by the tool length value of offset #1. **Tool length offset is not applied to the Z axis.**

To restore extended values to the original programmed values, a single reduction **must** be programmed in the opposite direction. See G46.

3.27 G46 TOOL OFFSET SINGLE REDUCTION

This code is used for reducing the program axis move by a value stored in the tool offset table. See G45.

EXAMPLE:

G0 G91
G46 X-1.0 H1

3.28 G47 TOOL OFFSET DOUBLE EXPANSION

This code is used for extending the program axis move by a value stored in the tool table. It is similar in function to a G45, but the value determined by the H word is doubled. See G45.

3.29 G48 TOOL OFFSET DOUBLE REDUCTION

This code is used for reducing the program axis move by a value stored in the tool table. It is similar in function to a G45, but the value determined by the H word is doubled. See G45.

3.30 G49 TOOL LENGTH OFFSET CANCEL

This code cancels the current tool length offset programmed by the H word. It causes the Z axis to move in the opposite direction and distance of the offset in the tool table.

- If the position of the Z axis is more than four inches above the program Z zero, using this code will cause the machine to over travel in the Z axis. If the tool is higher than four inches, use a G53 Z0 in place of the G49 Z0 codes. If the G53 Z0 is used, the M6 will automatically cancel the tool length offset.

EXAMPLE:

Z4.5 G80 M5 M9
G53 Z0 Use G53, not G49 when the Z is more than 4. above Z0
M6 T2
This code is similar to using the H0 code to cancel a tool length offset

EXAMPLE:

G90 G0 H0 Z0 or G90 G0 G49 Z0 Both would function the same
M6 T2
This code is used at the end of a program just before the M2 or M30 codes

EXAMPLE:

G90 G0 G49 Z0
E0 X0 Y0
M2
This code is used on a line just before each M6 line to prevent over travel or tool crashes during direct mid tape starts on the tool change line

3.31 G50 RAMP CONTROL CANCEL

This code resets the ramp control to the default values. See G51.

3.32 G50.1 MIRROR IMAGE CANCEL

This code is used to deactivate the mirror image mode.

EXAMPLE:

G51.1 X0
X3. Y-3.
G50.1 Deactivates mirror image

3.33 G51 RAMP CONTROL

TIME SAVER

This code is used to increase or decrease the length of time for the feed ramps between moves. A feed "ramp" is the time against feed rate on a graph. When a feed rate is specified it requires the user to specify the amount of time to reach that feed rate and a specify the amount of time to slow down at the end of a move. Imagine a truck at a stop sign. It takes a certain amount of time to get up to speed, as opposed to a car at the same stop sign, it would take less time to get up to speed. With less weight on the table, the ramps could be reduced. With a heavier weight on the table, the ramps may need to be lengthened.

A value between .5 and 2, default being 1, is specified with the R0 word. Values less than 1 will decrease the time, and values greater than 1 will increase the time.

The sign (+ or -) of the value identifies the controlled axis. For example, R0 used with a negative value controls the ramp length of the X and Y axis. Positive values control the Z axis.

EXAMPLE:

G51 R0-.6 This will decrease the length of all XY feed ramps
G51 R0+.6 This will decrease the length of a Z axis feed ramps

- Use the R0- and R0+ on separate lines.
- This code sometimes has a significant effect on the amount of time required to execute a particular program. When the feed ramps are shorter, the time to execute the program is reduced. The more moves that are involved in the program the more significant the time savings. The opposite effect would result if the ramps were lengthened.
- The shorter the ramps, the more stress is placed on the machine. If the machine does not respond favorably to shortened ramps, either don't adjust them or lengthen them. The undue stress will affect the axis system for each axis unfavorably and cause premature breakdown. The operator will notice excessive noise from the axes, hot motors, axis amplifier faults, and motor overload faults. If the operator notices any of these symptoms, lengthen the ramps.

3.34 G51.1 MIRROR IMAGE

This code activates the mirror image mode.

The axes to be mirrored are identified in the same block with the G51.1 code.

EXAMPLE:

G51.1 X0 Only the X axis will be mirrored
G51.1 X0 Z0 Both X and Z will be mirrored

- No motion will result from a block containing the G51.1 code.
- The mirror mode can be initiated from any position, but for all practical uses, it should be initiated from the zero position of the axis to be mirrored. This is especially true in absolute.
- Absolute and incremental moves can be mirrored.
- Use G50.1 to cancel the mirror mode.
- When mirroring contouring moves, the climb cuts become conventional and vice-versa. The program may require the G41 codes to be changed to G42. This is something that a programmer must determine. Sometimes left handed cutters with M4 can be used with contouring moves that have been mirrored.

EXAMPLE:

G0 G90 E1 X0 Y0 Move to the zero position of the axis to be mirrored
H1 Z.1 M7
G51.1 X0 Y0 Mirror X and Y
G1 Z-.25 F40.
X1. Y0 Mirror image position   X-1.0   Y0.0
Y-1. Mirror image position   X-1.0   Y1.0
X0 Mirror image position   X+0.0   Y1.0
G50.1 Cancel Mirror image

3.35 G51.2 TOOL LOAD COMPENSATION (TLC)

OPTIONAL FEATURE

This code activates the Tool Load Compensation (TLC) option. The G51.2 and the following parameters have been designed to automatically adjust the feed rate according to tool load conditions.

This option is a time saver because the feed rates can be increased automatically when conditions allow. Instead of using a generalized “safe” feed rate, the feed rate can be calculated for the maximum condition, and then automatically reduced by tool load conditions when it is being cut.

3.35.1 R1 = TARGET SPINDLE LOAD (PERCENTAGE)

The R1 variable represents the target spindle load to maintain. If the tool load is less than this amount, the feed rate will be increased; if the tool load equals or exceeds this amount, the feed rate will be reduced.

3.35.2 R2 = MINIMUM PERCENTAGE FEED RATE REDUCTION

The R2 parameter represents the lowest percentage to modify the feed rate. The lowest modification allowed is 20 percent. By reducing the feed rate the chip load will also be reduced. If the R2 parameter is too low, the reduced feed rate may cause excessive tool wear.

3.35.3 R3 = MAXIMUM PERCENTAGE FEED RATE INCREASE

When cutting conditions are correct and the spindle load is lower than what the R1 parameter is set for, the feed rate will be modified by the R3 percentage. This parameter must be considered carefully because it will affect the chip load of the tool. If the feed rate increases, so does the chip load. If the chip load increases too much it may cause the tool to break. It is suggested to select a maximum percentage for which the tool is designed. To determine this percentage, select an appropriate feed rate, multiply it by two thirds (.66666), and use the result for the feed rate in the program. Use an R3 value of +150. with G51.2 code. For example, if the appropriate feed rate is 30. then 30.*.66666=19.9998 or 20. Modifying 20. by 150% will result in maintaining the appropriate feed rate when the spindle load is lower than the target load factor.

3.35.4 R4 = NUMBER OF SECONDS AT MINIMUM FEED RATE UNTIL THE CONTROL ACTIVATES SLIDE HOLD

If the feed rate is programmed at the lowest feed rate modification established by the R2 parameter for longer than the R4 parameter value, the machine will be placed in SLIDE HOLD. The R2 parameter is the lowest feed rate modification. When used, it is an indication that the tool is getting dull or the cutting condition is excessive for the tool.

The time to remain in this condition must be determined carefully. It must be short enough to force the machine into slide hold when appropriate, and long enough to allow for intermittent periods of expected high load conditions.

Suggested parameters are given in the following example:

EXAMPLE:

G0 G90 E1 X0 Y0
H1 Z1. M7
G51.2 R1+60.0 R2+50.0 R3+150.0 R4+15.0 Activate TLC

*G1 F100. Z-.1 ** cut part***
G51.2 R1+0.0 Cancel TLC
M6 T2 An M6 will also cancel TLC

> NOTE
> The feed rate to be modified is on the line after the line where the G51.2 was used. No other feed rates should appear after the initial feed rate or until the G51.2 is canceled.

3.36 CANCELING G51.2 (TLC)

Use the G51.2 R1+0 in the program at the point where TLC is to be canceled. An M6 will also cancel the TLC mode. See the program example above.

3.37 TLC MANUAL TARGET POWER OVERRIDE

When the TLC is active in AUTO, the parameters will be displayed in the upper right portion of the screen. As the control adjusts the feed rates, the display will reflect the changes as they occur. During AUTO the operator may press the + or - button to manually override the specified target load parameters.

TAR%:60%	*MOD%:100%*	*POW%: 100%*
Target Power R1	*Programmed Feed Rate*	*Actual Power*

3.38 G51.3 AXIS SCALING

This allows the programmer to scale all or individual axis dimensions. The G51.3 code with the R1# parameter will scale all axes. The R2# will scale the X axis only. The R3# is used for the Y axis, and the R4# for the Z axis. The # with the parameters represents a percentage to scale. The percentage is represented in the decimal form. For example 2.0 would double the size, .5 would half the size.

EXAMPLE:

N1 O1 (PART 1234
*** Cut part ***
N2 M6T1 (TOOL #1
N3 G0 G90 S2500 M3 E1 X0Y0
N4 H1 D1 Z.1
N4 G51.3 R1+2. Scale all axes by 2 times scale factor 2
*** Cut part ***
N4074 G51.3 R1+1. Cancel scaling or scale factor 1
N4075 G0 G90 H0 Z0

Circular moves will be scaled according to the axis being scaled. If the X axis is scaled, the I for the circle center description will be scaled in the same proportion. The same would apply for the Y and Z axis. When the circles are to be scaled, it is suggested that the axes of the plane selection be scaled proportionally. For example, in G18 the X and Z axes should be scaled at the same percentage.

3.39 G52 COORDINATE SYSTEM SHIFT

This code is used to shift the current Tooling Coordinate System (TCS) similar to the way a fixture offset would be used except that the data for the shift is coded in the program. The current TCS would have been established by either the CS command, the SETH or SET (axis letter) commands, the G92 code, or the fixture offset codes G54-59 and E0-48.

This code is used when an absolute subroutine or subprogram needs to be used at different locations. Whereas an incremental subroutine or subprogram can be repeated from any location.

> NOTE
> G52 does not cause any motion to take place. It only references the new location on relation to the original zero.

EXAMPLE:

```
L100 (SUB FOR POCKET
G90 G0 X2.Y-2.
Z-.1 G1 F10.
X2.5 G41 F30.
Y-1.
X1.5
Y-3.
X2.5
Y-2.
X2. G40
Z.1 G0
M17
M30
 — Program Body —
X2. Y-2.  This is the original position
L101  Call subroutine 1, 1 time
G52 X2.  Shift original X0  position 2 inches from home
L101  Call subroutine 1, 1 time
G52 X4.  Shift original X0 position 4 inches from home
L101  Call subroutine 1, 1 time
```

3.39.1 CANCEL G52

G52 X0 Shift is canceled to original X0 home

The G52 shift amount is canceled by using another G52 in the program with a zero shift amount. See the program example above.

3.40 G53 MACHINE COORDINATE SYSTEM

This code causes the control to use the machine tool coordinate system. The machine tool coordinate system is established when the cold start, CS command was used. This code is useful when it is desired to move to an object that is secured to the table. The object may be something that is used by many fixtures or tools from many different jobs. One use may be the TS-27 probe for setting tools. Another use may be a diamond, used for dressing grinding tools.

The G53 is a non modal code. It will affect only the line in which it exists.

EXAMPLE:

G90 X0 Y2. This position is relative to the part home
G53 Y0 The tool will move to the cold start Y0 position
Y0 This position is relative to the part home

- The G53 should be the only G code in the line.
- Code an X position, Y position or any axis position with the G53 to indicate where to move in relation to the machine tool coordinate system.

EXAMPLE:

M5 M9
G53 Z0
M6 T4
G53 X-19.75 Y-9.8 (MOVE TO TABLE PROBE
Z-30. G1 F60. G31

- A G53 Z0 is usually used on the line just prior to an M6. This will make a quicker tool change, and it offers some insurance when doing mid tape starts that the tool will not crash into the part.

3.41 G54-G59 FIXTURE OFFSETS

These codes may be used for fixture offset locations E1 - E6. Specify a G54 code to access fixture offset number 1, a G55 code for number 2, and up to a G59 code for number 6. For fixture offsets after number 6, the E words must be used.

These codes are supported for compatibility and can be used in both format one and two. (See section 11, FIXTURE OFFSETS, MAN-0131.)

EXAMPLE:

G0 G90 S3000 M3 G54 X0 Y0
H1 Z1. M7

3.42 G66 MODAL SUBROUTINE

✓ SPECIALF EATURE

This code defines a subroutine as being modal. The subroutine is executed at each X, Y, A, B position programmed or when an M45 is coded, in the same manner as any fixed cycle would be repeated.

This code is a time saver (and a memory saver) in that the programmer does not have to type the sub call after each positioning move. It is a memory saver because the memory space used for each sub call is no longer needed.

- Use G67 to cancel the modal subroutine call.
- Subs can be incremental or written in absolute.

EXAMPLE:

O1
L100
G0 G90 Z.05
G1 G91 X.2 Z-.05 F10.
I-.2 G3 Z-.1
I-.2 G3
X-.2
G90G0 Z.05
M17
M30
M6 T1
G0 G90 S3000 M3 E1 X0 Y0
H1 Z1. M7
G66 L101 Defines subroutine 1 to be modal
X3. Y-3. Repeat Subroutine 1 at this location
X6. Y-3. Repeat Subroutine 1 at this location
G67 Cancel G66

- Fixed subroutines and Fixed Cycles cannot be used in a subroutine that will be modal, however they can be in a subroutine that will not be modal.

EXAMPLE: This is the incorrect form of fixed cycles and subroutines in a sub.

O1
L100
G81 G99 R0+.1 Z-.5 F40.
L9307 R0-.75 R1+0 R2+45.
M17
M30
M6T1 (DRILL
G90 G0 S10000 M3 E1 X3. Y-3.
H1 Z.1 M8
G66 L101

> NOTE
> Example 2 is not possible without modification.

This is the correctly modified form when fixed cycles and subroutines are in a sub.

O1
L100
G67 Cancel the modal subroutine at the beginning of the sub
G81 G99 R0+.1 Z-.5 F40.
L9307 R0-.75 R1+0 R2+45.
G66 L101 Make the subroutine L100 modal at this point
M17
M30
M6T1 (DRILL
G90 G0 S10000 M3 E1 X3. Y-3.
H1 Z.1 M8
L101 The sub will be repeated at X3. Y-3.
X6. The sub will be repeated at this location
Y-6. The sub will be repeated at this location
X3. The sub will be repeated at this location
G67 The modal sub is canceled here

3.43 G67 CANCEL MODAL SUBROUTINE

The G67 cancels a modal subroutine. The G67 works in the same way as a G80 cancels a fixed cycle.

X6. Y-3. Repeat Subroutine 1 at this location
G67 Cancel modal Subroutine 1

3.44 G68 COORDINATE SYSTEM ROTATION

The G68 activates a mode to rotate the coordinate system of the current plane (Selected by G17, G18 or G19). In G17 only X, Y, I, and J are rotated. In G18 only X, Z, I, and K are rotated. In G19 only Y, Z, and K are rotated.

The angle of rotation is coded in decimal degrees by the R0 word. A positive value designates counterclockwise rotation. A negative value designates clockwise rotation. An X, Y or Z word coded with the G68 defines the rotation center and must be in absolute (G90) terms. All parameters must be in the line with the G68 code.

EXAMPLE:

G68 R0+.56 X0 Y0 Rotate the program around X0 Y0 +.56 degrees
G68 R0-1.2 X1. Y-.5 Rotate the program around X1. Y-.5 by - 1.2 degrees

RULES

- CRC can be used after rotation is in effect and should be canceled before G69 is used. A part program cannot be rotated while CRC is in effect.
- Rotation continues until a G69 is coded.
- Fixture offsets are allowed with rotation. The moves to the offsets are not rotated.
- Rotation must be established prior to Fixed Cycle definitions and affects only the positions for execution. Fixed cycles and Fixed Subroutines will not be rotated to another plane.
- All X and Y (or X, Z or Y, Z or X, Y, and Z) positions are required for linear moves, even if they are zero or non-motion moves.
- In the selected plane, all X, Y, I and J (or X, Z, I, K or Y, Z, J, K) positions are required for circular moves, even if they are zero or non-motion moves.

EXAMPLE:

G17
G0 G90 E1 X0 Y.25
H1 Z.1
G1F5. Z-.3
G68 X0 Y.25 R0+45.
X0 Y0 G41 CRC turned on after rotation
X1.0 Y0 G1 F30. Code Y even though it is a non-motion move
X1.0 Y.5 I0 J.25 G3 Code X even though it is a non-motion move
X0 Y.5 Code Y even though it is a non-motion move
X0 Y.25 G40 Code X even though it is a non-motion move
G69 Cancel rotation

3.45 G69 COORDINATE SYSTEM ROTATION CANCEL

This code cancels the coordinate system rotation mode (see G68 for program example).

3.46 G70 INCH PROGRAMMING

This code is used to verify that the operator has set the CNC to the INCH mode. This code does NOT place the machine in the inches mode. The inch / metric mode is set with the SETIN or SETP command. (See SETIN and SETP commands, section 8.0 COMMANDS, MAN-0131.)

3.47 G71 METRIC PROGRAMMING

This code is used to verify that the operator has set the CNC to the METRIC mode. This code does NOT place the machine in the metric mode. The inch / metric mode is set with the SETME or SETP command. (See SETIN and SETP commands, section 8.0 COMMANDS, MAN-0131.)

3.48 G73-G76, G81-G89 FIXED CYCLES

These are a preset series of operations which direct Z axis movement and/or cause spindle operation to complete such actions as boring, drilling, tapping. The fixed cycle selection is modal. The cycle is repeated after each M45 or X, Y, A, or B axis move, until the cycle is canceled by a G80. (See section 4.0 FIXED CYCLES, MAN-0131.)

3.49 G80 FIXED CYCLE CANCEL

This code cancels the current fixed cycle.

EXAMPLE:

N13 X1.0 Y1.0
N14 G80

In Format 1, the Z axis will return to the initial plane.

In Format 2, the Z axis will return to the plane indicated by the use of the G98 or G99 code.

3.50 G90 ABSOLUTE INPUT

A control mode in which the motion data input is in the form of absolute dimensions. The values programmed with the axis words are the locations to move to in relation to the current zero position. (See section 11.0 COORDINATE SYSTEM, MAN-0131.)

Since blocks are processed in a left to right order, both G90 and G91 may appear in the same block. G90 and G91 are position sensitive, therefore the moves to the left of the G90 code will be in absolute until the G91 code is used. The G90 code is modal and will remain in effect until the G91 code is used.

EXAMPLE:

N12 G90 X2. G91 Y1. The X move will be absolute, the Y move will be incremental
N13 Z-.02 G5 This Z move will be incremental
N14 G90 X4. This X move will be absolute

3.51 G91 INCREMENTAL INPUT

This is a control mode in which the motion data input is in the form of incremental data. The values programmed with the axis words are the distance and direction to move in relation to the current location.

Since blocks are processed in a left to right order, both G90 and G91 may appear in the same block. G90 and G91 are position sensitive, therefore the moves to the left of the G91 code will be in incremental until the G90 code is used. The G91 code is modal and will remain in effect until the G90 code is used.

EXAMPLE:

N12 G90 X2. G91 Y1. The X move will be absolute, the Y move will be incremental
N13 Z-.02 G5 This Z move will be incremental
N14 G90 X4. This X move will be absolute

3.52 G91.1 HIGH SPEED EXECUTION

SPECIAL FEATURE

A control mode which enables high speed data block execution. **Using the 1400-2 processor**, the CNC executes up to 72 data blocks per second throughput, whereas normal execution is about 22 per second.

- When using the 1400-3 or -4 processor, it is not necessary to use G91.1, since the throughput is 250 data blocks per second.

In G91.1 mode, motion words must be programmed in incremental and be segmented. Mid-program (-tape) starts are not allowed in this mode. Subroutines or subprograms are not allowed in this mode.

The following codes are the **only** codes allowed during this mode of execution: G0, G1, G2,G3, G8, G9, M2, M3, M4, M5, M7, M8, M9, M95, X#, Y#, Z#, A#, B#, F#, I#, J#, K#, S#.

NOTE
This is best used in Format 2. The G91.1 code is canceled with the G91.2 code.

Format 1 G90 cancels the G91.2.

3.53 G91.2 HIGH SPEED EXECUTION CANCEL (FORMAT 2 ONLY)

The G91.2 is used to deactivate the high speed execution mode in Format 2 only. High speed execution is best used in Format 2. The G91.1 code is canceled with the G91.2 code. (See M94.1 for high feed rate machining, section 2.0 M FUNCTIONS, MAN-0131.)

3.54 G92 ABSOLUTE PRESET

The G92 is used to establish a temporary Program Coordinate System (PCS). The axis words coded in the same line with the G92 establish the current axis position to those axis words. For example, G92 X3. Y2. would establish the current position of the machine to X3. Y2. Then all subsequent axis words will be relative to this new position.

A G28 code can be used to return to the original tool coordinate system. To cancel the G92, move to the original tool coordinate system with a G90 G28 X0 Y0 or equivalent move, then code a G92 X0 Y0.

- No other codes are allowed in the same block with the G92 except X, Y, Z, A, or B.

EXAMPLE: *G90 G0 X3.75 Y-4.65*

G92 X2. Y0 Current location now is X2 Y0
G28 X0 Y0 Move to original home
G92 X0 Y0 Cancels the previous G92 preset

(See section 11.0 PROGRAM COORDINATE SYSTEM , MAN-0131.)

3.55 G93 I/T (INVERSE TIME) FEED RATE SPECIFICATION (IPM/ INCHES, DPM/ DEGREES)

A control mode in which the feed rate is specified as one divided by the time to complete the move. This value is usually computed by dividing the desired feed rate by the length of the actual tool path. (See section 12.0 ROTARY AXES, MAN-0131.)

3.56 G94 FEED RATE SPECIFICATION MMPM, IPM OR DPM

This is the default code and does not need to be coded in the program. The mode insures that the feed rate will be specified by Millimeters Per Minute, Inches Per Minute, or Degrees Per Minute. When rotary axes are programmed the feed rate is automatically in degrees per minute. When G93 is used, this code MUST be coded before a linear or rotary axis motion is programmed. (See section 12.0 ROTARY AXES, MAN-0131.)

3.57 G98 RETURN TO INITIAL PLANE

This is a control mode in which, after performing the fixed cycle, the Z axis is returned to the Initial plane. This location is identified by the Z axis location prior to a fixed cycle definition.

G90
H1 Z1. M7
G81 G98 R0+.1 Z-1. F20.X0 Y0
(AFTER DRILLING, Z RAPIDS UP TO INITIAL Z1.

(See section 4.0 FIXED CYCLES, MAN-0131.)

4.0 FIXED CYCLES

4.1 DEFINITION

A Fixed Cycle is a series of operations that directs Z axis and spindle movement to execute such actions as boring, drilling, and tapping. G codes 73-76 and 81-89 define the operation, while F, I, J, K, P, Q, R0, and Z words (cycle parameters) define the style of execution.

4.2 PROCEDURE TO INITIALIZE A FIXED CYCLE

1. Turn spindle on (M3,M4) at the desired RPM.
2. Position the tool to the I plane (Initial plane), a point above the part and fixture where the tool will clear all objects.
3. Define the Fixed Cycle with its parameters.
4. Specify a move with X, Y, A, or B locations to execute the cycle.

O1234 (DRILLING EXAMPLE
G0 G90 S10000 M3 E1 X.5 Y-.5 Start the spindle
H1 M8 Z.1 Position to the I plane
G82 G99 R0+.1 Z-.25 F45. P18 Define the cycle and its parameters
X1. Y-.6 Cycle is executed at this next location

4.3 I PLANE

The I plane (initial plane) is automatically established by the last Z axis position when the fixed cycle is defined. This is commonly called the maximum Z clearance position. It is usually established above the highest object on the part or fixture. It is used with the G98 code to “jump” over obstacles on the part or fixture. The I plane must always be equal to or above the R plane.

O1234 (DRILLING EXAMPLE
G90 G0 S10000 M3 E1 X.5 Y-.5 Start the spindle H1 M8 Z1.
Z.1 Position to the I plane. The I plane is the last Z position before the cycle line.
G81 G99 R0+.1 Z-.72 F45. Define the cycle. The R0 plane is equal to the I plane.
X1. Y-.6 Cycle is executed at this next location

4.4 R PLANE

This represents the location of the R0 plane (or minimum clearance plane). The R0 plane is where the tool will begin to feed into the material. This location must be below or equal to the I plane.

EXAMPLE:

G90 where the I is equal to the R0 plane
G90 Z.1
G81 R0+.1

EXAMPLE: *G91 where the I is equal to the R plane*
G90 Z.1
G91
G81 R0+0

EXAMPLE: *G90 where the R is below the I plane*
G90 Z.1
G81 R0-.2

EXAMPLE: *G91 where the R is below the I plane*
G90 Z.1
G91
G81 R0-.3

4.5 G 98 RETURN TO I PLANE AFTER FINAL Z

G98 is modal and will remain in effect until the G99 code is used. This code is also a default code. It is in effect after the HO command is used, or in format 1 when entering the MDI mode or when the auto button is pressed.

When this code is in effect, the tool will return to the I plane after reaching the final Z axis position. This code is used when the tool must "jump" over any obstacle. For this reason, it is important to establish the I plane above any obstacle. When this code appears on a line with a positioning move the tool will return to the I plane after the cycle has been executed and the tool has reached the final Z axis position.

4.6 G99 RETURN TO R0 PLANE AFTER FINAL Z

This code is modal and will remain in effect until the G98 code is used. When this code is in effect, the tool will return to the R0 plane after reaching the final Z axis position.

4.7 CYCLE EXECUTION

4.7.1 FORMAT 1

Fixed Cycles are activated by one of the following which is determined by the machine parameter settings (see SETP command):

1. Immediately execute a cycle when the cycle is defined. (Typical of format 2)
2. Wait for a positional move to execute a cycle. (Typical of format 1)

NOTE
For Option 2, the positional move can be in the cycle definition line. If any position words exist in the cycle definition line, the cycle will execute.

The advantage of option two is that a cycle can be initialized from any location, and then wait for positional moves. The positional moves can all be within a subroutine or subprogram. The first hole location does not have to be used in the cycle definition line.

4.7.2 FORMAT 2

Fixed cycles, in format two, will always execute immediately when the cycle is defined. The first location to execute the cycle must be in the cycle line or established prior to the cycle line. An L0, used for cycle suspension, can be in the cycle definition line to suspend execution.

4.7.3 FORMAT 1 & FORMAT 2

If one or more axis moves appear on the same line as the fixed cycle, the cycle will be executed at the specified position. The cycle will execute at the end of the move. The move can be a G0, G1, G2, or G3 move. The feed rate established before the cycle line is the feed rate that will be used for the positional moves made with G1, G2, or G3. The feed rate on the cycle line will only affect the Z axis move.

A cycle can remain in effect when moving from one fixture offset to another. If the position from the first offset to the next offset is the same, an M45 must be used to execute the cycle. If a cycle is to be executed without a positional move, use an M45 code to cause execution of the cycle at the current position.

A cycle definition must appear after a G68 line when rotation is being used. The positioning moves are the only moves that will be affected by the G68 code.

When using a rotary axis the brake may be applied before the execution of the cycle. Use an M-60 on the same line with the angular position. The machine will rotate to the angular position, apply the brake, then execute the cycle.

Fixed cycles may only be used in the G17 mode.

4.8 FIXED CYCLE PARAMETERS

4.8.1 F WORD

Feed rate for drilling and boring.

RPM for tapping (G74, G74.1, G75, G84 and G84.1) Format 1.

Feed rate for tapping (G74, G74.1, G84 and G84.1) Format 2

4.8.2 Q WORD

1. Incremental step distance for intermittent functions (G73, G83). I, J, K words may be used in place of the Q word.
2. Decimal thread lead for tapping (G74, G74.1, G75, G84, G84.1).

HOW TO COMPUTE THE Q VALUE FOR TAPPING CYCLES:

INCH MODE: Divide 1 by the threads per inch. Example:1/4-20 tap 1/20=.05

METRIC MODE: Multiply .03937 by the lead. Example: 8mm x 1.25 tap (1.25 is the lead): .03937 X 1.25= .0492. This will be the inch equivalent for the lead.

3. Y+ axis shift amount before raising the Z axis (G76)

4.8.3 P WORD

Dwell time in milliseconds (G76, G82, G88, G89).

1. Parameter used with G73 and G83 which specifies how far to stop above the bottom of the last peck. When no P word is specified, the tool will return, in rapid, to the bottom of the last peck.
2. Percentage factor to alter the retracting feed rate on tapping cycles (G74, G74.1, G75, G84, G84.1). When the value is 10 percent or less, only the feed rate is modified. Values over 10 percent modify the feed rate and the spindle speed.

4.8.4 I WORD: INITIAL PECK, X AXIS SHIFT G76

1. Initial Peck, the value of the first peck (G73, G83).
2. X axis shift amount before raising the Z axis (G76).

4.8.5 J WORD: REDUCING VALUE, Y AXIS SHIFT G76

1. Reducing Value of each peck until K is reached.
2. Y axis shift amount before raising the Z axis (G76).

4.8.6 K WORD: MINIMUM PECK

Minimum Peck Value (G73, G83).

4.8.7 L WORD: CYCLE REPEAT AND SUSPEND, L# OR L0

This represents a cycle repeat or suspend. The cycle will repeat at the line with this code the number of times indicated. The use of L0 will suppress the cycle at that line. This can be used to move around an obstacle without executing the cycle.

G81 G99 R0+.1 Z-.5 F20.
G91 X2. L5 Drill 5 holes 2.0 apart
G90 Y3. L0 Machine moves without drilling
X9. Machine moves and drills at this location

4.8.8 R PLANE: R0 +/- #

This represents the location of the R plane (or minimum clearance plane). The R plane is where the tool will begin to feed into the material. This location must be below or equal to the **I plane**.

In the absolute mode (G90), the value for this location is relative to Z zero. In the incremental mode (G91), the value for this location is relative to the **I plane**. The R0 value in the incremental mode can never be a positive value.

4.8.9 S WORD

Spindle Speed for tapping (G74, G74.1, G84 and G84.1) Format 2.

4.8.10 Z WORD

The Z word establishes two separate locations. The first is the Initial Plane. This is the location of the Z axis when the fixed cycle is defined. This should be the minimum clearance above all clamps and fixtures. When a G98 is coded the Z axis returns to the Z axis location after completion of the final Z depth. The second use for Z is the final Z depth of the cycle. In absolute mode (G90), it is the depth relative to zero. In incremental mode (G91), it is the depth below the R0 plane. The use of the Z word while a cycle is active will redefine the cycle's final Z depth until the cycle is canceled or changed. It does not redefine the Initial Plane. When it is desired to change the return point of the Z axis, the R0 must be redefined with the new final Z depth (see example).

4.8.11 FIXED CYCLE EXAMPLES

N1 O1234 (FIXED CYCLE EXAMPLE
N2 M6 T1 (TOOL #1
N3 G0 G90 S8000 M3 E1 X0 Y0 Start the spindle and move to fixture one
N4 H1 M8 Z.1 Move the tool to the initial plane .1 above the part.
N5 G73 G99 R0-.45 Z-1.2 F10. Q.26 X-.5 Y-.35 P.02 Define cycle and execute at hole #1.
N6 Y-.7 Execute cycle at hole #2 and remain at the R plane.
N7 Y-1.05 G98 Execute cycle at hole #3 and then return to the I plane.
N8 X-2.5 G99 Execute cycle at hole #4 and remain at the R plane.
N9 Y-.7 Execute cycle at hole #5 and remain at the R plane.
N10 Y-.35 Execute cycle at hole #6 and remain at the R plane.
N11 G80 Cancel the fixed cycle and return to the I plane.
The Z axis zero position for this example is established at the top of the part. The I plane is Z+.1 above the part. The R plane is established Z -.45 from the top of the part.

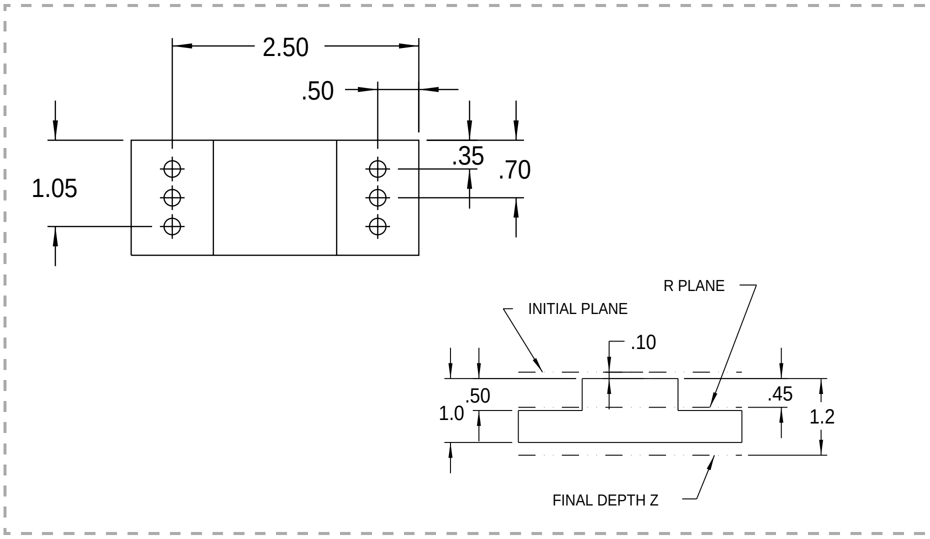


Figure 4-1: Fixed Cycle Examples

EXAMPLE:

The following example performs the same functions as before with various Z depth changes.

G90 Z.1
G81 G98 R0+.1 Z-.5 F40 X1.0 Y1.0
X2.0 Z-.7 R0+.1
X
X3.0 Z-.3 R0+.1
Y-1.0
Y-2.0 Z-.8 R0+.1

NOTE
Restate R0 even if not changed.

4.8.12 G73 PECK DRILLING USING Q

N3 G73 G99 R0+.1 Z-1.2 F10. Q.1 P.02 X-.50 Y-.35

Drill .25 each step (Q.26), returning incrementally +.05 after each peck, then down -.03 (.05 -.02 due to P.02) to the next peck plane, until the final depth is accomplished. The Q word can be calculated by dividing the distance from the R plane to the final Z by the number of desired steps. Example: 1.2+.1=1.3 for total distance then 1.3 (distance) / 10 (steps)=.1 for the Q value.

4.8.13 G73 PECK DRILLING USING I, J, K

N3 G73 G99 R0+.1 Z-1.2 F10. I.4 J.09 K.01 P.02 X-.50 Y-.35

Drill using varying step sizes, returning incrementally +.05 after each peck, then down -.03 (.05 -.02 due to P.02) to the next peck plane, until the final depth is accomplished. (See Deep Hole Drilling in this section for details.)

4.8.14 G74 LEFT HAND TAPPING

N3 G74 G99 R0+.4 Z-1.2 F550.2 Q.05 X-.50 Y-.35

Tap at 550 RPM in the high range, with a thread lead of .05 (20 threads per inch) to the final Z depth, then the spindle stops and reverses, retracting the Z axis. (See Tapping cycles in this section for details.)

4.8.15 G74.1 LEFT HAND RIGID TAPPING

N3 G74.1 G99 R0+.4 Z-1.2 F2000. Q.05 X-.50 Y-.35

Tap at 2000 RPM in the high range at a feed rate calculated by the CNC, the spindle stops and reverses, retracting the Z axis to the R plane. (See Tapping Cycles in this section for details.)

4.8.16 G75 TAPPING HEAD CYCLE

N3 G75 G99 R0+.4 Z-1.2 F2000. Q.050 X-.50 Y-.350

Tap at 2000 RPM in the low range, with thread lead of .050 (20 threads per inch) to the final Z depth, then feed up to the R plane,. (See Tapping cycles in this section for details.)

4.8.17 G76 FINE BORING

N3 G76 G99 R0+.1 Z-1.2 F10. Q.01 X-.50 Y-.35

Bore Z-.55, spindle stop-orient (the orientation will be the opposite of the starting spindle direction), shift the Y axis +.01, retract the Z axis, then move the Y axis -.01.

4.8.18 G81 SPOT DRILLING

N3 G81 G99 R0+.1 Z-1.2 F10. X-.50 Y-.35

Drill in one complete motion at a feed rate of 10.0 IPM.

4.8.19 G82 COUNTER BORING, CENTER DRILLING, SPOT FACING

N3 G82 G99 R0+.1 Z-1.2 F10. P23 X-.50 Y-.35

Bore to depth, dwell for 23 milliseconds then retract the Z axis at high speed. Dwell time for three revolutions: 180000/RPM = P word value.

4.8.20 G83 DEEP HOLE DRILLING USING Q

N3 G83 G99 R0+.1 Z-1.2 F10. Q.1 P.02 X-.50 Y-.35

Drill .1 each step, returning in rapid after each step to the R plane, then down in rapid to .02 (P.02) above the last step until the final Z depth is accomplished. The Q word can be calculated by dividing the distance from the R plane to the final Z by the number of desired steps. Example: 1.2+.1=1.3 for total distance then 1.3 (distance) / 13 (steps)=.1 for the Q value.

4.8.21 G83 DEEP HOLE DRILLING USING I, J, K

N3 G83 G99 R0+.1 Z-1.2 F10. I.4 J.09 K.01 X-.50 Y-.35

Drill using varying step sizes, returning after each increment to the R plane then down until final depth is accomplished. (See Deep Hole Programming Using G73 and G83 I, J, and K in this section for details.)

4.8.22 G84 RIGHT HAND TAPPING

N3 G84 G99 R0+.1 Z-1.2 F550.2 Q.05 X-.50 Y-.35 (Format 1)

Tap at 750 RPM in the high range, with a thread lead of .05 (20 threads per inch) to final Z depth, then the spindle stops and reverses, retracting the Z axis. (See Tapping Cycles in this section for details.)

4.8.23 G84 RIGHT HAND TAPPING USING P WORD

N3 G84 G99 R0+.1 Z-1.2 F550.2 Q.05 P2 X-.50 Y-.35

Tap at 750 RPM in the high range at a feed rate calculated by the CNC, the spindle stops and reverses, retracting the Z axis to the R plane at a rate faster than the in feed rate by 2%, as stated by the “P” word. (See Tapping Cycles in this section for details.)

4.8.24 G84.1 RIGHT HAND RIGID TAPPING

N3 G84.1 G99 R0+.1Z-1.2 F2000.2 Q.05 X-.50 Y-.35 (Format 1)

Tap at 2000 RPM in the high range at a feed rate calculated by the CNC, the spindle stops and reverses, retracting the Z axis to the R plane. (See Tapping cycles in this section for details.)

4.8.25 G85 BORE IN, BORE OUT

N3 G85 G99 R0+.1 Z-1.2 F10. X-.50 Y-.35

4.8.26 G86 BORE IN, SPINDLE OFF, ORIENT, RAPID OUT

N3 G86 G99 R0+.1 Z-1.2 F10. X-.50 Y-.35

4.8.27 G87 BORE IN, BORE OUT

N3 G87 G99 R0+.1 Z-1.2 F10. X-.50 Y-.35

4.8.28 G88 BORE IN, DWELL, BORE OUT

N3 G88 G99 R0+.1 Z-1.2 F10. P23 X-.50 Y-.35

Bore in at a feed rate of 10 IPM. Dwell for 23 milliseconds, then bore out. Dwell time for three revolutions: 180000/RPM = P word value

4.8.29 G89 BORE IN, DWELL, BORE OUT

N3 G89 G99 R0+.1 Z-1.2 F10. P23 X-.50 Y-.35

Bore in at a feed rate of 10 IPM. Dwell for 23 milliseconds, then bore out. Dwell time for three revolutions: 180000/RPM = P word value.

Table 4-1: Cycle Summary

CODE	MOVEMENT IN	DWELL AT BOTTOM	SPINDLE AT BOTTOM	MOVEMENT OUT	TYPICAL USAGE
G73	Steps	-	-	Rapid	Peck Drilling
G74	Feed	-	Reverses	Feed	LH Tap Compression Holder
G74.1	Feed	-	Reverses	Feed	Left Hand Rigid Tapping
G75	Feed	-	No Reverse	Feed	Self Reversing Tapping Head
G76	Feed	Yes	Orient, Axis Shift	Rapid	Fine Boring
G81	Feed	-	-	Rapid	Drill
G82	Feed	Yes	-	Rapid	Center Drill, Counter Bore
G83	Intermittent	-	-	Rapid	Deep Hole
G84	Feed	-	Reverses	Feed	RH Tap Compression Holder
G84.1	Feed	-	Reverses	Feed	Right Hand Rigid Tapping
G85	Feed	-	-	Feed	Boring
G86	Feed	-	Orient	Rapid	Boring
G87	Feed	-	-	Feed	Boring
G88	Feed	Yes	-	Feed	Boring
G89	Feed	Yes	-	Feed	Boring

G80 - Cancel Cycles

G98 - Return to initial plane after final Z

G99 - Return to R0 plane after final Z

4.8.30 G73 - PECK DRILLING USING Q

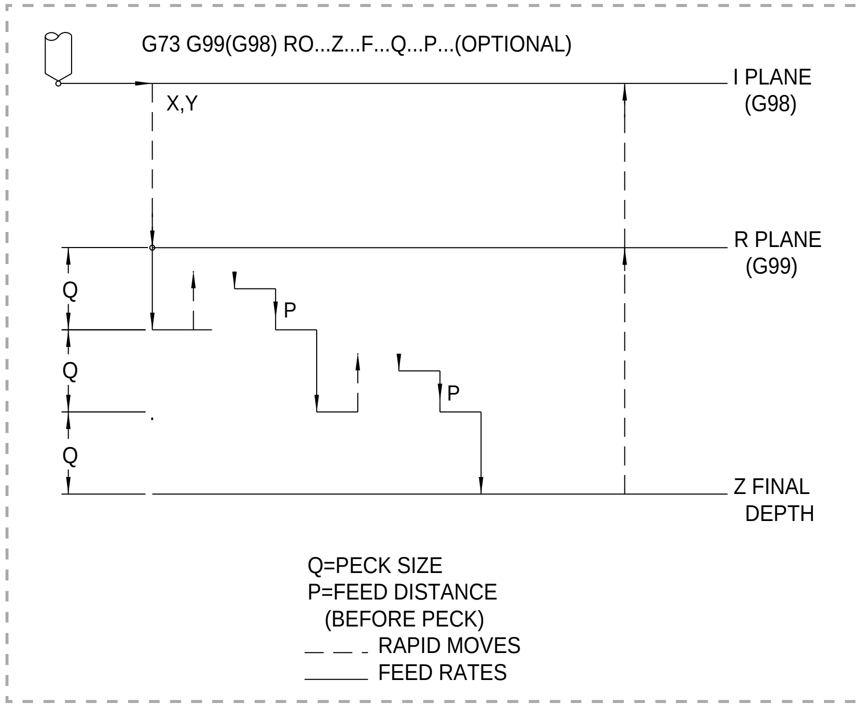


Figure 4-2: Peck Drilling Using Q

4.8.31 G73 - PECK DRILLING USING I, J, K

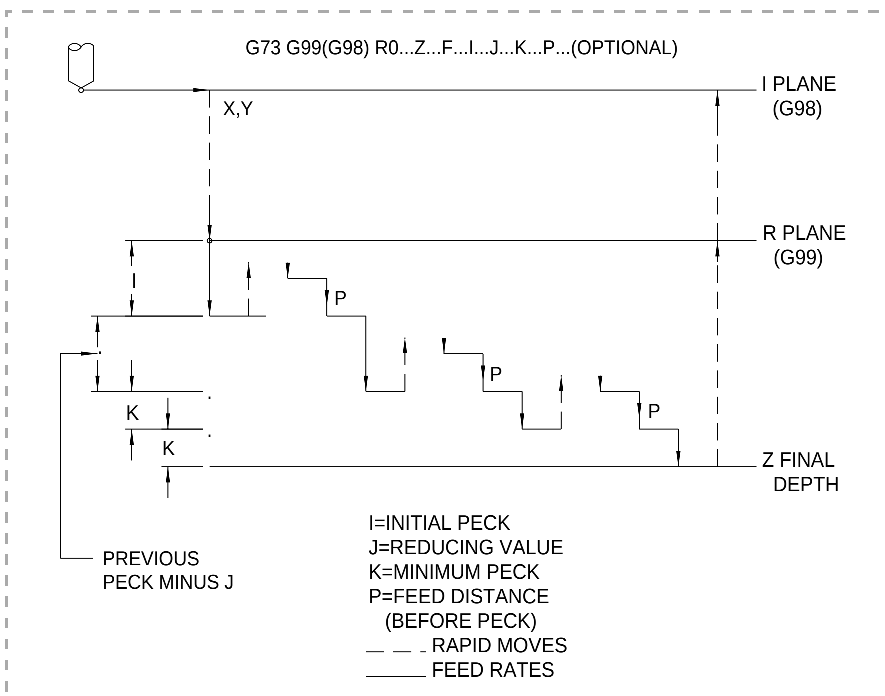


Figure 4-3: Peck Drilling Using I, J, K

4.8.32 G74 - LEFT HAND TAPPING (FORMAT 1)

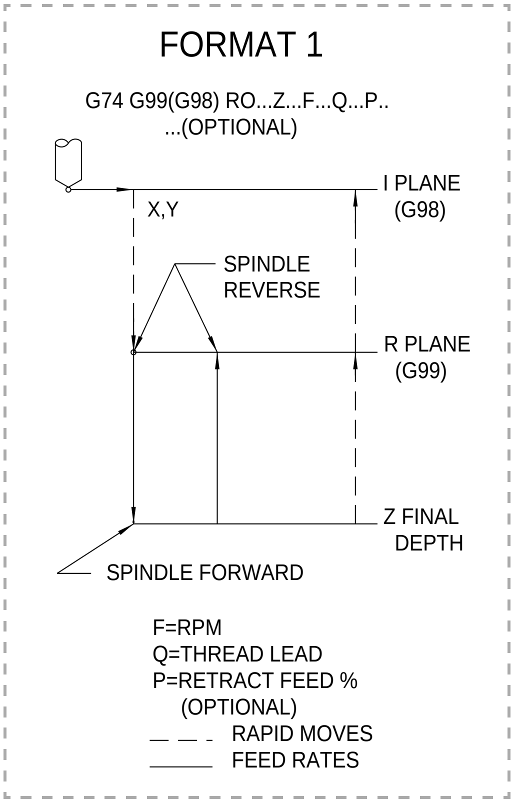


Figure 4-4: Left Hand Tapping (1)

4.8.33 G74 - LEFT HAND TAPPING (FORMAT 2)

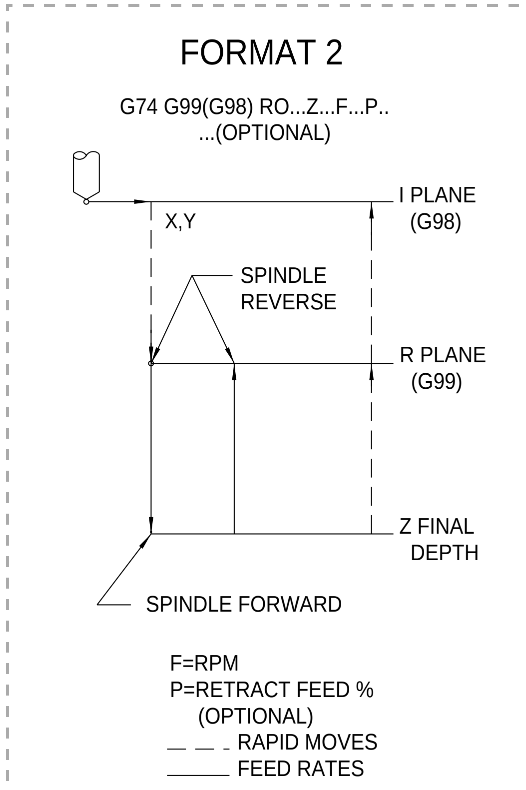


Figure 4-5: Left Hand Tapping (2)

4.8.34 G75 - TAPPING HEAD CYCLE (FORMATS 1 & 2)

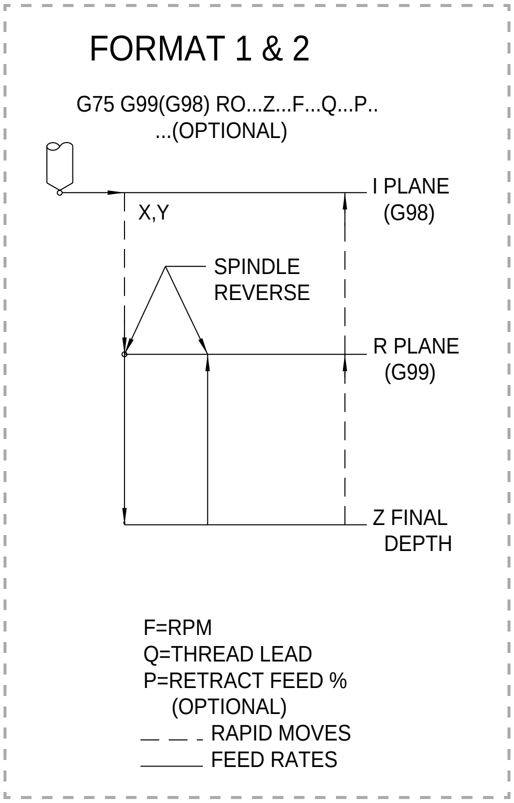


Figure 4-6: Tapping Head Cycle (1&2)

4.8.35 G76 - FINE BORING USING Q

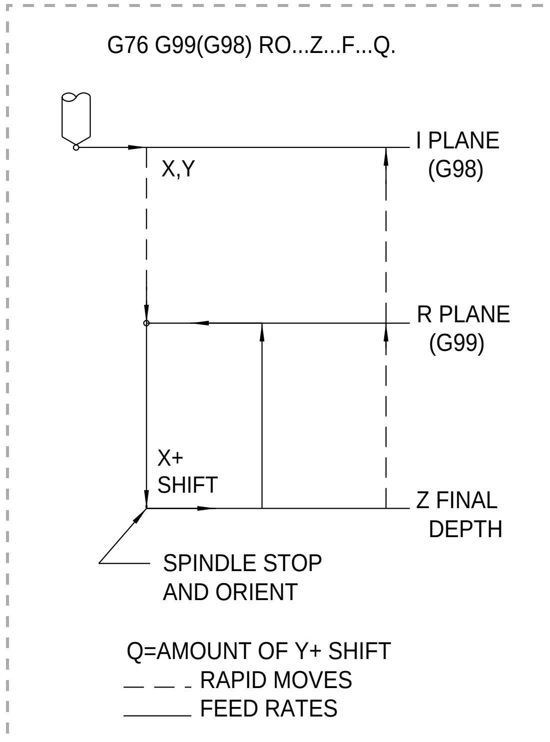


Figure 4-7: Fine Boring Using Q

4.8.36 G76 - FINE BORING USING I, J

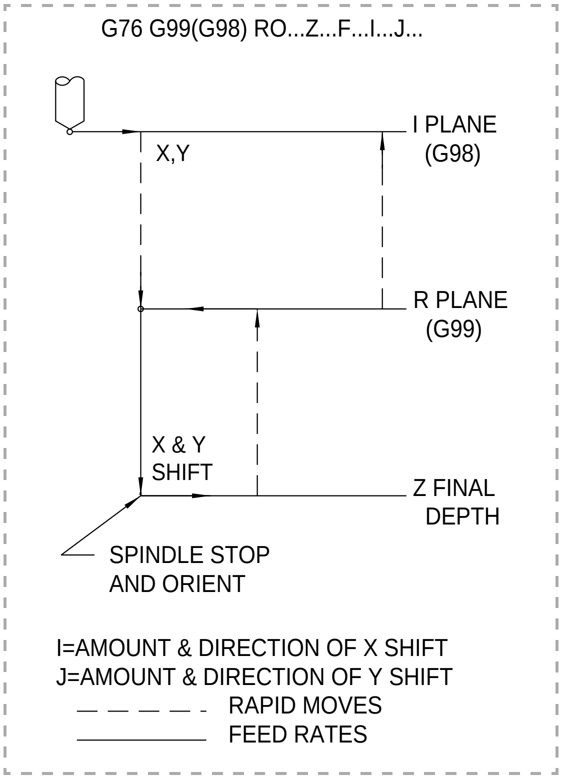


Figure 4-8: Fine Boring Using I, J

4.8.37 G81 - SPOT DRILLING

Use G82 for center drilling.

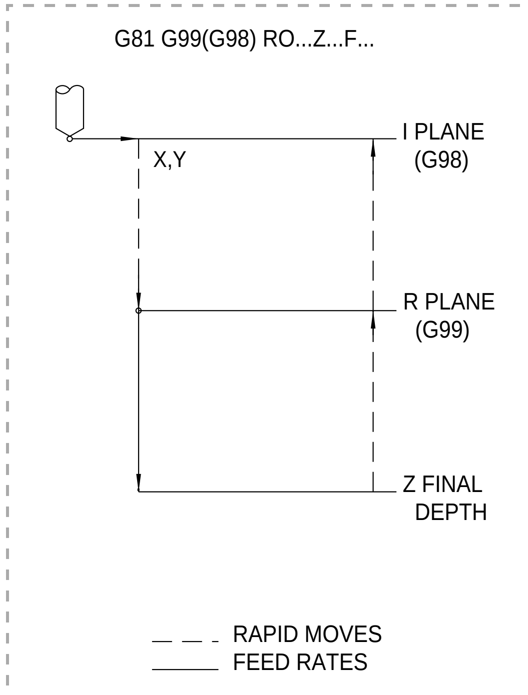


Figure 4-9: Spot Drilling

4.8.38 G82 - COUNTER BORING

Use this for center drilling, counter sinking, and counter boring.

Dwell time for three revolutions is calculated: 180000/RPM=P word value

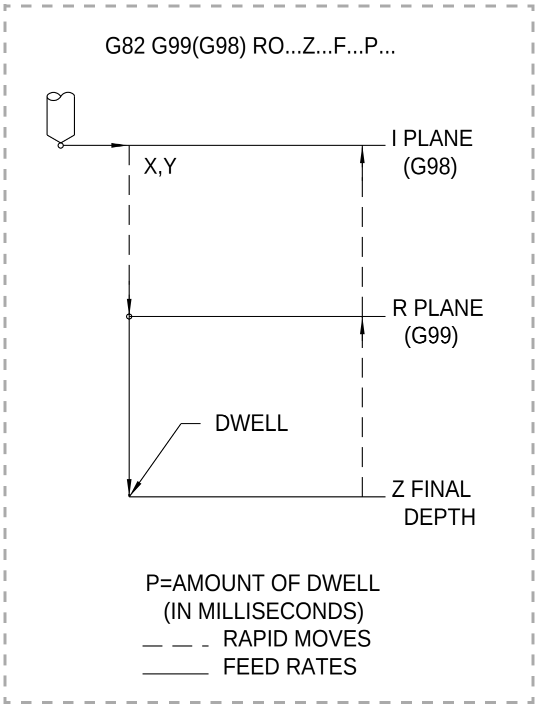


Figure 4-10: Counter Boring

4.8.39 G83 - DEEP HOLE DRILLING USING Q

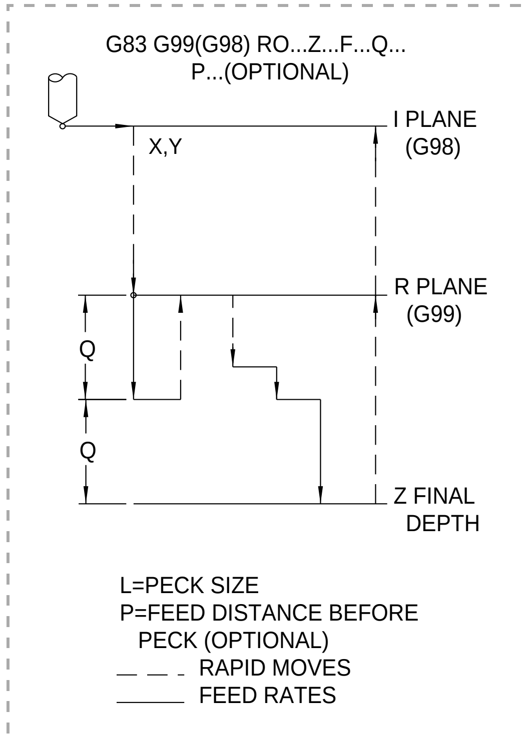


Figure 4-11: Deep Hole Drilling Using Q

4.8.40 G83 - DEEP HOLE DRILLING USING I, J, K

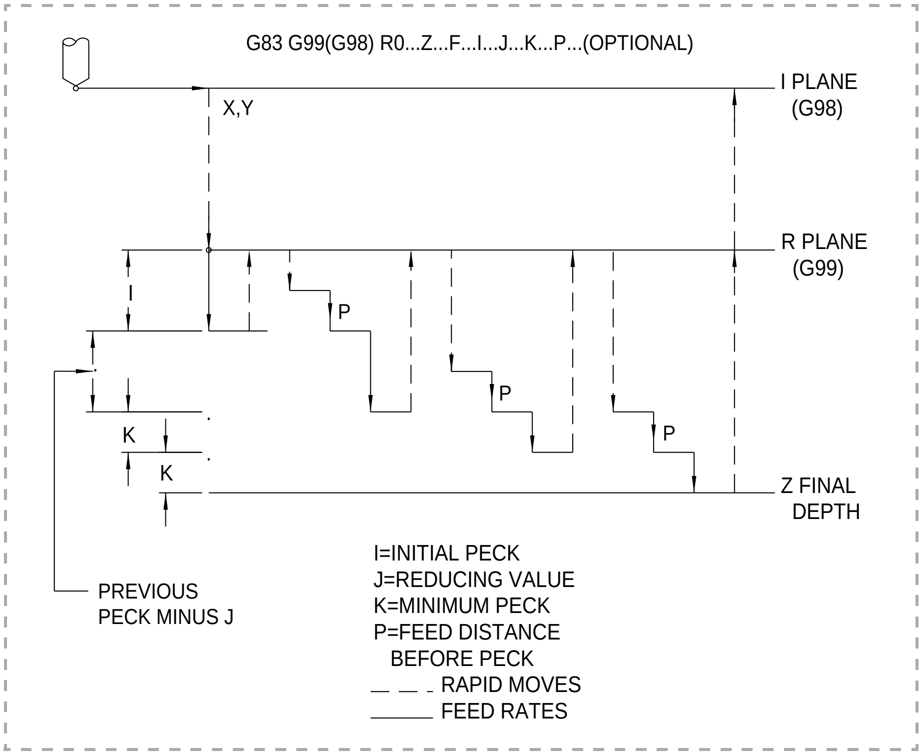


Figure 4-12: Deep Hole Drilling Using I, J, K

4.8.41 G84 - RIGHT HAND TAPPING (FORMAT 1)

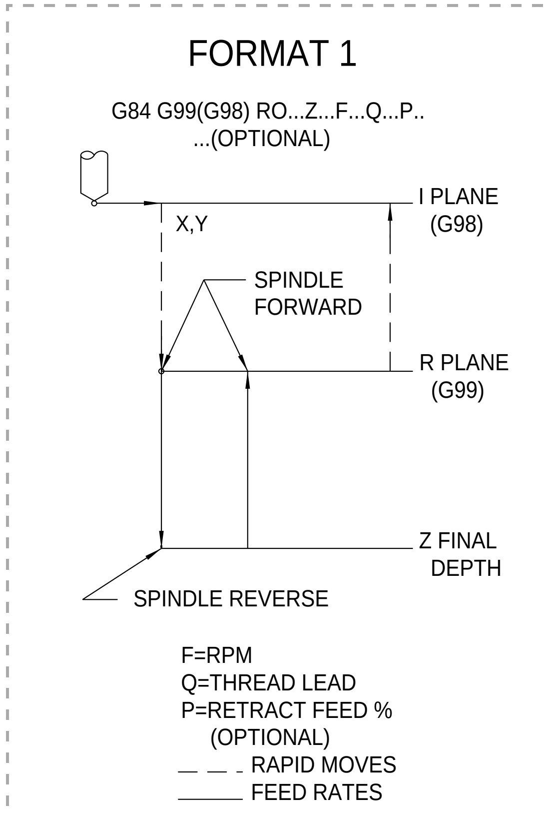


Figure 4-13: Right Hand Tapping (1)

4.8.42 G84 - RIGHT HAND TAPPING (FORMAT 2)

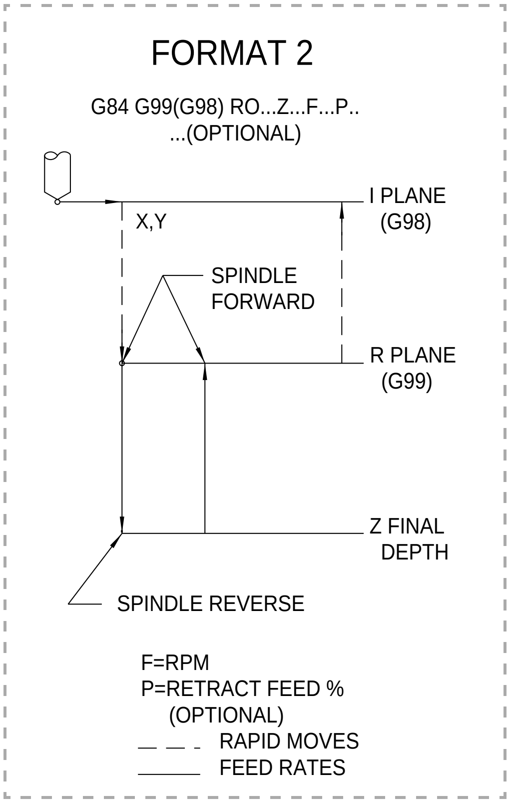


Figure 4-14: Right Hand Tapping (2)

4.8.43 G85 - BORE IN, BORE OUT

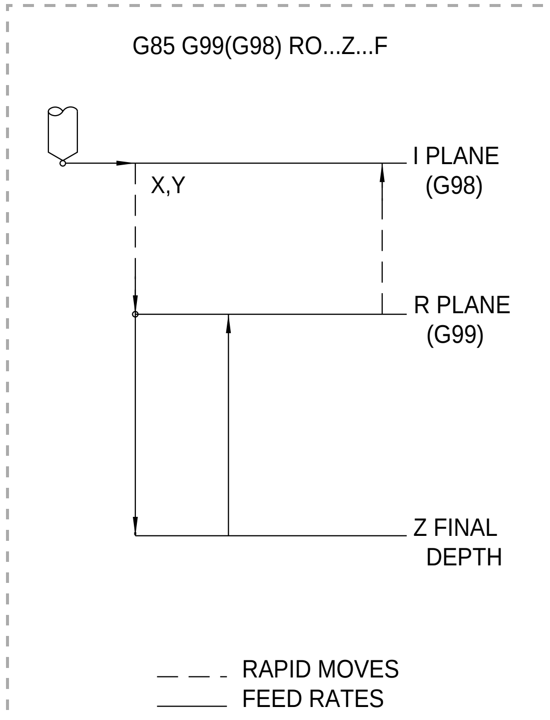


Figure 4-15: G85-Bore In, Bore Out

4.8.44 G86 - BORE IN, SPINDLE OFF, RAPID OUT

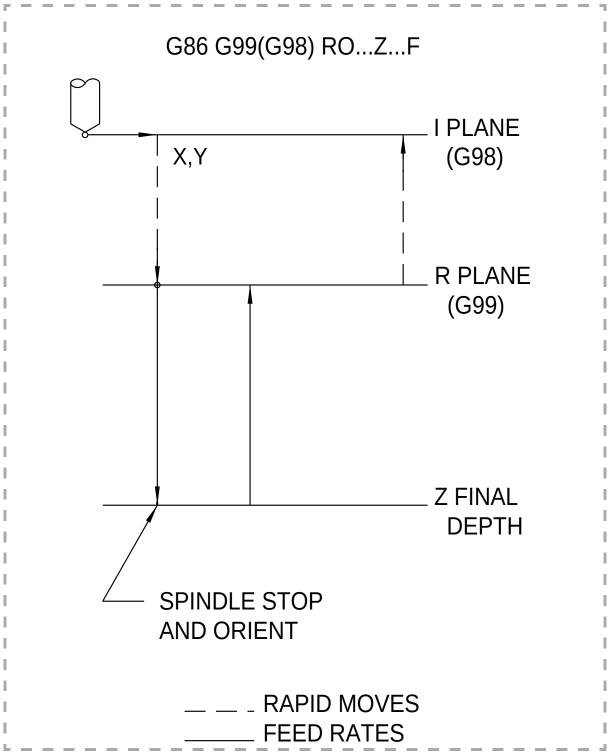


Figure 4-16: G86-Bore In, Spindle Off, Rapid Out

4.8.45 G87 - BORE IN, BORE OUT

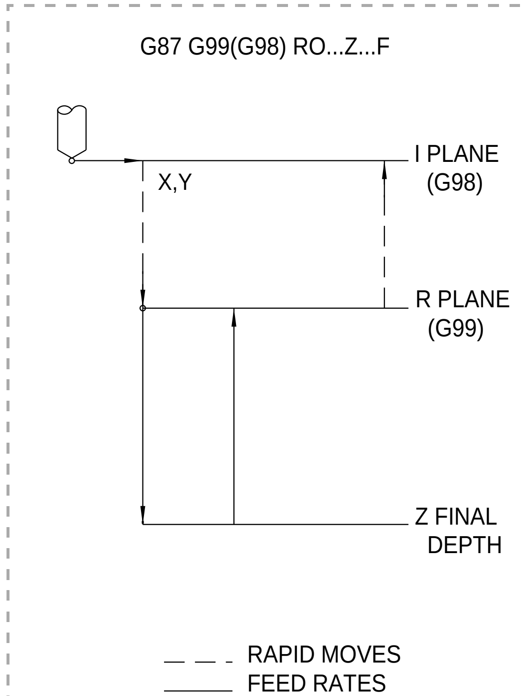


Figure 4-17: G87-Bore In, Bore Out

4.8.46 G88 - BORE IN, DWELL, BORE OUT

Dwell time for three revolutions is calculated: 180000/RPM=P word value

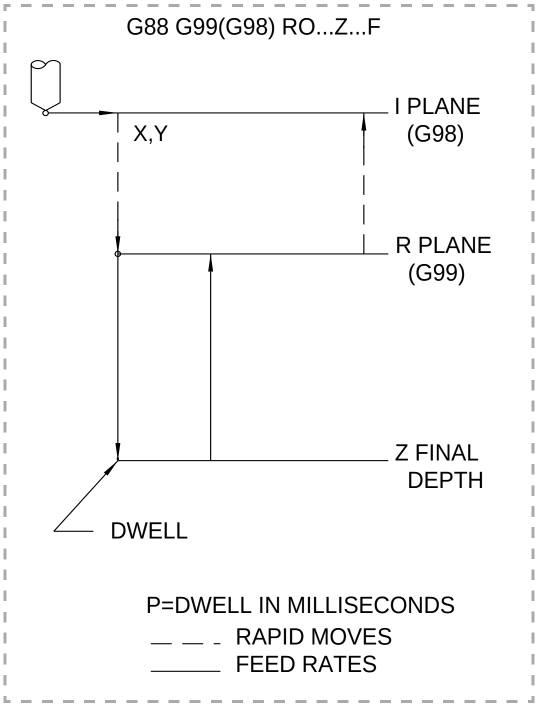


Figure 4-18: G88-Bore In, Dwell, Bore Out

4.8.47 G89 - BORE IN, DWELL, BORE OUT

Dwell time for three revolutions is calculated: 180000/RPM=P word value

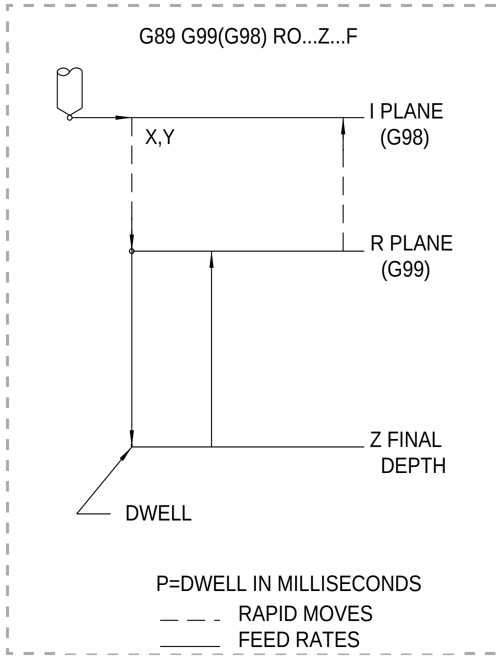


Figure 4-19: G88-Bore In, Dwell, Bore Out

4.9 TAPPING CYCLES

The fixed cycle used for tapping is depends on which tapping attachment is being used. The cycles and examples given here have been approved by the tapping attachment manufacture specified.

4.9.1 GENERAL TAPPING RULES

RAMPING:

Program a G8 (No Ramps) for the tap operation. This allows the tool to feed at a constant rate in and out of the hole.

CLEARANCE:

The rapid plane (R0 Plane) should be at least .4 above the hole to be tapped. This insures that the tap fully retracts prior to movement to the next position.

SPINDLE SPEED:

When using the G74 and G84 cycles, the spindle should be programmed for the high gear range. This will provide better spindle reversal for tapping. This is accomplished by programming a “.2" at the end of the S word or F word. For example, S1000.2 or F1000.2 sets the spindle speed at 1000 rpm in the high gear range.

FEED RATE:

The feed rate is determined by the thread lead and spindle RPM. The thread lead is calculated as 1 divided by the number of threads per inch. When using G74 and G84, the feed rates may be programmed differently for Format 1 and Format 2. When using the G85 cycle for tapping, the feed rate must be calculated by the programmer for Format 1 or Format 2. When using rigid tapping in Format 2 and the F word requires decimal entries, use at least three place accuracy.

4.9.2 FORMAT 1

The feed rate is calculated by the control. These cycles are programmed using the F word for RPM and the Q word for thread lead. G74, G75 and G84 calculate the feed rate as the RPM times the thread lead. These cycles automatically use 100 percent of this calculation for the feed rate into the hole and 105 percent for the feed rate out of the hole. The P word may be used to increase or decrease the feed rate out of the hole. Use a positive P to increase and a negative P to decrease the feed rate out of the hole.

4.9.3 FORMAT 2

The feed rate must be calculated by the programmer. These cycles use the F word as feed rate and do not use the Q word. The F word in the cycle indicates the calculated feed rate.

4.9.4 FEED RATE CALCULATION

The feed rate programmed should use 100 percent of the optimum feed rate. The optimum feed rate is the RPM times the thread lead.

EXAMPLE:

1/4-20 Tap at 2000 RPM
*Feed rate = (1 / threads per inch) * RPM*
*= (1 / 20) * 2000*
*= .05 * 2000*
= 100.

BLIND HOLES:

The procedure described is ONLY a guideline for blind hole tapping. Begin tapping to a depth 2/3 of the desired final Z depth. Gradually increase the programmed depth until the final Z depth is reached. This method reduces the possibility of feeding the tool into the material at the bottom of the hole.

TAP SIZES:

Tapping attachments vary with the desired size of the tap. Insure the proper attachment is used for the tap size. The chart below lists the recommended minimum and maximum size tap for each attachment.

Table 4-2: TAPMATIC

SERIES	MAX RPM	TAP CAPACITY	MAX CUTTING	MAX FORMING
NCR-00	5000	#00 - #4	#4	#3
NCR-0A	3000	#4 - 1/4	1/4	1/4
NCR-1A	2000	#10 - 1/2	1/2	1/2
NCR-2A	1000	3/8 - 3/4	3/4	9/16
SPD-3	2000	#0 - 1/4	1/4	1/4
SPD-5	1500	#6 - 1/2	1/2	1/2
SPD-7	1200	#10 - 3/4	3/4	9/16

Table 4-3: PROCUNIER

SERIES	MAX RPM	TAP CAPACITY	MAX CUTTING	MAX FORMING
15001	3000	#0 - 1/4	1/4	#10

NOTE

The maximum cutting and forming tap sizes for the Tapmatic are based on free machining steel. When tapping aluminum, the tap size should not exceed the tapping attachment maximum size.

PROGRAM CODING:

Program coding varies with the tapping attachment and the machine format. Below are examples of the most common uses and attachments.

METRIC THREADS:

Metric threads may be cut in the inches mode. The Q word is calculated using the thread pitch of the tap. The thread pitch is multiplied by .03937 to convert to inches for the control. For an m8 x 1.25 thread, the Q word is calculated as follows:

EXAMPLE:

Format 1
*Q = 1.25 * .03937*
Q = .0492
H1 Z1.0 M8
G84 G98 R0+.4 Z-1.025 F500.2 Q.0492 X.5 Y-.5

4.9.5 PROGRAM EXAMPLES

The following examples describe the program for a 1/4-20 thread, .3 deep, at 2000 rpm.

EXAMPLE:

Format 1
N52 G0 G90 S2000 M3 E1 X0 Y0
N53 H1 M7 Z.4
N54 G84 G99 R0+.4 Z-.3 F2000 Q.05 X0 Y0
N55 X.5
N56 X1.
N57 G80

4.9.6 RIGID TAPPING

OPTIONAL FEATURE

Rigid tapping may be accomplished for right or left hand threads. The rigid tapping cycles are programmed in the same manner as the compression tap holders. The taps may be held in a collet or solid tool holder.

NOTE
Rigid tapping requires a special drive for operation. The machine must also have a C axis controller for the spindle. In the back panel of the VMC, the spindle driver should be labeled with Horsepower and Rigid Tap. If the VMC only has a Horsepower label, it does not have Rigid Tap. Please refer to the VMC Maintenance Manual for further information.

> **NOTE**
> 10,000 rpm spindle 3,000 rpm max, 7,500 rpm spindle 1,500 rpm max, 15,000 rpm spindle 3,000 rpm max.

The program codes for rigid tapping are G74.1, G74.2, G84.1, and G84.2. The G74.1 and G84.1 are coded the same as G74 and G84. Prior to beginning the tapping cycle, the machine prepares the spindle. This is an automatic process accomplished by four revolutions of the spindle. The preparation is done after the H offset is applied and just before the cycle begins. The use of the G74.2 or G84.2 prepares the cycle for execution during the H offset application.

The machine uses the low range for tapping at 750 rpm and below. When the rpm is above 751 the machine uses the high range. When using the G84.2 or G74.2, and the programmed rpm is above 750, the S#.2 should be used on the initial spindle speed call. This sets the proper belt range for the cycle preparation. The G84.1 and G74.1 automatically set the proper range for preparation. The G74.1 and G84.1 automatically release the spindle orientation lock. The RPM specified, before and after the H word must be the same.

The Gain parameter setting (in SETP) is used to adjust the spindle speed and feed rate correlation. The gain may be set from 0 to 255. Please contact your dealer or refer to the VMC maintenance manual for further information.

The Ramp parameter setting (in SETP) is used to adjust the acceleration time of the spindle. The ramp may be set from 0 to 100. The higher the ramp number, the longer the time for acceleration.

EXAMPLE:

N52 G0 G90 S.2 M5 G80 M90 Do not turn the spindle on!
N53 G84.2
N54 H1 M7 Z.4
N55 G84.1 G99 R0+.4 Z-.3 F2000.2 Q.05 X0 Y0
N56 X1.
N57 G80

EXAMPLE:

N52 G0 G90 S2000.2 M5 G80 M90 Do not turn the spindle on!
N53 G84.2
N54 H1 M7 Z.4
N55 G84.1 G99 R0+.4 Z-.3 S2000.2 F100. X0 Y0
100% feed calculation here

N56 X1.
N57 G80

4.9.7 TAPMATIC NCR SERIES

The tapping heads in this series should use the G85 Bore Cycle. The feed rate and spindle potentiometers should be disabled. The following formula calculates the required feed rate.

M48, M49:

When using the Bore Cycle (G85), the feed rate and spindle speed potentiometers must be disabled. With the potentiometers disabled the machine uses the programmed feed rates. This is accomplished with the M49 code. At the completion of the cycle the potentiometers should be enabled with the M48 code.

*Feedrate = ((1 / threads per inch) * RPM) * 95%*
*= ((1 / 20) * 2000) * .95*
*= (.05 * 2000) * .95*
*= 100. * .95*
= 95.

EXAMPLE: (FORMAT 1 OR FORMAT 2):

```
N1 O1 (TAPMATIC NCR SERIES
N2 G0 G8 G90 S2000 M3 E1 X0 Y0
N3 H1 M7 Z.4
N4 M49
N5 G85 G99 R0+.4 Z-.3 F95. X0 Y0
N6 X1.
N7 G80 M48
N8 M5 M9
```

4.9.8 TAPMATIC SPD SERIES

The tapping heads in this series should use the G75 Tapping Cycle.

EXAMPLE: (FORMAT 1 OR FORMAT 2):

```
N1 O1 (TAPMATIC SPD SERIES
N2 G0 G8 G90 S2000 M3 E1 X0 Y0
N3 H1 M7 Z.4
N4 G75 G99 R0+.4 Z-.3 F2000. Q.05 X0 Y0
N5 X1.
N6 G80
N7 M5 M9
```

4.9.9 PROCUNIER SERIES

The tapping heads in this series should use the G75 Tapping Cycle.

EXAMPLE:

(FORMAT 1 OR FORMAT 2):

N1 O1 (PROCUNIER SERIES
N2 G0 G8 G90 S2000 M3 E1 X0 Y0
N3 H1 M7 Z.4
N4 G75 G99 R0+.4 Z-.3 F2000. Q.05 X0 Y0
N5 X1.
N6 G80
N7 M5 M9

4.9.10 COMPRESSION (NON SELF-REVERSING) TAP HOLDER SERIES

These tapping heads should use the G84 or G74 Tapping cycle. It is best to use these in the high range (S word with a .2). The P word may be used to increase the feed rate of the tap when it reverses, if the F word is used for the RPM and the Q word is used for the lead. The G74 is used in Format 1 style ONLY.

EXAMPLE:

FORMAT 1:

N1 O1 (COMPRESSION TAP HOLDER SERIES
N2 G0 G8 G90 S1000.2 M3 E1 X0 Y0
N3 H1 M7 Z.4
N4 G84 G99 R0+.4 Z-.3 F1000. Q.05 X0 Y0
N5 X1.
N6 G80
N7 M5 M9

EXAMPLE:

FORMAT 2:

The following formula calculates the required feed rate:

*Feed rate = ((1 / threads per inch) * RPM)*
*= ((1 / 20) * 1000)*
*= (.05 * 1000)*
= 50.
N1 O1 (COMPRESSION TAP HOLDER SERIES
N2 G0 G8 G90 S1000.2 M3 E1 X0 Y0
N3 H1 M7 Z.4
N4 G84 G99 R0+.4 Z-.3 S1000.2 F50. X0 Y0 100% feed calculation here
N5 X1.
N6 G80
N7 M5 M9

4.9.11 DEEP HOLE PROGRAMMING USING G73 AND G83 I, J, AND K

The G73 and G83 I, J, K programming options allow the programmer to customize the drill cycle steps up to the final Z depth. This would be used when the steps can be large at the top of the hole and need to be reduced as the tool feeds deeper into the part.

4.9.12 INITIAL PECK: I#

The I word represents the depth of the first peck starting from the R plane (minimum clearance plane). The rule of thumb for this value is 2 to 3 times the diameter of the drill but may be larger or smaller as required. This value is always an incremental value.

4.9.13 REDUCING FACTOR: J#

The J word represents a reducing value. The J value will be subtracted from the I value for the second peck and from each subsequent peck. Each peck will be smaller than the last peck by the amount of the J value until the peck size is equal to the K value. This value is always an incremental value.

4.9.14 MINIMUM PECK: K#

The K word represents the minimum peck value. The I value will be reduced by the J value until the K value is reached. The remainder of the hole will be drilled in a series of pecks each equal to the K value.

4.9.15 FEED DISTANCE BEFORE NEXT PECK: P#

The P word represents the distance to feed before the next peck. Without the P, the tool will rapid all the way to the next peck. The P will specify how far above he next peck to begin feeding into the material. This parameter is especially helpful if chips fall into the hole during a retraction move. The tool will not crash into the chips if the P value is large enough.

4.9.16 PECK DRILL: G73

The G73 peck drilling cycle is a chip breaker cycle that does not retract all of the way out of the hole. The retract distance is set by the CNC and cannot be changed by the programmer. It is an excellent cycle for free machining materials and drill geometries that allow for efficient chip removal. In these cases there is a time savings in the cycle because the drill does not retract out of the hole.

4.9.17 DEEP HOLE DRILL: G83

The G83 Peck Drilling Cycle retracts to the R plane (minimum clearance plane) on the completion of each peck. It is an excellent cycle to break and clear chips and for drill geometries that are prone to packing chips.

EXAMPLE: This example is for a 1/4 dia drill drilling 1 1/4 inches deep.

EXAMPLE: G73:

G73 X0 Y0 Z-1.25 R+.1 I.5 J.2 K.1 F10. G98 (G99)

5.0 SUBROUTINES & SUBPROGRAMS

5.1 SUBROUTINES

Subroutines are used for contours, hole patterns, or any actions that repeat or are used in many locations. Typically subroutines will contain only positional moves. Feed rates, tool changes, spindle speeds, rotation, and other codes are reserved for the main program. However, most codes can be in a subroutine.

All user defined subroutines must be at the beginning of the program before the main section of the program. Only the O word and comments may be used prior to the first subroutine. **Subroutines cannot be defined in a subprogram.** However, if the START macro command is used, subroutines can be defined in subprograms (see section 18.0 MACROS, MAN-0131).

5.2 BEGINNING A SUBROUTINE

The format of the L word for a subroutine definition is LNNKK.

NN is the subroutine number (01-89).

KK will always be 00 (zero, zero).

EXAMPLE:

L0100 (or L100) This would define the beginning of subroutine number 1
L2300 This would define the beginning of subroutine number 23

The maximum number allowed for NN is 89. Subprograms 90 - 99 are used by the control for Fixed subroutines (see section 6.0 FIXED SUBROUTINES, MAN-0131). The line with the L word that defines the subroutine can only have a parenthesis or an asterisk for a comment. No other codes are permitted.

5.3 CALLING A SUBROUTINE

The format for the subroutine call is LNNKK. NN is the subroutine number (01-99). KK is the number of repetitions (01-99).

EXAMPLE:

L101 This would call, or use, subroutine number 1, one time
L2315 This would call, or use, subroutine number 23, fifteen times

The LNNKK word must be the only word in the block in which it appears with the exception of R parameter definitions, G66, and a parenthesis or an asterisk for a comment. After a subroutine has been executed, it will return to the line where it was called and the program will continue from that line.

5.4 ENDING A SUBROUTINE

A subroutine ends with the L word that starts the next subroutine or with an M17. The M17 must be the only word in the block in which it appears.

The last subroutine in the program MUST have an M17 coded at the end.

5.5 MAIN PROGRAM

An M30 marks the end of the subroutine section and the start of the main program. An M17 marks the end of the last subprogram, which must be on a line before an M30. The M30 must be the only word in the block in which it appears.

When the operator presses the auto button the control will process the program. The control will recognize the existence of subroutines by the L #00 at the beginning of the program. The control will then recognize the beginning of the main program by the M30 code. When the control is ready to run, the line just after the M30 will be the first line to appear on the screen of the pendent.

When the M30 is used to end the subroutine section, an M2 is used to end the program. At the end of the program, the M2 will cause the program to begin again at the line after the M30 code.

EXAMPLE:

N10 M17 This marks the end a sub
N11 M30
N12 G0 G90 S2000 M3 E1 X0 Y0 End of subroutine section
N13 H1 M7 Z.1 Beginning of the Main program
N14 L201 Sub #2 one time. When sub #2 is complete it will return here
N15 M5 M9 G80

5.6 NESTING

A subroutine may be called for execution from another subroutine. This is called subroutine “nesting.”

Subroutines may be nested as many as seven deep. This means that at some point in a subroutine another subroutine can be called, and then from that subroutine another can be called and so forth up to seven times.

SUBROUTINES CANNOT BE DEFINED IN A SUBPROGRAM. However if the START marco command is used, subroutines can be defined in subprograms (see section 18.0 MACROS, MAN-0131).

EXAMPLE:

N1 O1234 (SUBROUTINE EXAMPLE PROGRAM
N2 L100 This marks the beginning of sub #1
N3 X.5 Y.5
N4 X-.5
N5 G80
N6 M17 This marks the end a sub
N7 L200 This marks the beginning of sub #2
N8 G81 G99 R0+.1 Z-.1 F35.
N9 L101 Sub#1 is being called from sub #2. Sub #1 is nested inside sub#2
N10 M17 This marks the end a subroutine

N11 M30 End of subroutine section
N12 G0 G90 S2000 M3 E1 X0 Y0 Beginning of the Main program
N13 H1 M7 Z.1
N14 L201 Sub #2 one time. When sub #2 is complete it will return here
N15 M5 M9 G80
N16 G0 G49 G90 Z0
N17 M2 End of the Main program. In the auto mode the program will rerun from line N12

EXAMPLE: *Drill and Tap 2 holes using subroutine to define positions.*
N1 O1 (SAMPLE PROGRAM
N2 L100 Define Subroutine 1
N3 X.5 Y.5
N4 X-.5
N5 G80
N6 M17 End Subroutine
N7 M30 End of Subroutine definition
N8 M6 T1
N9 Tool #1 drill
N10 G0 G90 S3500 M3 E1 X0 Y0
N11 H1 M7 Z.25 Start main program
Drill cycle
N12 G81 G99 R0+.1 Z-.475 F20.
N13 L101 Call Subroutine 1
N14 N15 M6 T2 Tool #2 tap
N16 G0 G90 S600 M3 E1 X0 Y0
N17 H2 M7 Z.25
N18 G84 G98 R0+.1 Z-.5 F600. Q.05 Tap Cycle
N19 L101 Call Subroutine
N20 M5 M9
N21 G0 G49 G90 Z0
N22 E0 X0 Y0
N23 M6 T1
N24 M2

- Block N2 uses the L word to identify the beginning of the subroutine.
- Block N3 through N4 identify the X and Y locations.
- Block N6 uses M17 to define the end of the subroutine.
- Block N7 uses M30 to define the end of subroutine definition and the beginning of the main program.

Upon execution of the program, the CNC always begins processing from the first block. When the first block contains the L word, the CNC examines each following block, until the M17, M30 codes are encountered. The execution begins with the block following the M30.

The example program begins execution from block N8. Block N13 causes program execution from block N2 until the M17 is encountered at block N6.

After completing the subroutine call, the program execution is returned to the next block following the subroutine call (N14).

5.7 PARAMETRIC PROGRAMMING

Generalized subroutines can be written with the use of subroutine parameters. In a generalized subroutine, the numerical value of the A, B, E, F, G, H, I, J, K, L, M, P, Q, R, S, T, X, Y, Z words need not be specified directly. Values that are to be determined at the time of the subroutine call are specified indirectly by the use of the parametric reference "R". There are ten parameters, R0 through R9.

X+R1 directs the CNC to take the current value of parameter R1 as the value for the X word. X-R1 directs the CNC to take the negative of the current value of parameter R1 as the value for the X word.

The values of the parameters are modal. They are modified by programming an R word in a line of code. For example:

R0+.137 defines the value of parameter R0 to +.137. This value is used by any R0 in the program until it is redefined.

In the example below, subroutine L100 is a generalized subroutine to create a “D” pattern. Block N7 of the example calls the subroutine with the parameters R0 and R1 set to 2.0 and 1.0 respectively.

EXAMPLE:

```
N1 L100  (DEFINE SUBROUTINE 1
N2 G1 Y+R0  (FIRST LEG OF “D” PATTERN
N3 G2 X+R1 Y-R1 J-R1
N4 G2 X-R1 Y-R1 I-R1  (BACK TO BEGINNING
N5 M17
N6 M30  (END OF SUBROUTINE DEFINITION
N7 L101 R0+2. R1+1.  (CALL SUBROUTINE 1, (EXECUTE 1 TIME
```

All R values are modal, and are not cleared at the beginning of a program. The values are cleared at power on and are zero until they are defined.

5.8 INDEFINITE SUBROUTINE REPETITIONS

In some cases a subroutine needs to be repeated an indefinite number of times. This is accomplished by using a .1 extension at the end of a subroutine call.

N1 L100
N2 E1 X.45 Y-1.05
N3 G81 G99 R0+.1 Z-.75 F80. X.5 Y-1.
N4 X2.5
N5 G80
N6 Z1.
N7 E0 X0 Y0
N8 G4 P66000 The machine is in the Waiting state, spindle & coolant on
N9 M17
N10 M30
N11 G90 G0 S10000 E1 X.45 Y-1.05
N12 H1 Z1. M7
N13 L101.1 The .1 extension repeats sub #1 an indefinite number of times
N14 M2

5.9 SUBPROGRAMS

Subprograms function for the same purpose as a subroutine. They can be used instead of subroutines. For program editing purposes a program that uses subroutines is easier to edit. The operator can edit both the main and subroutine section of the program without switching to another program. Editing a subprogram requires that the operator first switch to the subprogram, edit, then switch back to the main program. The main program must be currently active to execute the program.

Subprograms are generally used in Format 2 style programs. A main program is identified by the use of the M30 code at the end, which functions like the M2 would in a Format 1 style program. The subprogram is identified by using the M99 code at the end. The M99 functions as the point to either return to the beginning of the subprogram for a repeat, or to return to the line where the subprogram was called.

SUBPROGRAMS CANNOT CONTAIN SUBROUTINES. However a subprogram can be called from a subroutine. If START macro command is used, subroutines can be defined in subprograms (see section 18.0 MACROS, MAN-0131).

EXAMPLE:

O1 (MAIN PROGRAM
G90 G0 G17 G80 G40 G49 Z0
M6 T1 (TOOL #1
S2000 M3 G54 X0 Y0
H1 D1 Z1. M8
G82 G99 R0+.1 Z-.25 F45.

M98 P2 L1 Call subprogram #2 one time. This is where the sub returns after execution
M5 M9
G80
G90 G0 G49 Z0
M6 T2 (TOOL #2
G90 G0 S2000 M3 G54 X0 Y0
H2 D2 Z1. M8
G83 G99 R0+.1 Z-2.1 F37. Q.3143

M98 P2 L1 Call subprogram #2 one time. This is where the sub returns to after execution
M5 M9
G80
G0 G90 G49 Z0
G59 X0 Y0
M30
O2 (SUBPROGRAM FOR HOLE LOCATIONS
X2. Y1.
X3. Y1.
M99 End of subprogram and return to main program

6.0 FIXED SUBROUTINES

6.1 DEFINITION

Fixed Subroutines are dedicated cycles, standard in the memory of the control. They are called by the use of an L word (L9101 - L9901) and will use parameters (R0-R4, Z, & F). Fixed subroutines can be used during programs run from memory or DNC operations. Subroutine calls are not allowed in MDI.

6.1.1 L9101 PROBE FUNCTIONS

See L9101 Applications, section 15.0 TOUCH PROBE, MAN-0131.

6.2 ENGRAVING

6.2.1 L9201 ENGRAVING FUNCTIONS

Engraving functions are invoked by using the L9201 code. The L9201 function is capable of engraving either a constant text/number string or a serialized text/number string. All lettering is uppercase.

6.2.2 PARAMETERS

1. A tool must have been specified by an H or D word.
2. The Z word defines the final depth of cut for the cycle. The maximum Z depth from R0 plane to final is 2.5 inches.
3. The R0 word is used to define the clearance plane for the tool to move above the part. The tool retracts to this plane when moving between characters, changing position to continue the same character, or after the last character is engraved.
4. R1 defines mode selection of four options:
   a. A value of 0 for standard Gothic font (R1+0.).
   b. A value of 1 for stencil type Gothic font (R1+1.).
   c. A value of 2 for serialization standard font (R1+2.).
   d. A value of 3 for serialization stencil font (R1+3.).
5. R2 represents the height of characters to be engraved minus the tool diameter. The maximum character height is 2.5 inches.
6. R3 represents the angle at which the characters are to be engraved.(see figure
7. R4 is the serialization increment selector (1-9). Enter the R4 value for the increment amount. R4+1. increments the number by one for each part. R1 must be R1+2 or R1+3 to allow R4 to function.
8. F is the feed rate.
9. The comment or words typed after the ((left parenthesis) will be engraved.

6.2.3 RESTRICTIONS

1. The maximum number of characters (including spaces) that can be engraved in each use of the L9201 code is 63 minus the number of characters used to code the first part of the L9201 line itself.
2. When serializing, the last part of the text/number string must be the number to increment.

6.2.4 ENGRAVING A CONSTANT STRING

The procedure to engrave the word “ENGRAVE” .125 inch high, using a .015 inch diameter engraving tool, is shown below:

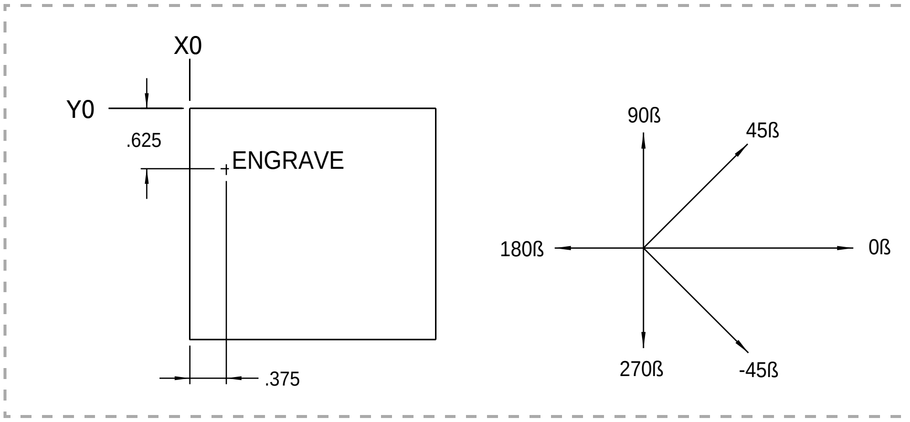


Figure 6-1: Engraving a Constant String

1. Position the X and Y to the start position (see 6.2.12 Computing Start Position, 6.0 FIXED SUBROUTINES, MAN-0131).

> NOTE
> This is true for engraving at an angle of zero degrees (see figure above). When engraving on a non-zero angle the X and Y axis must be shifted properly for the angle programmed.

2. Position the Z axis to the Initial Plane.
3. Set up the engraving cycle.

EXAMPLE:

```
N1 O1 (SAMPLE ENGRAVING PROGRAM
N2 M6 T1
N3 (TOOL #1 ENGRAVING TOOL .015 CENTER DRILL
N4 G0 G90 S10000 M3 E1 X.375 Y-.6175
N5 H1 D1 M8 Z.05 (DIAMETER .015 IN OFFSET PAGE
N6 L9201 R0+.05 R1+0 R2+.125 R3+0 Z-.005 F40. (ENGRAVE
N7 M5 M9
N8 G90 G0 H0 Z0
N9 E0 X0 Y0
N10 M2
```

6.2.5 ENGRAVING A SERIALIZED STRING

Serialization is used to engrave numbers on a series of parts while changing the number on each part. The R1 value must be either a 2 or 3 to identify the serialized engraving style. The R4 value must be used to identify the increment amount. This amount is in whole numbers only (to increment decimal numbers, the decimal point must be engraved as a separate character). The numbers to be serialized MUST be at the end of the engraving characters.

When serialization is coded, the machine changes the program code to reflect the next number to be engraved. The program code is changed after the control processes the engraving line. To reset the number, the operator must manually change the program code.

> NOTE
> When the same number is to be engraved on each part, DO NOT use serialized engraving.

EXAMPLE:

```
N1 O1 (SAMPLE ENGRAVING PROGRAM
N2 M6 T1
N3 (TOOL #1 ENGRAVING TOOL
N4 G0 G90 S10000 M3 E1 X.375 Y-.6175
N5 H1 D1 M8 Z.05
N6 L9201 R0+.05 R1+2. R2+.125 R3+0 R4+1. Z-.005 F40. (ENGRAVE 1
N7 M5 M9
N8 G90 G0 H0 Z0
N9 E0 X0 Y0
N10 M2
```

6.2.6 SERIALIZATION RANGE

Serialized engraving uses the number of digits at the end of the initial engraving string to set the range of serialization. For instance, if the end of the string is 5 digits (00001), the maximum number engraved will be 99999, and the number engraved after 99999 will be 00000.

The preceding example engraves ENGRAVE 1 on the first part and ENGRAVE 2 on the second and so on. This example will engrave the numbers 1, 2, 3, 4, 5, 6, 7, 8, 9, and 0. After the 0 the number 1 is engraved again. Further, the maximum number of digits that could be coded in this example is five (00000 through 99999), because 58 characters on the L9201 line are used by program code, including the spaces, right parenthesis, and constant portion of the engraving text.

EXAMPLE: ENGRAVE 01 serializes the numbers 01 through 99, then 00, and then restarts at 01.

ENGRAVE 0001 serializes the numbers 0001 through 9999, then 0000, and then restarts at 0001.

6.2.7 SPACING, POSITIONING, & TOOL PATH CALCULATIONS

The L9201 Fixed Subroutine uses the Letter Box (the rectangular area around each character) to calculate the tool path for the engraving, beginning at the X, Y position specified in the program.

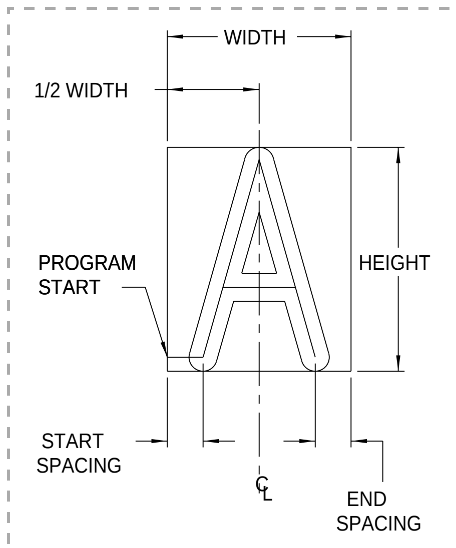


Figure 6-2: Letter Box

6.2.8 COMPUTING ACTUAL LETTER HEIGHT FROM THE R2 VARIABLE

The L9201 Fixed Subroutine calculates the actual height of the tool path as follows:

Actual height = (desired letter height) - (tool diameter)

6.2.9 COMPUTING START SPACING

The L9201 Fixed Subroutine calculates the start spacing for each character as follows:

Start spacing = (actual height) * (start factor)

6.2.10 COMPUTING END SPACING

The L9201 Fixed Subroutine calculates the end spacing for each character as follows:

End spacing = (actual height) * (end factor)

6.2.11 COMPUTING START POSITION

X: The L9201 Fixed Subroutine will move to the X position specified in the program, and then move in X the amount calculated for the start spacing of the first character, before beginning the engraving cut.

Y: The program must specify a starting Y position that takes into account the tool diameter, so that the tool is initially positioned one-half the tool diameter above the desired baseline of the engraving.

X and Y should be specified together before each L9201 line.

6.2.12 COMPUTING ENGRAVING LENGTH

Occasionally, it is necessary to calculate the length of the engraving. The procedure for calculating the length is as follows:

1. Establish a desired height for the engraving.
2. Choose the tool diameter. A recommended value is a diameter equal to .14 times the desired height. Remember to place this diameter in the offset page.
3. Calculate the width factor for the engraving, using the Letter Width Table to sum the width for each character or space.
4. Calculate the total engraving length using the formula:

Engraving length = (actual height) * (total width factor)

EXAMPLE:

Calculate the length of the word “ENGRAVE” to be engraved at a height of .125 inches with a tool diameter of .015 inches.

1. Tool diameter = .015 inches
2. Actual height = (desired letter height) - (tool diameter) = .125 - .015 inches = .110 inches
3. Total width factor = the sum of the Letter Width Table’s width factors for each character or space.

   E(.8853)+N(.9573)+G(.9588)+R(.9749)+A(1.0506)+V(1.0147)+E(.8853)=6.7269

4. The actual engraving length is now given by:

   Engraving length = (actual height) * (width factor)
   or, in this example, engraving length = (.110) * (6.7269) = .73996

> NOTE
> The tool diameter must be entered in the offset page, and an H word and/or D word must be used in the program prior to calling the L9201.

6.2.13 LETTER WIDTH FACTOR VALUES

The letter width is based on the box space for each letter. The Start and End factors are used to calculate the distance from the edge of the letter box to the start and end points of the letter.

Table 6-1: Letter Width Factor Table

CHARACTER	WIDTH	START	END
A	1.0506	.2279	.2279
B	.9455	.2279	.1284
C	.9471	.1837	.2677
D	.9441	.2279	.1927
E	.8853	.2279	.2280
F	.9118	.2279	.2294
G	.9588	.2153	.2282
H	.9706	.2279	.2353
I	.4559	.2279	.2280
J	.95	.2279	.2280
K	1.0249	.2279	.2278
L	.8941	.2279	.2280
M	1.0824	.2279	.2280
N	.9573	.2279	.2279
O	.9647	.1779	.1779
P	.9485	.2279	.2279
Q	.9647	.1779	.1779
R	.9749	.2279	.2278
S	.9853	.2279	.2280
T	.9485	.2279	.2279
U	1.0000	.2279	.2280
V	1.0147	.2279	.2280
W	1.2059	.2279	.2279
X	.9559	.2279	.2280
Y	1.0441	.2279	.2280
Z	.9441	.2279	.2280

The letter width is based on the box space for each letter. The Start and End factors are used to calculate the distance from the edge of the letter box to the start and end points of the letter.

Table 6-2: Letter Width Factor Table

CHARACTER	WIDTH	START	END
\	.9559	.2279	.2280
!	.7353	.3676	.3677
#	1.0441	.1926	.1927
$	.9559	.2276	.2280
%	.8676	.2279	.2280
‘	.7353	.3676	.3677
&	.9853	.2345	.1453
(	.5855	.2279	.2279
)	.5855	.2279	.2279
*	.9559	.2279	.2280
	1.1029	.2279	.2280
,	.7353	.2941	.2960
-	1.1029	.2279	.2280
.	.7353	.3676	.3677
/	.9559	.2279	.2280
0	.9647	.1779	.1779
1	.6059	.2279	.2280
2	.8926	.2279	.2271
3	.9632	.2279	.2281
4	1.0779	.2279	.2279
5	.9485	.2279	.2089
6	.9118	.2153	.2267
7	.9691	.2279	.2279
8	.9706	.2271	.2282
9	.9118	.2259	.2123
:	.7353	.3676	.3677
;	.7353	.2941	.2957
“	.7704	.2274	.2274
=	1.1029	.2279	.2280
?	.9853	.2279	.2280
Space	.8823		

6.3 BOLT CIRCLE

6.3.1 L93NN BOLT CIRCLE

1. R0 represents the I definition of a circle. This is **the X direction and distance** from the starting position to the center.
2. R1 represents the J definition of a circle. This is **the Y direction and distance** from the starting position to the center.
3. R2 represents **the angular step between holes.** A positive angular step will move CCW around the bolt circle, while a negative angular step will move CW around the bolt circle.
4. NN is the number of holes to be drilled. For example, L9304 is for 4 holes.

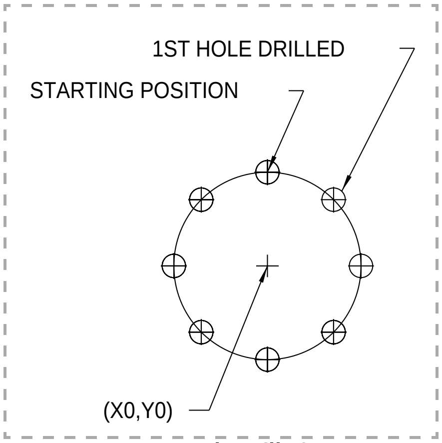


Figure 6-3: 1st Hole Drilled

The drawing above is a 3.0" diameter, 8 hole, evenly spaced bolt circle. The procedure is as follows:

1. Position the X,Y axes to the starting position (the last hole of the bolt circle to be drilled) and the Z axis to the I plane.
2. Select desired Fixed Cycle.
3. Start Bolt Circle subroutine.

EXAMPLE: (FORMAT 1):

N1 O1 (BOLT HOLE EXAMPLE
N2 G0 G90 S2000 M3 X0 Y1.5 Position to starting position
N3 H1 M7 Z.1
N4 G81 G99 R0+.1 Z-1.0 F10. Set up Fixed Cycle
N5 L9308 R0+0 R1-1.5 R2-45. Call Bolt Circle Subroutine
N6 M5 M9
N7 G80

6.4 MILL BORING

6.4.1 L94NN MILL BORING CYCLE COUNTERCLOCKWISE

1. A tool must have been specified by an H or D word and the tool diameter MUST be in the tool table.
2. R0 represents the feed rate.
3. R1 represents the diameter of hole to be bored.
4. NN represents the number of repetitions desired.

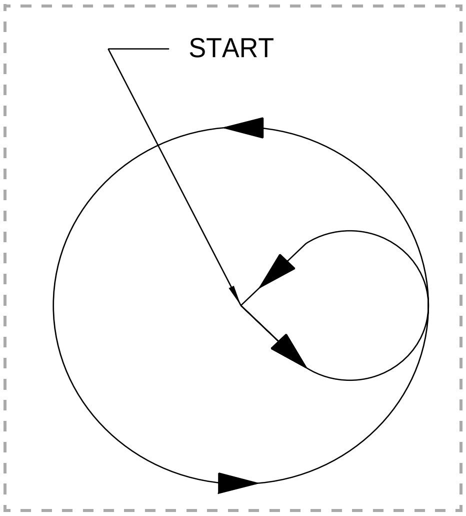


Figure 6-4: Boring Cycle Counter Clockwise

The drawing above is a 1.5 diameter hole to be mill bored. The procedure is as follows:

1. Position the X,Y axes to the center.
2. Position the Z axis to finished depth.
3. Start Mill Boring Cycle.

EXAMPLE:

```
N1 O1 (MILL BORING CCW EXAMPLE
N2 M6 T1
N3 G0 G90 S2000 M3 X1.0 Y-1.0
N4 H1 D1 M7 Z.1
N5 G1 F10. Z-1.
N6 L9401 R0+10. R1+1.5
N7 M5 M9
N8 G0 H0 G90 Z0
```

6.4.2 L95NN MILL BORING CYCLE CLOCKWISE

1. A tool must have been specified by an H or D word and the tool diameter MUST be in the tool table.
2. R0 represents the feed rate.
3. R1 represents the diameter of hole to be bored.
4. NN represents the number of repetitions desired.

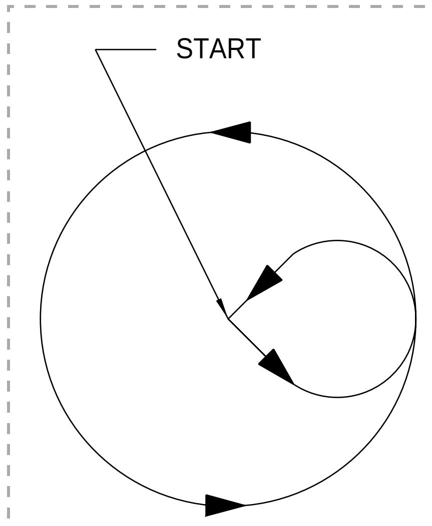


Figure 6-5: Hole Diameter

The drawing above is a 1.5 diameter hole to be mill bored. The procedure is as follows:

1. Position the X,Y axes to the center.
2. Position the Z axis to finished depth, selecting the proper H word or D word to specify the diameter the tool is using.
3. Start Mill Boring Cycle.

EXAMPLE:

N1 O1 (MILL BORING CW EXAMPLE
N2 M6 T1
N3 G0 G90 S2000 M3 X1.0 Y-1.0
N4 H1 D1 M7 Z.1
N5 G1 F10. Z-1.
N6 L9501 R0+10. R1+1.5
N7 M5 M9
N8 G0 G49 G90 Z0

6.5 RECTANGULAR POCKET CLEAN-OUT

6.5.1 L9601 RECTANGULAR POCKET CLEAN-OUT COUNTERCLOCKWISE

1. A tool must have been specified by an H or D word and the tool diameter MUST be in the tool table.
2. R0 represents the feed rate.
3. R1 represents the radius on the corner of the tool.

> **NOTE**
> This can be used to regulate the step over distance. The larger this number, the less the amount of step over.

4. R2 represents the overall X dimension.
5. R3 represents the overall Y dimension.

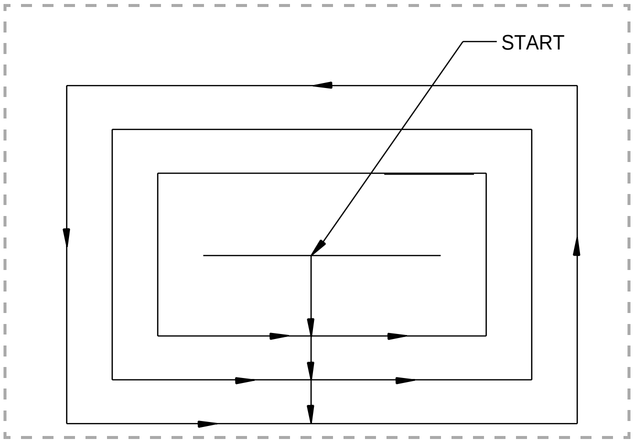


Figure 6-6: Counterclockwise Rectangular Pocket Clean-Out

The tool path drawing above shows how a 3.25 by 1.75 rectangular pocket would be cleaned out with a .25" diameter tool. The procedure is as follows:

1. Position the X,Y axes to the center of the pocket.
2. Position the Z axis to finished depth, selecting the proper H or D word to specify the diameter the tool is using.
3. Start Rectangular Pocket Subroutine.

EXAMPLE:

N1 O1 (RECT CLEANOUT CCW EXAMPLE
N2 M6 T1
N3 G0 G90 S2000 M3 X1.0 Y1.0
N4 H1 D1 M7 Z.1
N5 G1 F10. Z-1.
N6 L9601 R0+10. R1+.01 R2+3.25 R3+1.75
N7 M5 M9
N6 G0 G49 G90 Z0

6.5.2 L9701 RECTANGULAR POCKET CLEAN-OUT CLOCKWISE

1. A tool must have been specified by an H or D word and the tool diameter MUST be in the tool table.
2. R0 represents the feed rate.
3. R1 represents the radius on the corner of the tool.

> NOTE
> This can be used to regulate the step over distance. The larger this number, the less the amount of step over.

4. R2 represents the overall X dimension.
5. R3 represents the overall Y dimension.

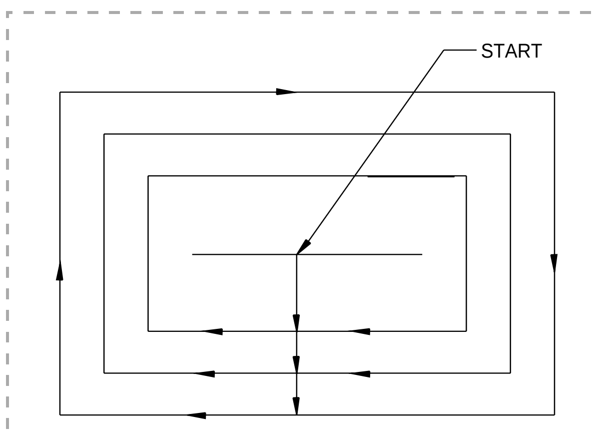


Figure 6-7: Clockwise Rectangular Pocket Clean-Out

The tool path drawing above shows how a 3.25 by 1.75 rectangular pocket would be cleaned out with a .25" diameter tool. The procedure is as follows:

1. Position the X,Y axes to the center of the pocket.
2. Position the Z axis to finished depth, selecting the proper H or D word to specify the diameter the tool is using.

3. Start Rectangular Pocket Subroutine.

EXAMPLE:

N1O1 (RECT CLEANOUT CW EXAMPLE
N2 M6 T1
N3 G0 G90 S2000 M3 X1.0 Y1.0
N4 H1 D1 M7 Z .1
N5 G1 F10. Z-1.
N6 L9701 R0+10. R1+.01 R2+3.25 R3+1.75
N7 M5 M9
N8 G0 G49 G90 Z0

6.6 CIRCULAR POCKET CLEAN-OUT

6.6.1 L9801 CIRCULAR POCKET CLEAN-OUT COUNTERCLOCKWISE

1. A tool must have been specified by an H or D word and the tool diameter MUST be in the tool table.
2. R0 represents the feed rate.
3. R1 represents radius on the corner of the tool.

> NOTE
> This can be used to regulate the step over distance. The larger this number, the less the amount of step over.

4. R2 represents diameter of the pocket.

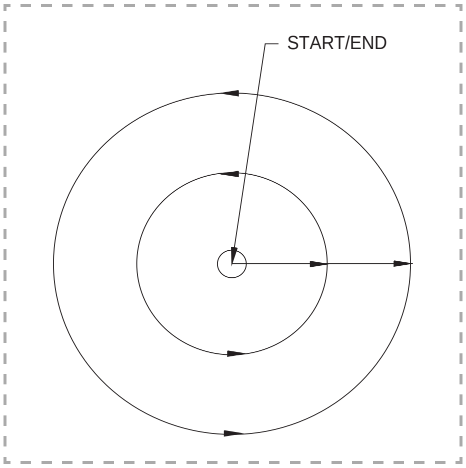


Figure 6-8: Counterclockwise Circular Pocket Clean-Out

The tool path drawing above shows how a 1.75 diameter circular pocket would be cleaned out by a .25" diameter tool. The procedure is as follows:

1. Position the X,Y axes to the center.
2. Position the Z axis to finished depth, selecting the proper H or D word to specify the diameter the tool is using.
3. Start Pocket Subroutine.

EXAMPLE:

N1 O1 (CIRC CLEANOUT CCW EXAMPLE
N2 M6 T1
N3 G0 G90 S2000 M3 X1.0 Y-1.0
N4 H1 D1 M7 Z.1
N5 G1 F10. Z-1.
N6 L9801 R0+10. R1+.01 R2+1.75
N7 M5 M9
N8 G0 G49 G90 Z0

6.6.2 L9901 CIRCULAR POCKET CLEAN-OUT CLOCKWISE

1. A tool must have been specified by an H or D word and the tool diameter MUST be in the tool table.
2. R0 represents the feed rate.
3. R1 represents radius on the corner of the tool.

> NOTE
> This can be used to regulate the step over distance. The larger this number, the less the amount of step over.

4. R2 represents the diameter of the pocket.

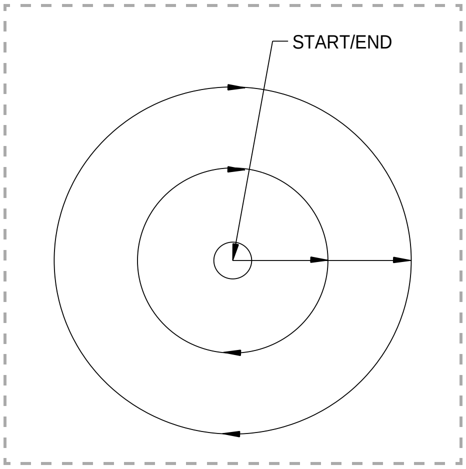


Figure 6-9: Clockwise Circular Pocket Clean-Out

The tool path drawing above shows how a 1.75 diameter circular pocket would be cleaned out by a .25" diameter tool. The procedure is as follows:

1. Position the X,Y axes to the center.
2. Position the Z axis to finished depth, selecting the proper H or D word to specify the diameter the tool is using.
3. Start Pocket Subroutine.

EXAMPLE:

N1 O1 (CIRC CLEANOUT CW EXAMPLE
N2 M6 T1
N3 G0 G90 S2000 M3 X1.0 Y-1.0
N4 H1 D1 M7 Z.1
N5 G1 F10. Z-1.
N6 L9901 R0+10. R1+.01 R2+1.75
N7 M5 M9
N9 G0 G49 G90 Z0

7.0 KEYBOARD

7.1KEYLOCKSWITCH (EDIT PROTECTION)

VERTICAL POSITION: Enable editing the CNC program.

HORIZONTAL POSITION: Disable editing the CNC program.

7.2 BLOCK SKIP SWITCH

DOWN POSITION: The down position (OFF) causes the control to ignore the

“/” symbol and execute the program block (see Run-time Menu, section 8.0 COMMANDS, MAN-0131).

UP POSITION: The up position (ON) causes the control to read the “/” symbol and skip the program block. The /N# line in the program will suppress the look ahead feature of the control with the switch in the up position (ON) (see Run-time Menu, section 8.0 COMMANDS, MAN-0131).

7.3 OPTIONAL STOP SWITCH

DOWN POSITION: The down position (OFF) causes the CNC to stop at the M1 function code and enter the WAITING state (see section 2.0 M FUNCTIONS. Also, see Run-time Menu, section 8.0 COMMANDS, MAN-0131).

UP POSITION: The up (ON) position causes the CNC to ignore the M1 function code (see Run-time Menu, section 8.0 COMMANDS, MAN-0131).

7.4 LIGHT ON/OFF SWITCH

This switch controls power to the work lamp.

7.5 VIDEO ON/OFF SWITCH

This switch controls the power going to the video screen. The operation of the machine is not affected by this switch.

7.6 RAPID TRAVEL SELECTOR

This switch selects the rapid movement speed (G0, G5).

100% of maximum rapid traverse

50% of maximum rapid traverse

25% of maximum rapid traverse

7.7 FEED RATE OVERRIDE POTENTIOMETER

This potentiometer controls the speed of axis motion.

1. AUTOMATIC mode: Axis motion can be increased to 150% or reduced to 0% of the programmed feed rates. Rapid moves are not affected. The maximum non-rapid feed rate is 375 ipm. A programmed feed rate above 250 ipm may only be overridden to 375 ipm. (For a 900 ipm rapid travel machine, the maximum non-rapid feed rate is 600 ipm. A programmed feed rate above 400 ipm may only be overridden to 600 ipm.)
2. SINGLE STEP mode: All AXIS motion, rapid, or feed rate is controlled by this override potentiometer.
3. DRY RUN mode: All AXIS motion, rapid, or feed rate is controlled by this override potentiometer.
4. SLIDE HOLD key:
   - After pressing the SLIDE HOLD key, the completion of the current move is subject to the override potentiometer. During a rapid move, if the Slide Hold is pressed, when the START key is pressed the feed rate potentiometer can override the rapid feed rate. If the next program block is programmed as a rapid move, the feed rate potentiometer will not override the feed rate.
   - An M49 code in the CNC program disables this potentiometer (see M49 Function, section 2.0 M FUNCTIONS, MAN-0131).

7.8 SPINDLE SPEED OVERRIDE POTENTIOMETER

This Potentiometer controls the programmed spindle RPM. The maximum override is 200% of the last programmed spindle speed (S word) executed by the CNC program or in MDI.The override percentage will not exceed the current drive RPM range (see S Function, section 1.0 SUMMARY, MAN-0131).

> WARNING
> A setting of 0% will not stop the spindle.

7.9 JOG KEY AND THE HAND WHEEL

Pressing the JOG key while in the command mode, or during the WAITING state while in the AUTO mode, initiates the JOG mode.

Once in the JOG mode, you can select an axis by turning the axis selector switch on the control panel, or using the keyboard to select the axis to jog by pressing X, Y, Z, A, B, or C keys.

The jog direction is selected by pressing the + or - key or by the direction the hand wheel is rotated clockwise for positive and counterclockwise for negative.

The speed of axis motion is selected by pressing the H, M, or L key (High, Medium, or Low), or by turning the jog increment selector switch.

H = .01 M = .001 L = .0001

Short moves can be made by pressing the JOG key repeatedly or by turning the hand wheel. If you hold down the JOG key, continuous motion can be made.

The speed of axis motion can be modified by the feed rate override pot. This can be used only when the JOG key is held for jog motion. The feed rate override pot does not affect the speed when using the hand wheel.

To exit the JOG mode, press the MANUAL key. The keyboard uses the familiar typewriter format for alphanumeric and special characters. The special function keys are described below (see SLIDE HOLD for JOG AWAY).

7.10 SHIFT

This key must be pressed along with another key to get the “upper” character displayed on that key. The shift key has no effect when pressed with keys that have only a single character. This key must be depressed while pressing the Spindle On/Off key to start the spindle manually.

7.11 ENTER

This key is used to indicate that you have completed the entry of your command or a line of data.

7.12 DELETE

This key stops input of data on the program line. The data entered prior to the delete key is disregarded and the control is ready to receive the new data.

7.13 BACKSPACE

This key allows you to move the cursor backward to correct a keying error. All characters to the right of the cursor, after backspacing, are disregarded by the control when the enter key is pressed. This data must be entered again to be entered in the program.

7.14 SPINDLE ON/OFF

This is an alternating switch that controls the spindle manually. As a safety feature, in order to turn the spindle on, you are required to first have the SHIFT key depressed before pressing the SPINDLE ON key. The spindle must be started manually after the external slide hold has been activated.

SPINDLE OFF does not require the use of the SHIFT key.

Spindle speed may be changed by an S word in the program or in MDI (see Section Nine, MDI command). The spindle orientation may be released by pressing the SPINDLE ON/OFF key. This will place the spindle in neutral by releasing the orientation arm.

7.15 AUTO

This key initiates execution of the current CNC program.

Pressing this key in the command mode the CNC displays the AUTO MODE screen. Pressing AUTO again initiates program processing. The control continues to process the program until the buffer is filled. If the AUTO key is not pressed the control enters the WAITING state when the buffer is filled.

When the machine enters the WAITING state, the program may be executed by pressing the START key or returning to the COMMAND mode by pressing the MANUAL key.

The AUTO key is also used to end the WAITING state or when you want to end the SINGLE STEP mode.

Typing MU during the machining cycle causes the CNC to display the BACKGROUND EDIT MENU. This allows the operator to select dry run options (run time menu), the offset menu or the functions menu. The changes made in the BACKGROUND EDIT MENU are not effective until after the last program block that was processed prior to the change being entered.

Pressing the space bar toggles between the BACKGROUND EDIT MENU and the AUTO MODE screen (see Background Editing).

7.16 SLIDE HOLD

The SLIDE HOLD button stops the movement of the X,Y, Z, A, & B. The Spindle and coolant are not affected by the SLIDE HOLD. The CNC will resume motion after the START or AUTO button is pressed.

> WARNING
> Before continuing AUTO, check to see if the Spindle or Coolant was shut off. If so, manually turn on the Spindle by depressing the SHIFT and SPINDLE ON button. To turn on the Coolant, press the COOLANT button.

7.17 SLIDE HOLD, JOG AWAY

By pressing the **JOG button** while in SLIDE HOLD, any axis can be moved away from its current position. This Jog Away feature allows an optional automatic return of the jogged axes to that position to complete the machining cycle. The MANUAL key exits JOG and returns the CNC to SLIDE HOLD.After pressing the MANUAL key the CNC prompts the operator:

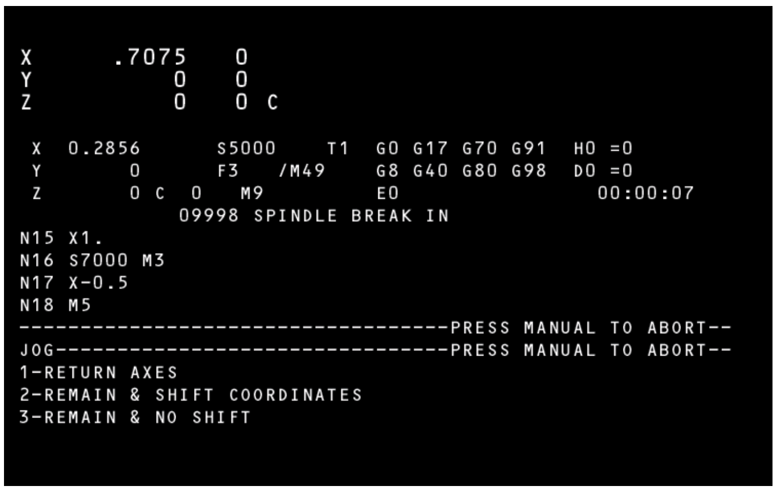


Figure 7-1: Slide Hold, JOG AWAY

PRESSING 1: The CNC will return the axes to the position before the JOG was initiated. This motion is accomplished at a rapid feedrate. Make sure the Z axis is in a position to clear work and fixturing.

> WARNING
> Slide Hold or Feedrate Override Pot will not function while the CNC is returning the axis to original location! The Rapid Travel selector will change the return speed.

PRESSING 2: The CNC will continue the machining operation, change the position read-out to the last machined position, while retaining the JOG offset until one of the following conditions cancels the jog offset:

- aborting the AUTO mode.
- a G28 with an axis movement will return all axis to zero.
- a tool change is commanded (M6) Z axis only.

PRESSING 3: The CNC will continue the machining operation, retain the JOG offset, and the position read-out will reflect the current position including the JOG offset. This will remain active until one of the following are initiated:

- aborting the AUTO mode.
- a G28 with an axis movement will return all axis to zero.
- a tool change is commanded (M6) Z axis only.

NOTE

A tool change is commanded (M6) **does not** cancel JOG AWAY for X or Y axis.

7.18 EXTERNAL SLIDE HOLD

When the machine is running a program and the doors are opened the external slide hold is activated. The machine motion and the spindle are stopped. When the doors are closed and the start button is pressed a message to turn the spindle on will appear if the spindle is turned off manually.

7.19 SPACEBAR, BACKGROUND EDITING

To use background editing, the programmer must have the control in AUTO and press the SPACE BAR at the keyboard.

The space bar toggles the screen between the BACKGROUND EDITING MENU and the AUTO MODE screen. While in the BACKGROUND EDIT MENU the programmer now has three options to choose from. The active program currently running in AUTO will be displayed to the screen, and below it will be displayed the background edit options:

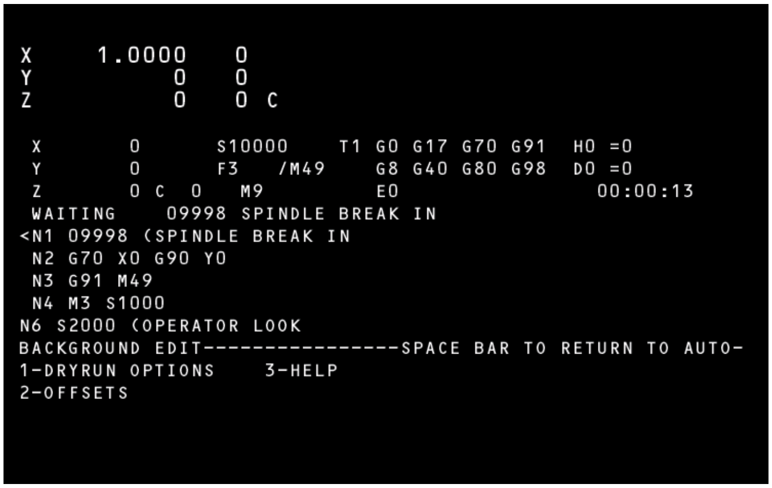


Figure 7-2: Spacebar, Background Editing

OPTION 1 DRYRUN OPTIONS: Selection will display the dryrun options run time menu.

OPTION 2 OFFSETS: The operator can edit the FIXTURE and TOOL OFFSETS, TOOL TIME and the MACRO VARIABLES tables (See the DF, DT, DTT and FO commands in section 8.0 COMMANDS, MAN-0131.)

OPTION 3 HELP: The operator has several options that are similar to the PAGE EDIT (PA) command. The active program currently running in AUTO can be edited.

U-UPT-TOPC-CHANGES-SEARCHP-PROGRAM
D-DOWNB-BOTTOMI-INSERTR-REPLACEJOG
ENTER-PAGE DOWN BACKSPACE-PAGE UP DEL-DELETE
U KEY, move the cursor up.
D KEY, move the cursor down.
T KEY, move the cursor to the top of the program.
B KEY, move the cursor to the bottom of the program.
C KEY, change line or edit the line on which the cursor points.
I KEY, insert below the cursor line.
S KEY, search for a specified word.

R KEY, search and replace a specified word.
P KEY, change or review programs in memory.
JOG KEY, JOG away from the current position.

> WARNING
> Be extremely careful when making changes to the current program in auto!

Background editing will allow the programmer to change, add, or delete any line of program code other than the lines that are currently being processed by the control. If a line of code to be changed is within the LOOK AHEAD BUFFER the control will display a message “CONFLICT WITH AUTO” and the programmer must wait until the BUFFER has passed by the line to be edited. This prevents the interruption of continuous machining of the current part during AUTO. The macro WAIT will prevent the LOOK AHEAD at the M2 (see Section 18.0 MACROS, MAN-0131).

7.20 SINGLE STEP

This key issues the SINGLE STEP command which causes the CNC to enter the WAITING state between each program block. While in the SINGLE STEP mode all rapid moves are subject to the feed rate pot override. The WAITING state is ended by pressing one of the following keys:

- START - executes one block of the CNC program.
- AUTO - exits the SINGLE STEP mode for continuous program operation.
- JOG - initiates the JOG Away feature (see SLIDE HOLD).
- MANUAL - terminates program execution and enters the COMMAND mode.

7.21 START

This key is used to terminate a WAITING state that has been initiated in one of the following situations:

1. You have just initiated the operation of the machine by use of the AUTO, HOME or MANUAL DATA INPUT command.

2. The WAITING state has been entered to give you time to make any necessary preparations for the move.

3. The CNC program has been terminated by an M2 or M30 (end of program) instruction and the CNC is ready to begin execution from the start.

4. The SLIDE HOLD key was pressed.

5. The CNC program has issued a M0 or M1 command.

6. The CNC is in the SINGLE STEP mode. This key is also used to override a programmed dwell command (see G4 code, section 3.0 G CODES, MAN-0131.).

7.22 EMERGENCY STOP

This switch is used to disconnect the control from all machine axes and the spindle. The control will be put in the COMMAND mode. To restart the program after an emergency stop, reset the EMERGENCY STOP button by turning it clockwise and press the JOG key to restart the servo amplifiers. The axes in motion and the spindle will **coast** to a stop.

7.23 MANUAL

This key interrupts the current activity of LIST PROGRAM, PAGE PROGRAM, SUM PROGRAM, DISPLAY TOOL TABLE, DISPLAY FIXTURE OFFSETS, MENU or JOG and returns the CNC to the COMMAND mode.

When the CNC is executing a CNC program, this key is ignored unless the CNC has been put in the WAITING state by the SLIDE HOLD command, M0 command, M1 command, a problem during tool change, the SINGLE STEP command or an M2 or M30 (end of program) function.

7.24 COOLANT-1 (FLOOD)

This is an alternating switch that controls the COOLANT-1 function manually. Same as M7 or M8.

7.25 COOLANT-2 (MIST)

This is an alternating switch that controls the COOLANT-2 function manually. Same as M7 or M8.

7.26 TURRET CW

When this button is pressed, the turret will be rotated clockwise to the next tool station. If this button is depressed, the turret will rotate as long as the button is held down.

7.27 TURRET CCW

When this button is pressed, the turret will be rotated counterclockwise to the next tool station. If this button is depressed, the turret will rotate as long as the button is held down.

8.0 COMMANDS

8.1 COMMANDS

Table 8-1: 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	
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	

Table 8-1: (Continued) Command List

DESCRIPTION	COMMAND	PARAMETERS
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	

8.2 AUTO

8.2.1 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.

8.3 BACKLASH

8.3.1 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

8.4 CHANGE DEVICE

8.4.1 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 14.0 COMMUNICATIONS, MAN-0131).

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 Protocol Types, section 14.0 COMMUNICATIONS, MAN-0131.)

> 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.

8.5 CHANGE PROGRAM

8.5.1 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.

8.6 COMMAND LOCK

8.6.1 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.

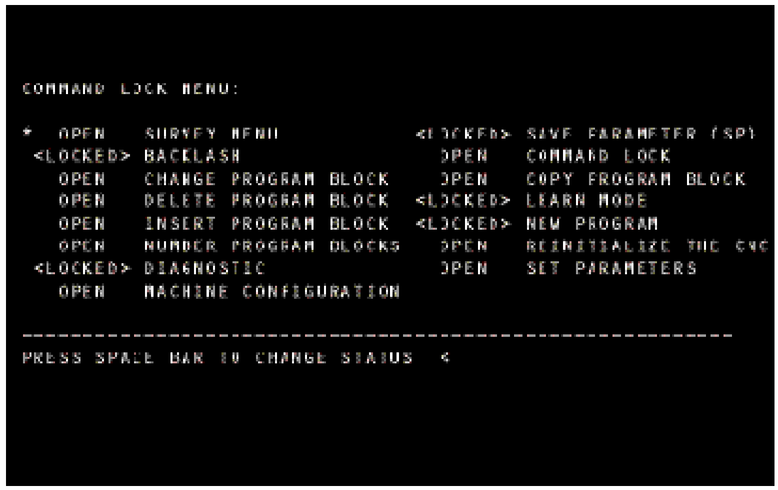


Figure 8-1: Command Lock Menu

8.7 COPY PROGRAM

8.7.1 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 8-2: 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.

8.8 COLD START

8.8.1 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 11.0 MACHINE COORDINATE SYSTEM, MAN-0131). 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.

8.9 DISPLAY BUCKET

8.9.1 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.

8.10 DELETE BLOCKS

8.10.1 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.

8.11 DISPLAY FEED FORWARD PARAMETERS

8.11.1 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

8.12 DISPLAY FIXTURE OFFSETS

8.12.1 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

8.13 DIAGNOSTICS

8.13.1 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.

8.14 DIRECT NUMERICAL CONTROL

8.14.1 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 DNC, section 14.0 COMMUNICATIONS, MAN-0131).

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.

8.15 XMODEM DIRECT NUMERICAL CONTROL

8.15.1 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.

8.16 DRAW

8.16.1 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.

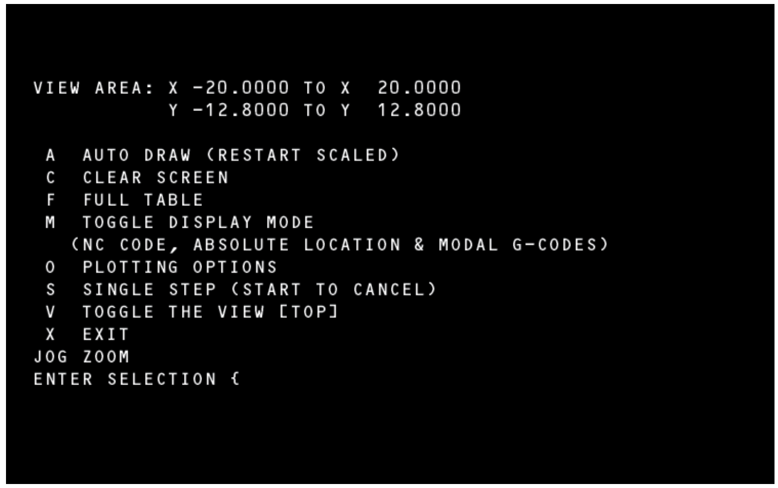


Figure 8-4: Graphics Menu

8.16.2 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).

8.16.3 C = CLEAR

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

8.16.4 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.

8.16.5 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.

8.16.6 O = OPTIONS PLOTTING

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

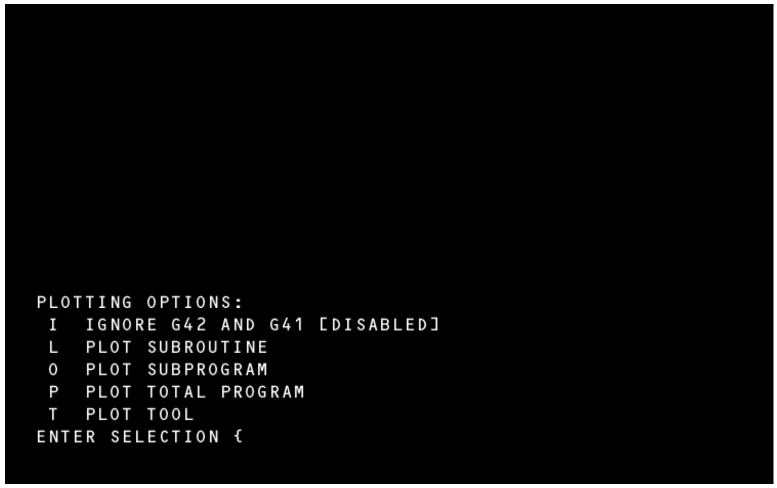


Figure 8-5: Plotting Options Menu

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

8.16.7 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.

8.16.8 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.

8.16.9 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.

8.17 DISPLAY TOOL TABLE

8.17.1 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.

8.18 DISPLAY TOOL TIME TABLE

8.18.1 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.

8.18.2 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.

8.19 DISPLAY VARIABLE TABLE COMMAND

8.19.1 DV

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

8.20 FIXTURE OFFSET

8.20.1 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 11.0 FIXTURE OFFSETS, MAN-0131).

8.21 HOME AXIS

8.21.1 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.

8.22 INSERT BLOCKS

8.22.1 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.

8.23 JOG AXIS

8.23.1 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 Jog Key and the Hand Wheel, section 7.0 KEYBOARD, MAN-0131).

8.24 LEARN MODE

8.24.1 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.

8.25 LIST PROGRAM

8.25.1 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).

8.26 MACRO

8.26.1 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

8.27 MANUAL DATA INPUT

8.27.1 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.

8.28 MEMORY

8.28.1 ME

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

8.29 MENU

8.29.1 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.

8.30 NEW PROGRAM

8.30.1 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.

8.31 RENUMBER PROGRAM

8.31.1 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.

8.32 PROGRAM PAGE EDIT

8.32.1 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.

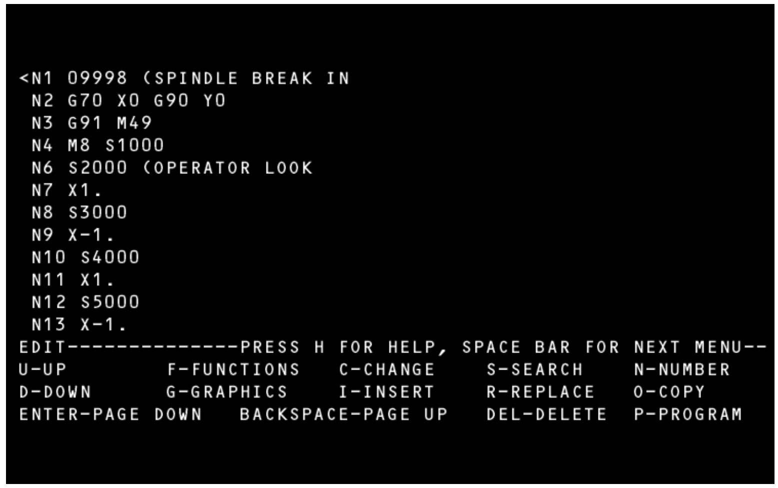


Figure 8-8: **Program Page Edit Menu**

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.

8.33 FUNCTION MENU

8.33.1 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.

8.33.2 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.

8.33.3 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.

8.33.4 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.

8.34 GRAPHICS MENU

see Draw command.

8.35 BACKGROUND EDITING

8.35.1 SPACE BAR OR MU (WHEN USED IN THE AUTO MODE)

To use background editing the programmer must have the control in AUTO and press the SPACE BAR at the keyboard. Pressing the space bar changes the 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.

8.36 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 Optional Stop Switch, section 7.0 KEYBOARD, MAN-0131). Status of the switch is displayed on the auto mode screen as OPT when on.
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.

NUMBER	DRY RUN SUMMARY
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.

8.37 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.

8.38 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:

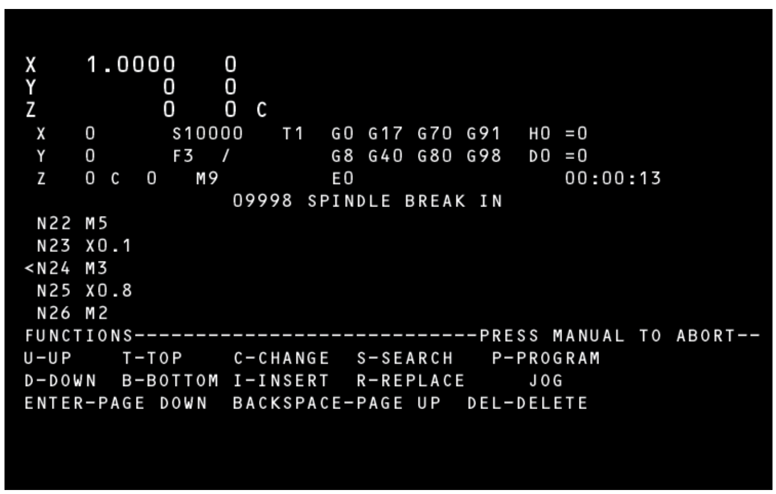


Figure 8-9: **Editing Features**

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!

8.39 PROGRAM MAINTENANCE LIBRARY

8.39.1 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:

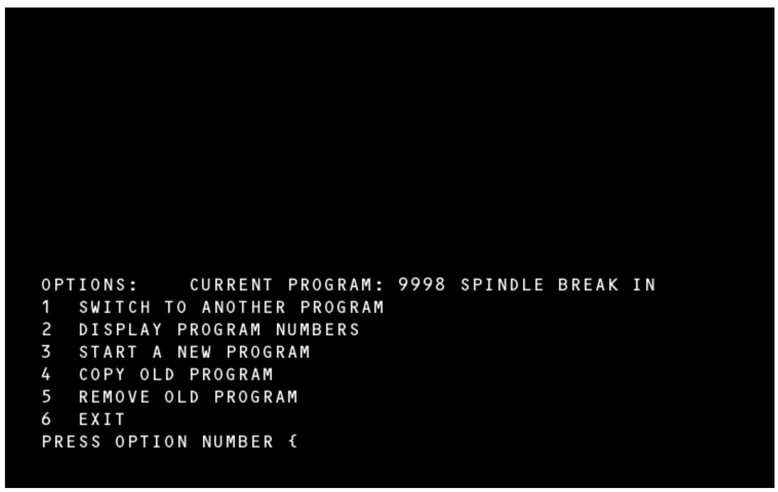


Figure 8-10: Program Maintenance Library Menu

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 Multiple Part Program, section 1.0 SUMMARY, MAN-0131).

8.40 PUNCH PROGRAM TAPE

8.40.1 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.

8.41 REINITIALIZE

8.41.1 RI

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

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.

8.42 SET COLD START

8.42.1 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.

8.43 SET HOME POSITION OF ALL AXES

8.43.1 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.

8.44 SET HOME POSITION FOR ONE AXIS

8.44.1 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.

8.45 METRIC PROGRAMMING

8.45.1 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.

8.46 INCH PROGRAMMING

8.46.1 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.

8.47 SET SYSTEM PARAMETERS

8.47.1 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.

8.48 PALLET PROGRAMMING

8.48.1 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.

8.49 PROGRAMMING FORMATS

8.49.1 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.

8.49.2 FORMAT 1

Figure 8-12: Format 1 Screen Display

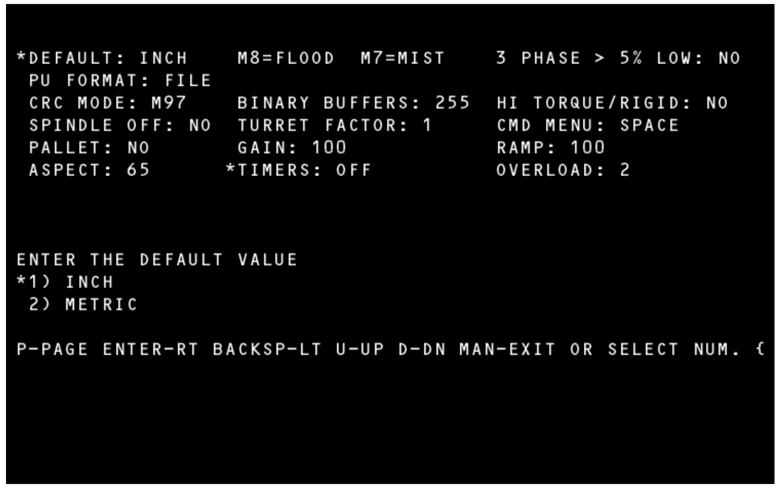


Figure 8-13: Format 1 Screen Display (continued)

Figure 8-14: Format 1 Screen Display (continued)

8.49.3 FORMAT 2

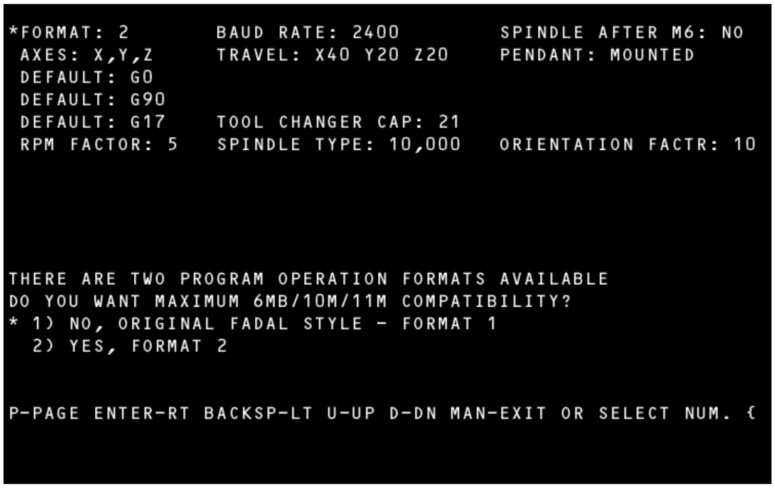


Figure 8-15: Format 2 Screen Display

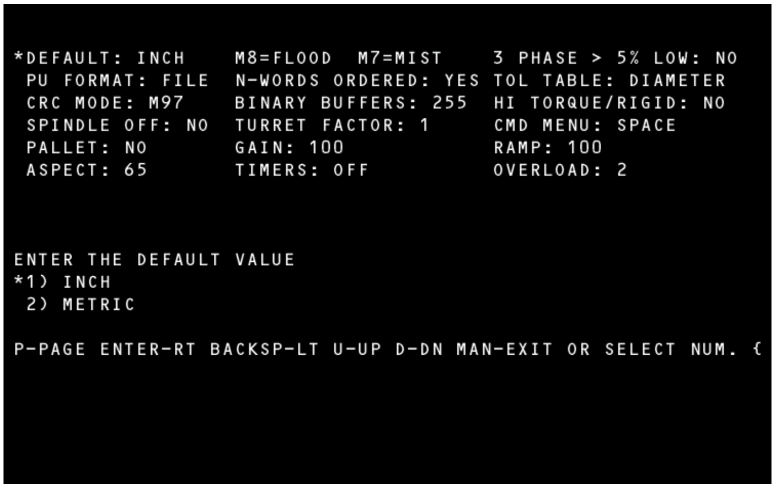


Figure 8-16: Format 2 Screen Display (continued)

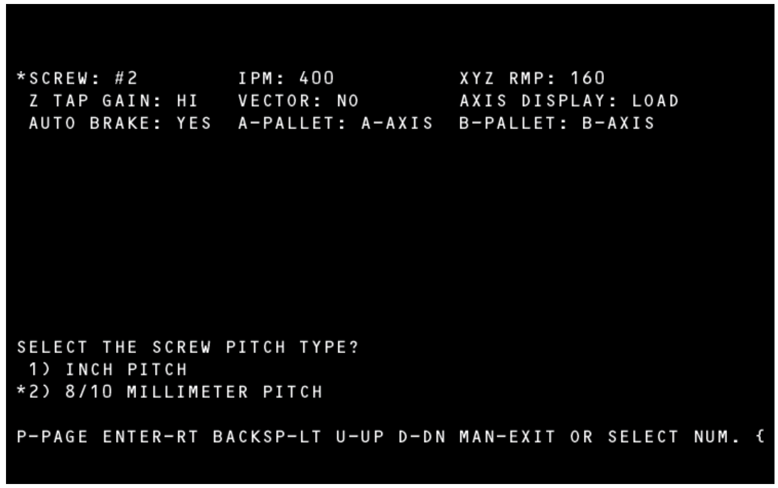


Figure 8-17: Format 2 Screen Display (continued)

> **NOTE**
> Depending on the parameter that the cursor is selecting, not all parameters are displayed.

8.50 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.

8.51 AXES:X,Y,Z

ENTER THE AXIS CONFIGURATION

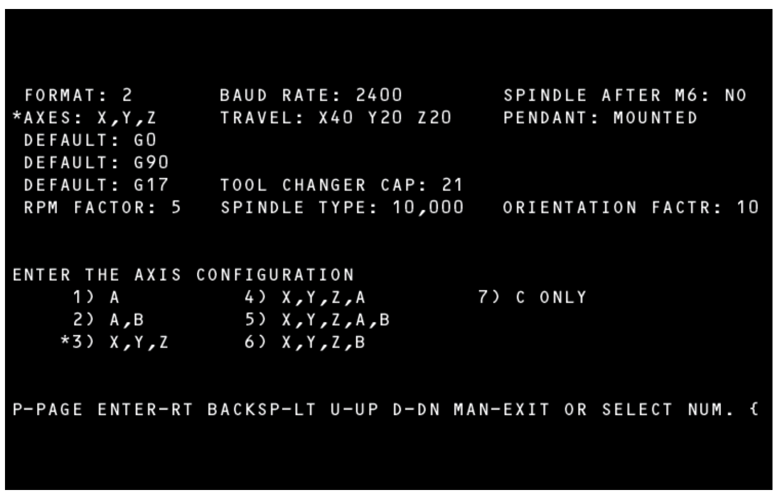


Figure 8-19: Axis Configuration

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

8.52 DEFAULT: G0

ENTER THE DEFAULT VALUE

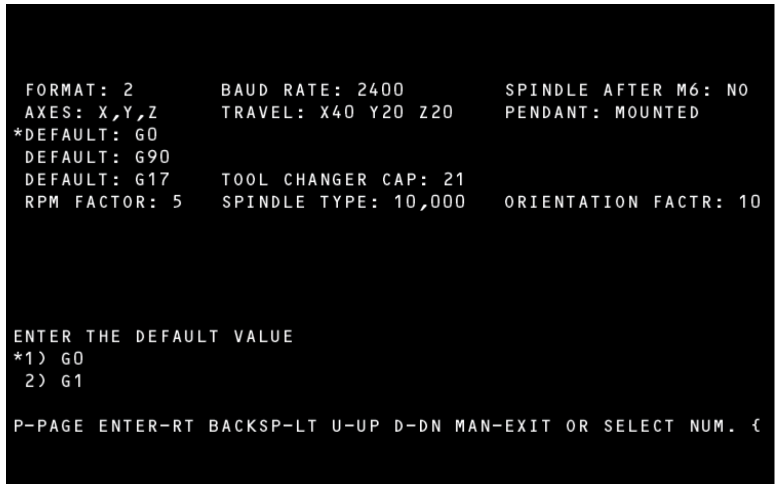


Figure 8-20: Default Value G0

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

8.53 DEFAULT: G90

ENTER THE DEFAULT VALUE

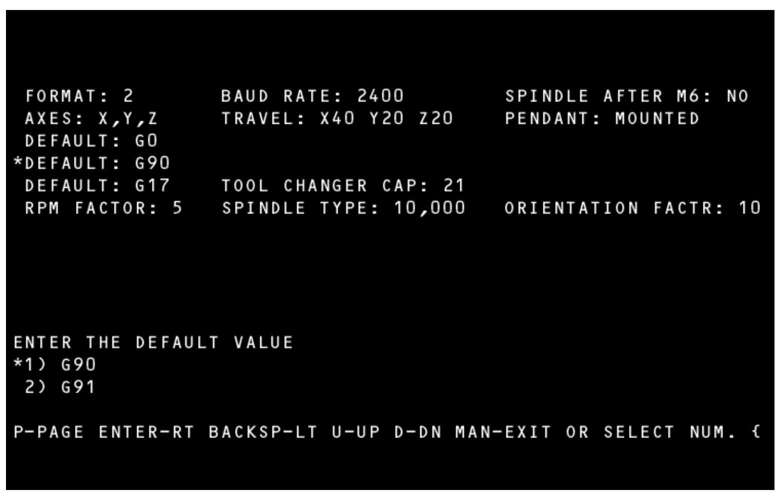


Figure 8-21: Default Value G90

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

8.54 DEFAULT: G17

ENTER THE DEFAULT VALUE

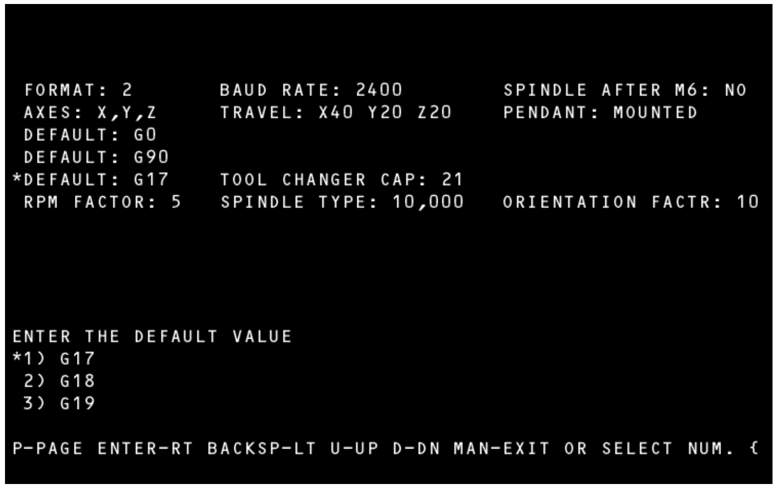


Figure 8-22: Default Value G17

This parameter is used to select the default machine plane.

8.55 RPM FACTOR

ENTER THE SPINDLE RPM ADJUSTMENT FACTOR

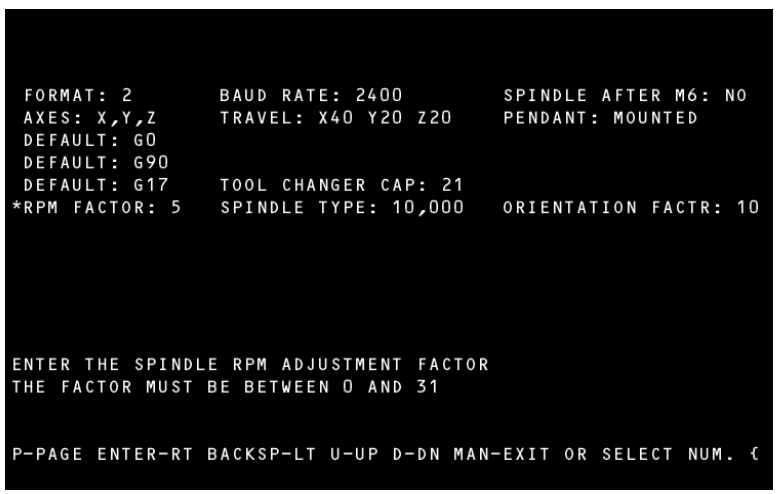


Figure 8-23: RPM Factor

This parameter should only be adjusted by trained maintenance personnel.

8.56 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.

8.57 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.

8.58 TOOL CHANGER CAP

ENTER THE TOOL CHANGER CAPACITY

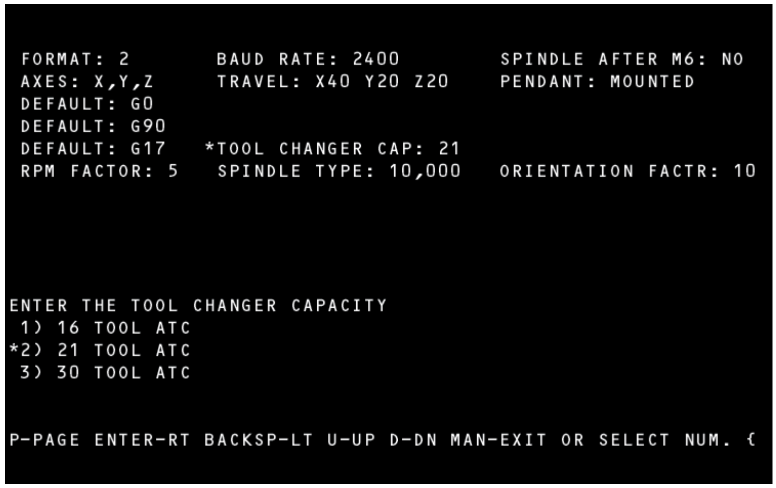


Figure 8-26: Tool Changer Capacity

Select the appropriate tool changer capacity.

8.59 TIMERS

SELECT THE AUTOMATIC TOOL TIMER MODE

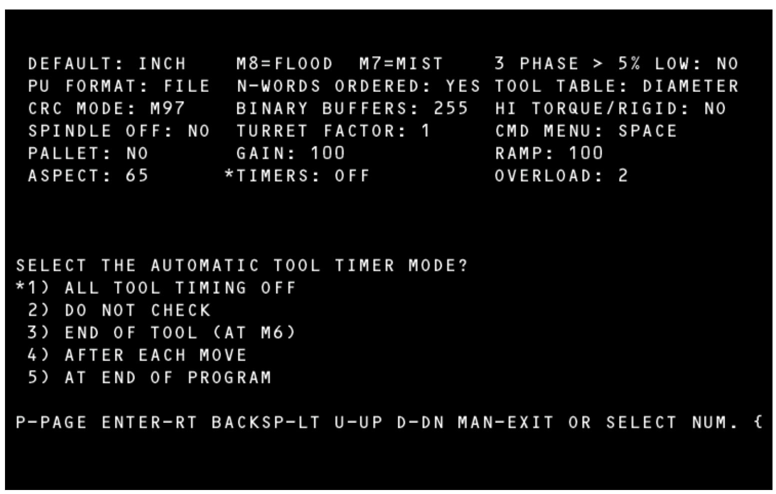


Figure 8-27: Automatic Toll Timer Mode

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.

8.60 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.

8.61 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.

8.62 PENDANT

ENTER THE PENDANT STYLE

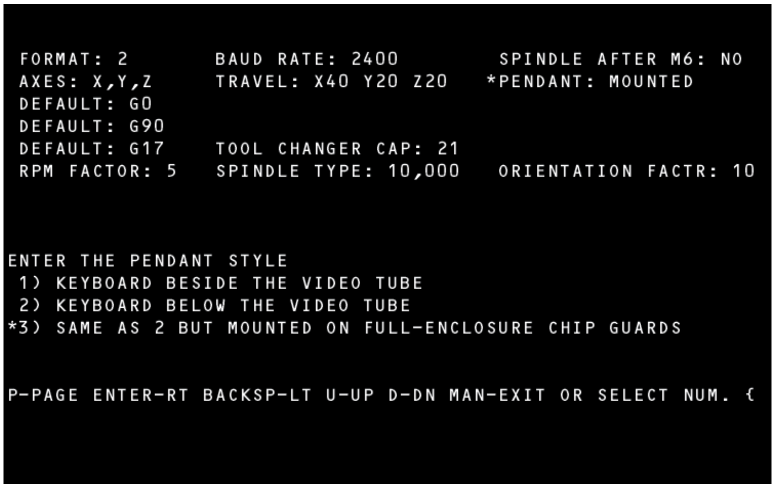


Figure 8-30: Pendant

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.

8.63 IMM. FIXED CYCLE

SHOULD A FIXED CYCLE EXECUTE IMMEDIATELY?

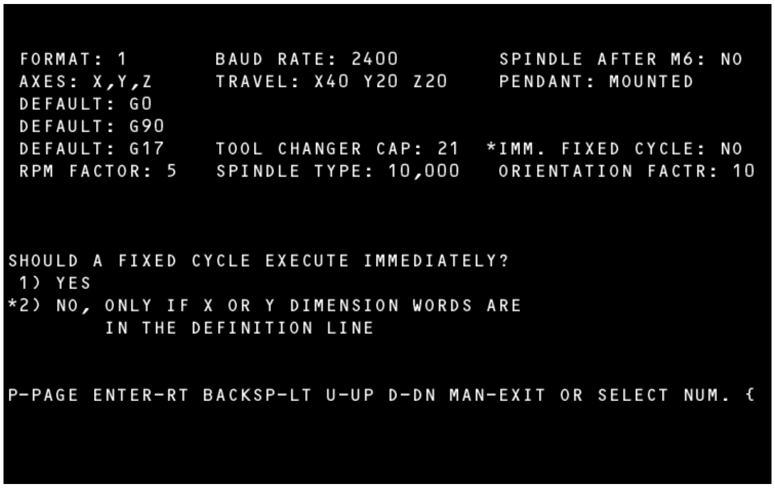


Figure 8-31: Imm. Fixed Cycle

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.

8.64 ORIENTATION FACTOR

ENTER THE SPINDLE RPM ADJUSTMENT FACTOR

THE FACTOR MUST BE BETWEEN 0 AND 31

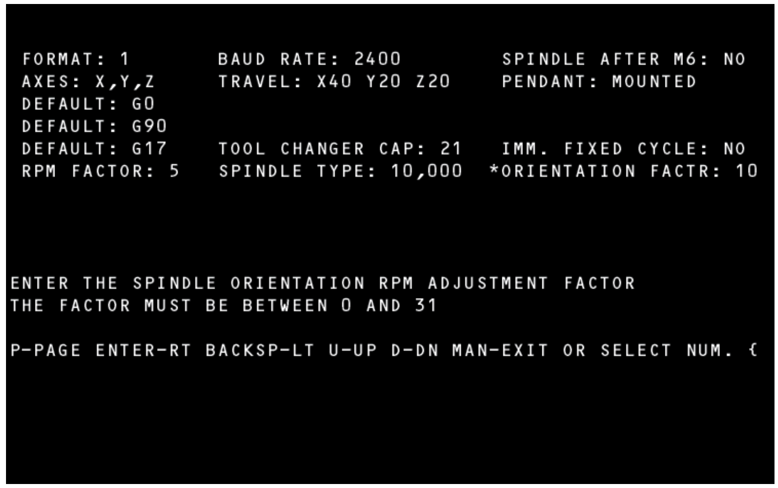


Figure 8-32: Orientation Factor

This parameter should only be adjusted by trained maintenance personnel.

8.65 DEFAULT: INCH

ENTER THE DEFAULT VALUE

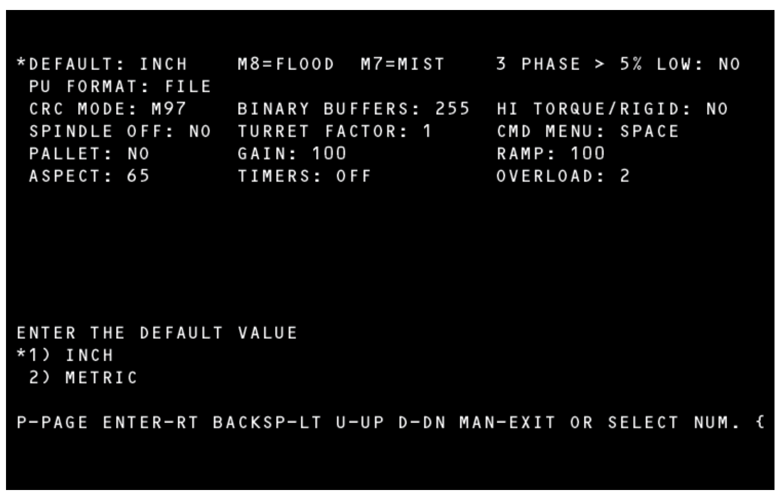


Figure 8-33: Default Value: Inch

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.

8.66 PU FORMAT

SELECT PUNCH OUTPUT FORMAT

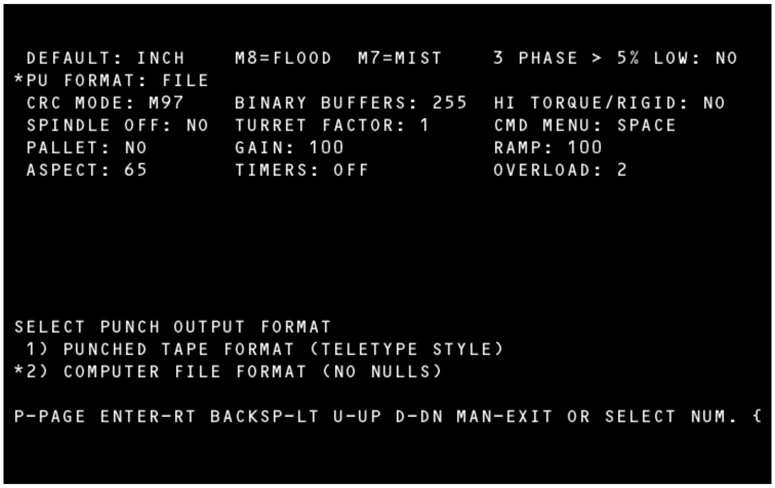


Figure 8-34: PU Format

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

8.67 CRC MODE

ENTER THE DEFAULT OUTSIDE CORNER MOVEMENT

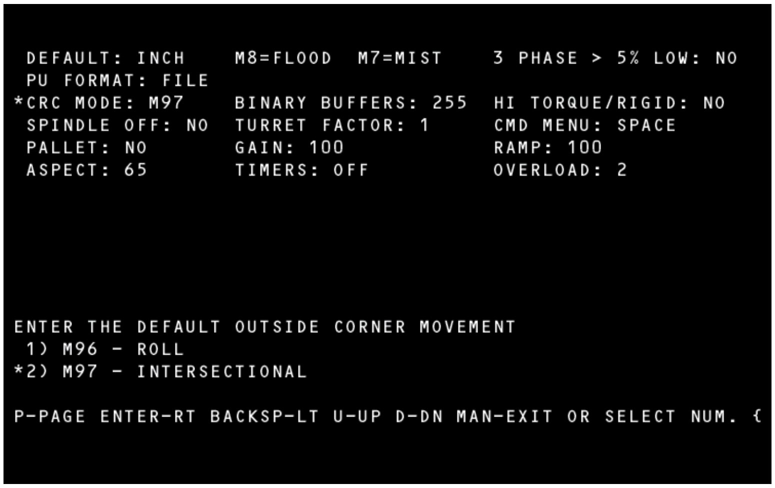


Figure 8-35: CRC Mode

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

8.68 PALLET

DO YOU HAVE A PALLET CHANGER?

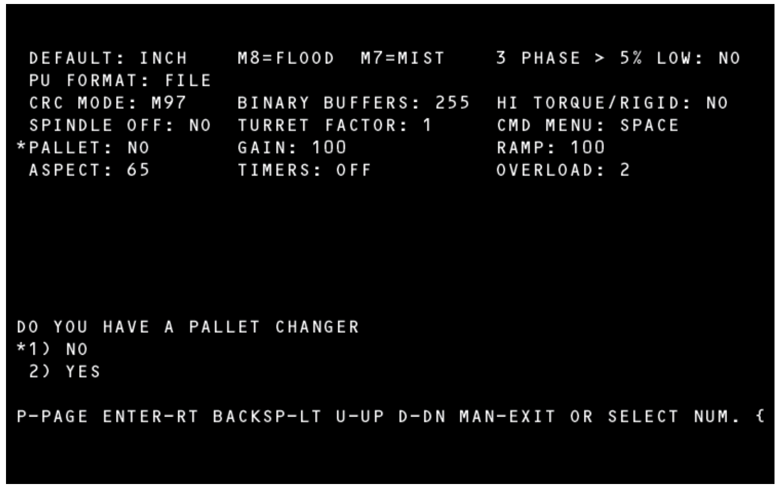


Figure 8-36: Pallet

Select the option appropriate for the machine.

8.69 M7-FLOOD, M8-MIST

ENTER M7, M8 PREFERENCE

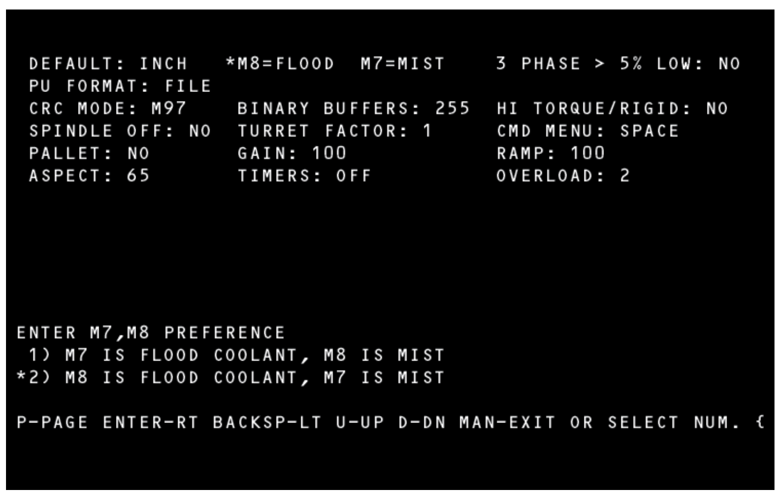


Figure 8-37: M7-Flood, M8-Mist

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

8.70 BINARY BUFFERS 255

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

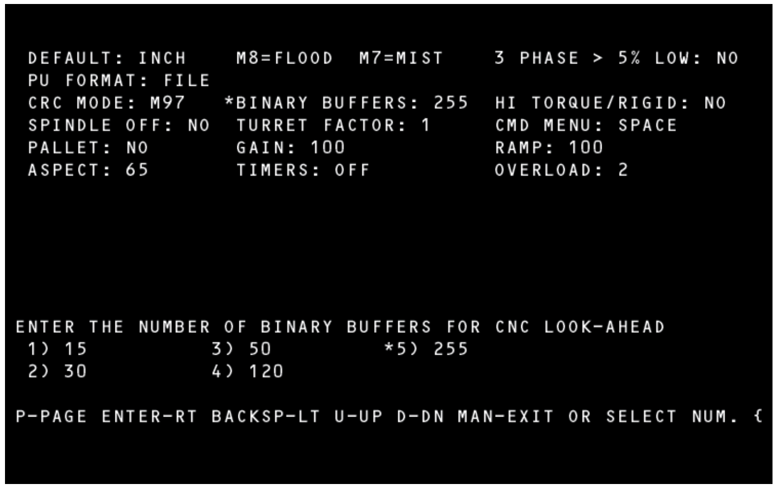


Figure 8-38: Binary Buffers: 255

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.

8.71 TURRET FACTOR

ENTER THE ENGAGEMENT FACTOR FOR THE TOOL TURRET GENEVA GEAR

THE FACTOR MUST BE BETWEEN 1 AND 50

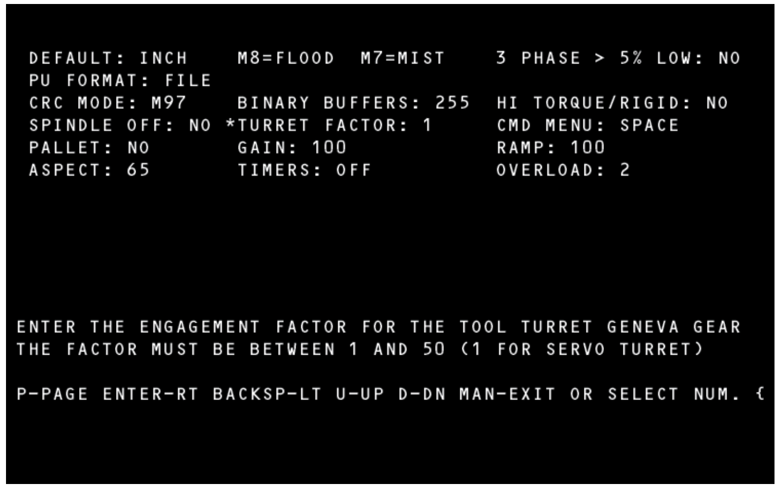


Figure 8-39: Turret Factor

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

8.72 GAIN

ENTER THE GAIN FACTOR FOR RIGID TAPPING

THE FACTOR MUST BE BETWEEN 0 AND 255

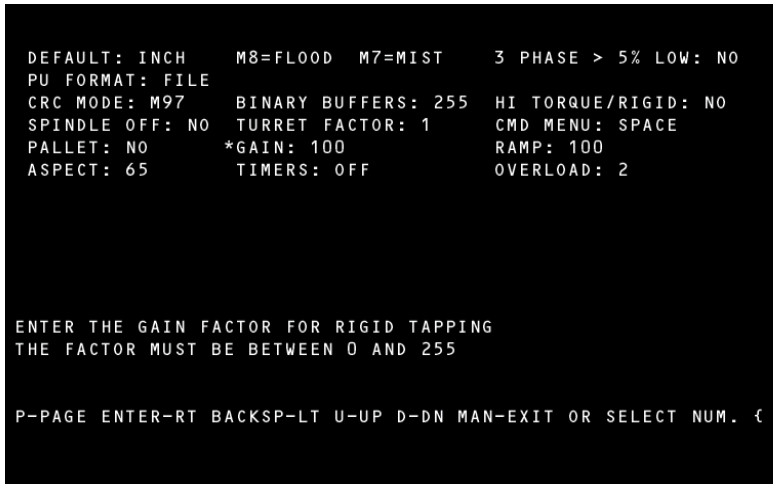


Figure 8-40: Gain

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.

8.73 3 PHASE 5% LOW: NO

IS YOUR 3 PHASE POWER MORE THAN 5% LOW?

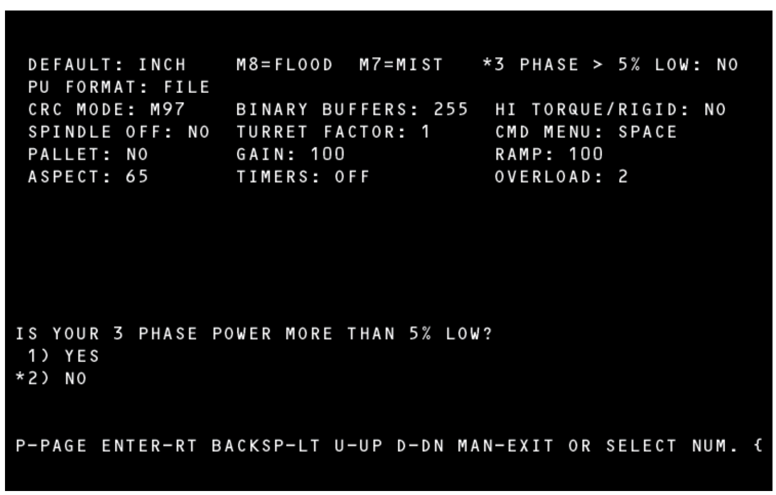


Figure 8-41: 3 Phase 5% Low

The selection chosen is based upon the building power supply.

8.74 HIGH TORQUE/ RIGID TAP

DO YOU HAVE THE HIGH TORQUE OR RIGID TAP OPTION?

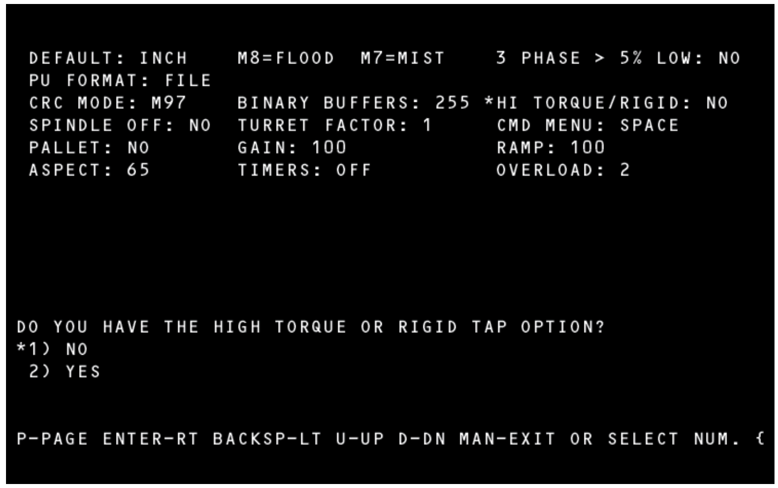


Figure 8-42: HighTorque/Rigid Tap

This parameter is set at the factory.

8.75 CMD MENU

TURN COMMAND MENUS:

Figure 8-43: CMD Menu

The operator may select the command menu structure.

8.76 RAMP

ENTER THE RAMP FACTOR FOR RIGID TAPPING

THE FACTOR MUST BE BETWEEN 0 AND 255

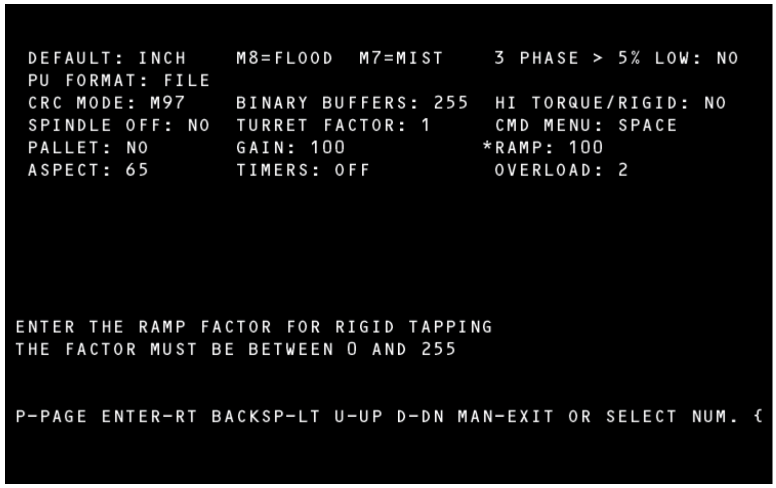


Figure 8-44: Ramp

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

8.77 A-AXIS RATIO

ENTER A-AXIS RATIO

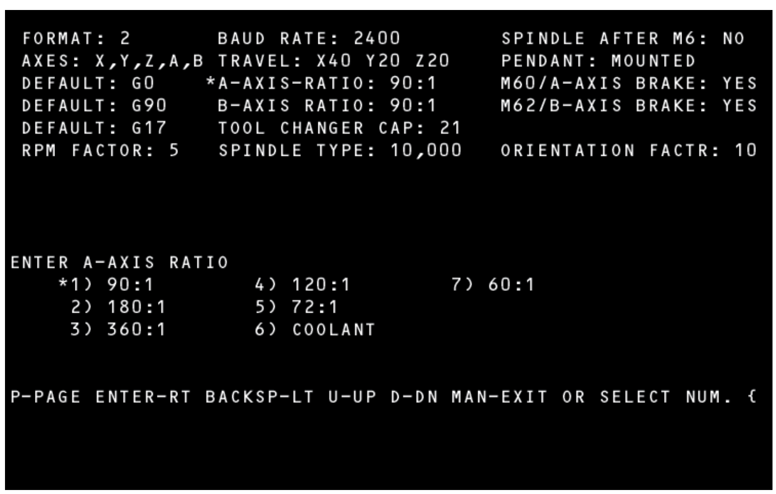


Figure 8-45: A-Axis Ratio

Select the appropriate option for the rotary table being used.

8.78 B-AXIS RATIO

ENTER B-AXIS RATIO

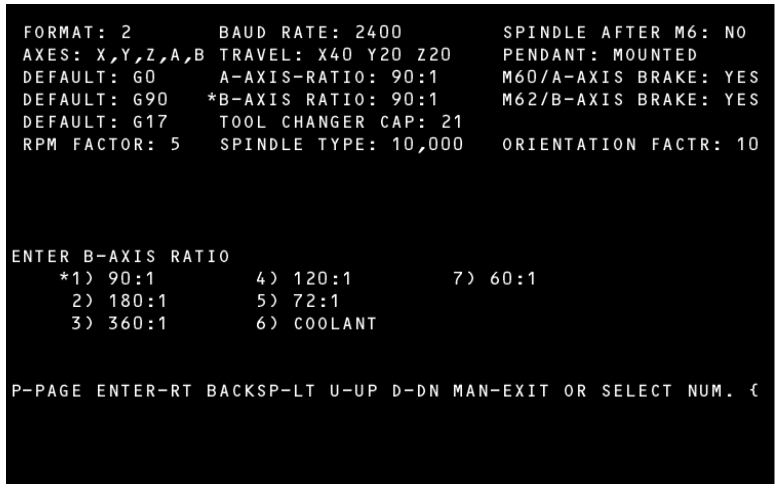


Figure 8-46: B-Axis Ratio

Select the appropriate option for the rotary table being used.

8.79 M60/A-AXIS BRAKE

DOES M60 TURN ON THE A-AXIS BRAKE?

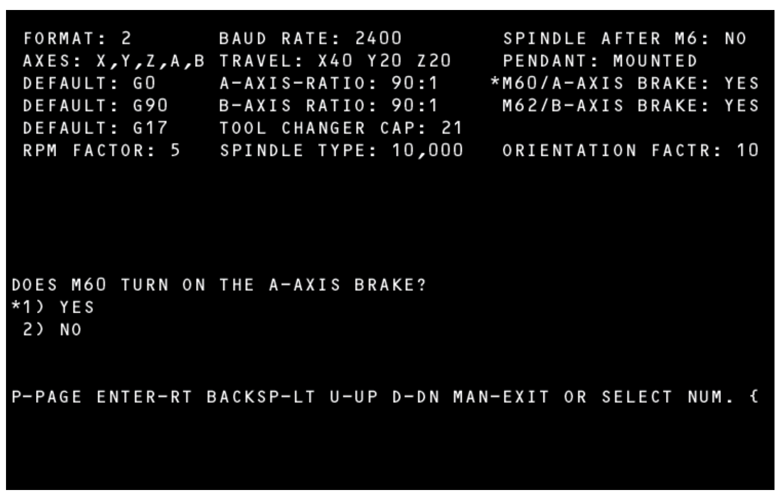


Figure 8-47: M60/A-Axis Brake

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

8.80 M62/B-AXIS BRAKE

DOES M62 TURN ON THE B-AXIS BRAKE?

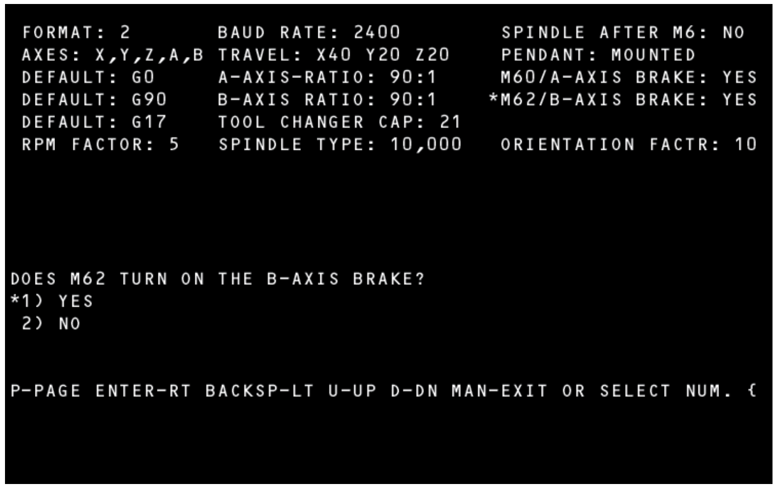


Figure 8-48: M62/B-Axis Brake

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

8.81 N-WORDS ORDERED

ENTER THE N-WORD SEQUENCE CONFIGURATION

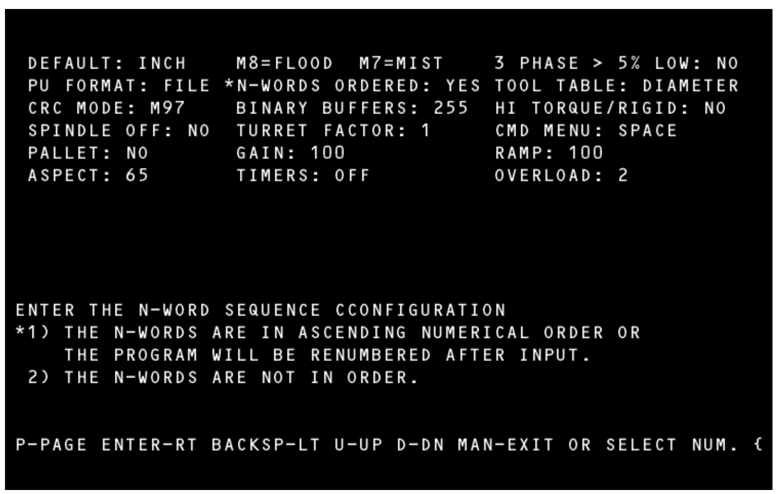


Figure 8-49: N-Words Ordered

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.

8.82 TOOL TABLE

WILL THE TOOL COMPENSATION TABLE HAVE THE RADIUS OR DIAMETER?

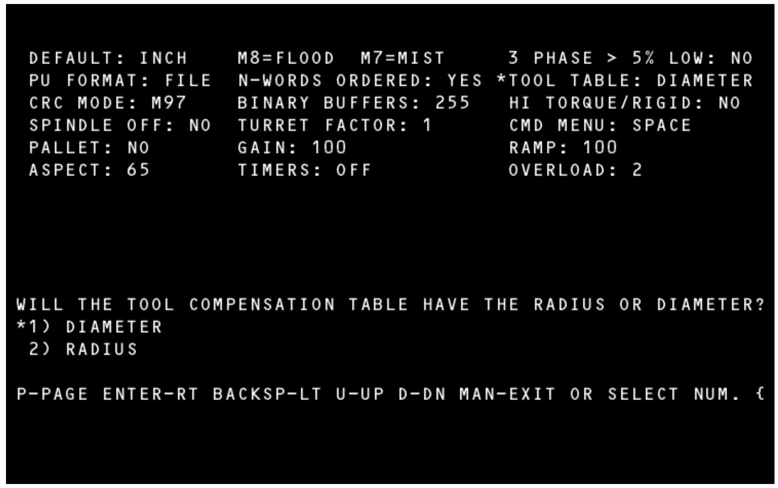


Figure 8-50: Tool Table

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).

8.83 SET TURRET ORDER

8.83.1 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.

8.83.2 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.

8.84 SET TOOL LENGTH OFFSET

8.84.1 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.

8.85 SUM PROGRAM

8.85.1 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.

8.85.2 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=.

8.85.3 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=.

8.85.4 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=.

8.86 SURVEY

8.86.1 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.

8.87 TAPE (PROGRAM) INPUT

8.87.1 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.

8.88 TOOL CHANGER OPEN

8.88.1 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.

8.89 TOOL PARAMETER DEFINITION

8.89.1 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.

8.90 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.

8.91 UTILITY

8.91.1 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.

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.

8.91.2 OFFSET UTILITY OPTIONS:

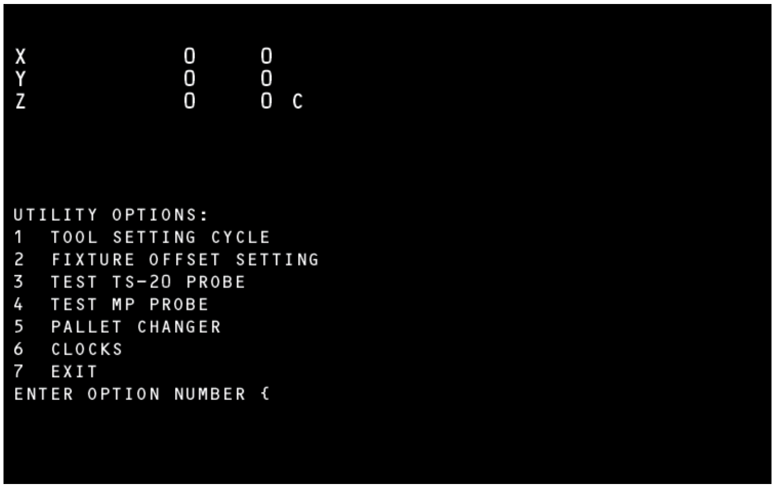


Figure 8-52: Offset Utility Options

8.91.3 OFFSET UTILITY OPTION 1

TOOL SETTING CYCLE

This option is used to set tool length offsets. (See section 15.0 TOUCH PROBE, MAN-0131.)

8.91.4 OFFSET UTILITY OPTION 2

Fixture Offset Setting

This option is used to set fixture offset locations.

FIXTURE OFFSET Menu Items:

8.91.5 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.

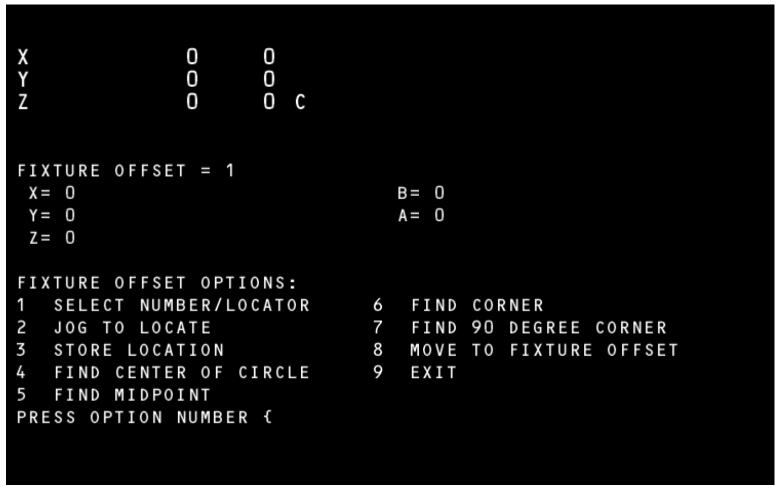


Figure 8-53: Select Number/Locator

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.

8.91.6 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.

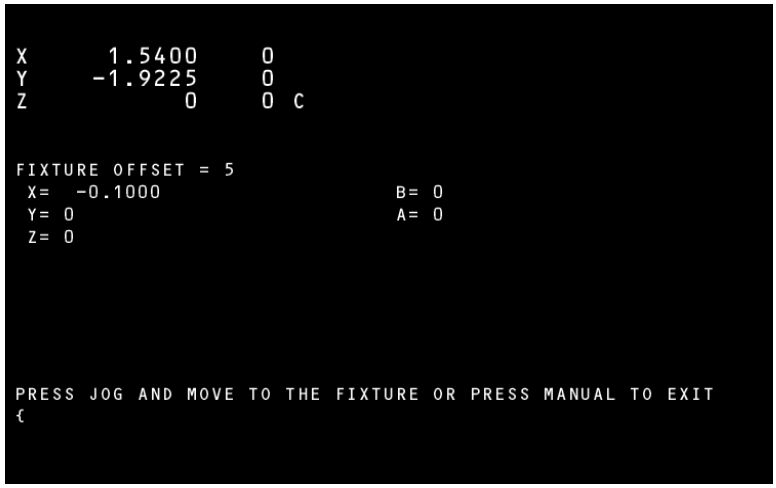


Figure 8-54: Jog to Locate

8.91.7 ITEM 3

STORE LOCATION

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

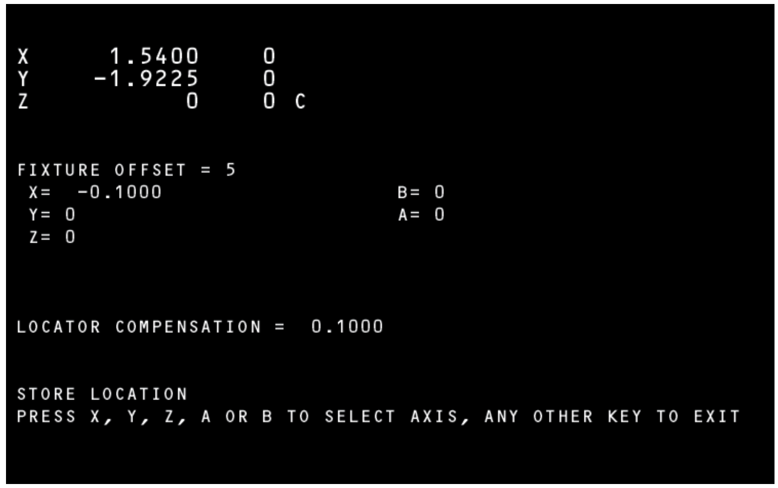


Figure 8-55: Store Location

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.

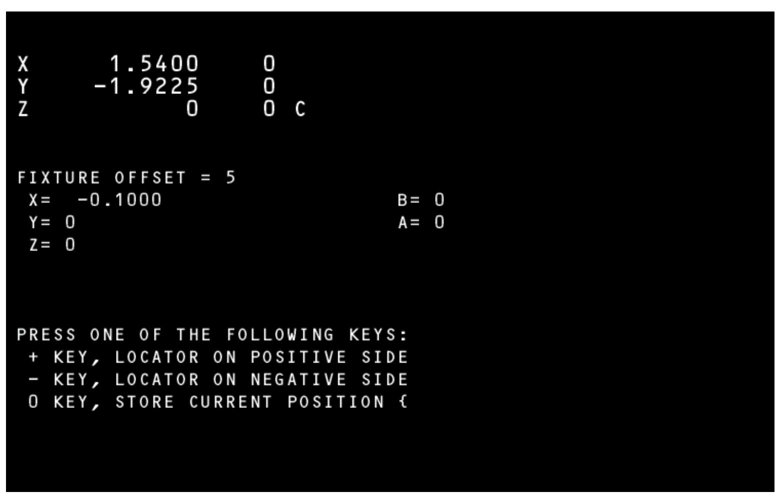


Figure 8-56: Locator Diameter

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.

8.91.8 ITEM 4

FIND CENTER OF CIRCLE

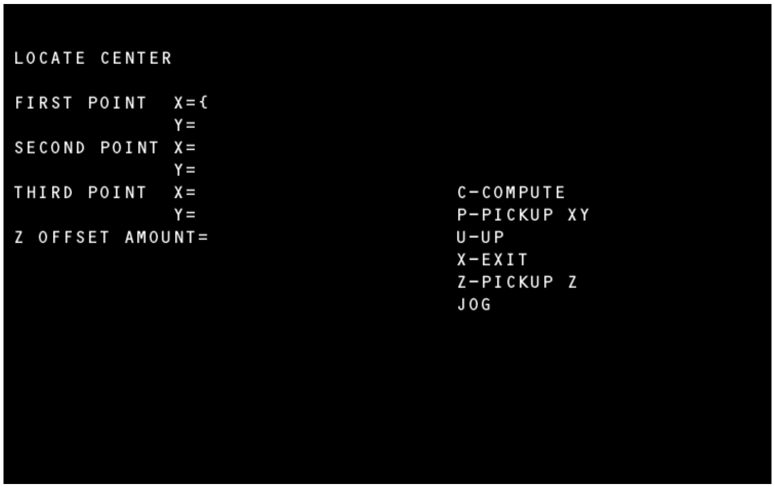


Figure 8-57: Find Center of Circle

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.

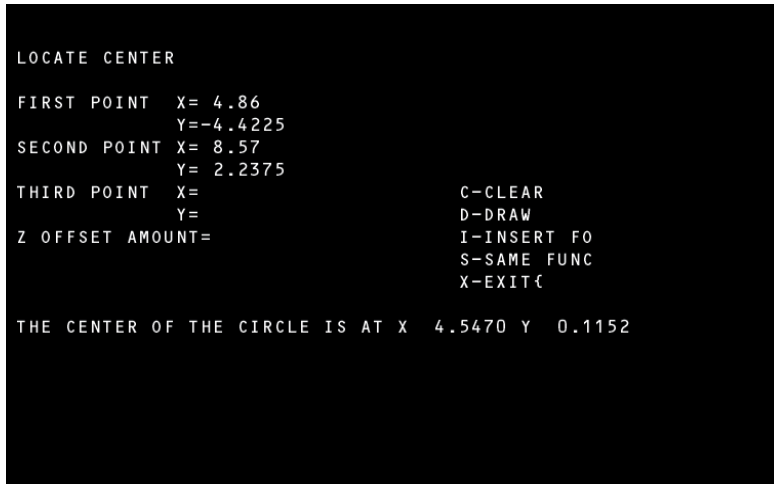


Figure 8-58: Edge Finder

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

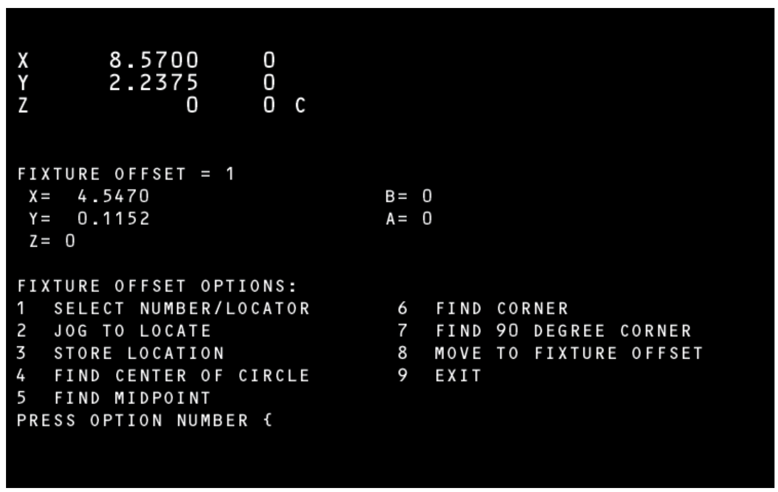


Figure 8-59: Fixture Offset

To exit this option press the X button.

8.91.9 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.

8.91.10 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.

8.91.11 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.

8.91.12 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.

8.91.13 ITEM 9

Exit

Selecting this option exits to the Utilities menu.

8.91.14 OFFSET UTILITY OPTION 3

TEST TS-20 PROBE

This option is used to test the TS- touch probe. (See section 15.0 TOUCH PROBE, MAN-0131.)

8.91.15 OFFSET UTILITY OPTION 4

TEST MP PROBE

This option is used to test the MP probe. (See section 15.0 TOUCH PROBE, MAN-0131.)

8.91.16 OFFSET UTILITY OPTION 5

Pallet Changer

This option displays the pallet changer utility menu. (See section 15.0 TOUCH PROBE, MAN-0131.)

8.91.17 OFFSET UTILITY OPTION 6

Clocks

This option is used to display the clock service utility.

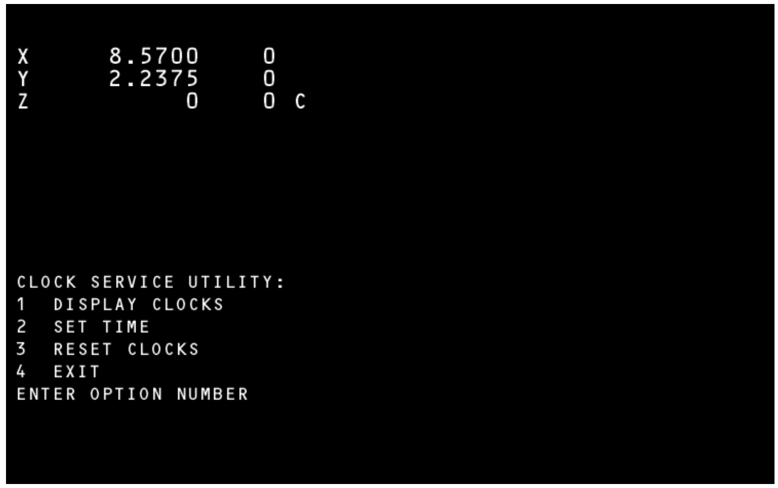


Figure 8-60: Clock Service Utility

8.91.18 ITEM 1

DISPLAY CLOCKS

Select this option to display all current clock settings.

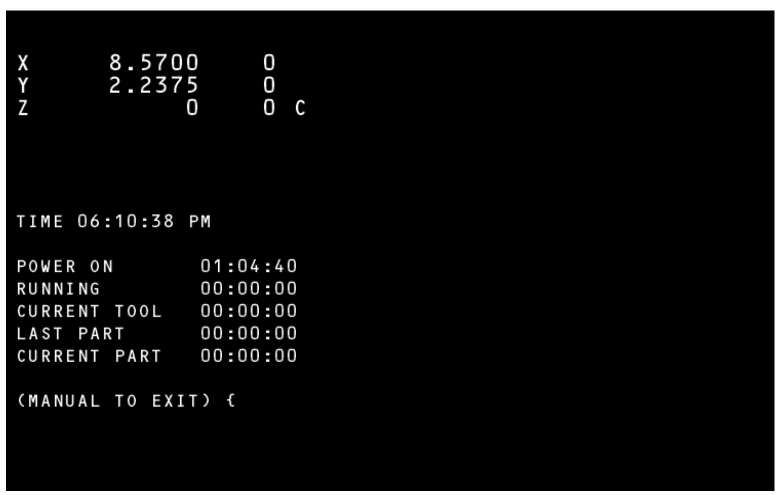


Figure 8-61: Display Clocks

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.

8.91.19 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.

8.91.20 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.

8.91.21 ITEM 4

Exit

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

8.92 TAPE VERIFICATION

8.92.1 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.

9.0 CUTTER RADIUS COMPENSATION

9.1 CUTTER RADIUS COMPENSATION

Cutter Radius Compensation (CRC) is used in a program to allow the operator to alter the path of a cutter.

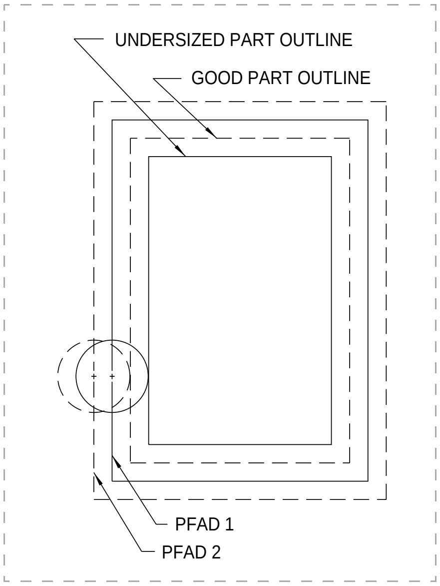


Figure 9-1: Cutter Radius Compensation

EXAMPLE: After cutting the part with path 1, the operator measured the part and determined that the part was undersized. By increasing the amount of the diameter in the tool table and running the program again with another part in the fixture, path 2 cut the part to the correct dimension.

9.1.1 FORMAT 1

The H word in the program will pick up the tool length offset (TLO) and the tool diameter offset. It should be used before using the G41 or G42 codes. The D word can be used to pick up a new diameter, however, it is not necessary in Format 1. If a D word is used, it should appear in the program after the H word. Whatever diameter is picked up by the H word is then overwritten by the D word. It can be used on the same line as the G41 or G42 codes.

M6 T1
G0 G90 S7500 M3 X-2. Y-1.
H1 M7 Z.1 The H will pick up the TLO and the diameter from the tool table

9.1.2 FORMAT 2

The H word in the program will only pick up the tool length offset (TLO). It should be used before using the G41 or G42 codes.

The D word must be used to pick up the tool diameter. It can be used on the same line as the G41 or G42 codes or on any line before the G41 or G42 codes.

M6 T3
G0 G90 S800 M3 X3.641 Y-2.224

H3 D3 M8 Z.1 The H and the D words are used to pick up the TLO and the tool diameter

9.2 G40 - CANCEL CUTTER RADIUS COMPENSATION

See G42.

9.3 G41 - CLIMB CUT (CUTTER LEFT)

See G42.

9.4 G42 - CONVENTIONAL CUT (CUTTER RIGHT)

The G40, G41, or the G42 can be used before or after other codes in a line, without changing what happens. G41 X1. G1 F35. and G1 X1. F35. G41 would both function the same way.

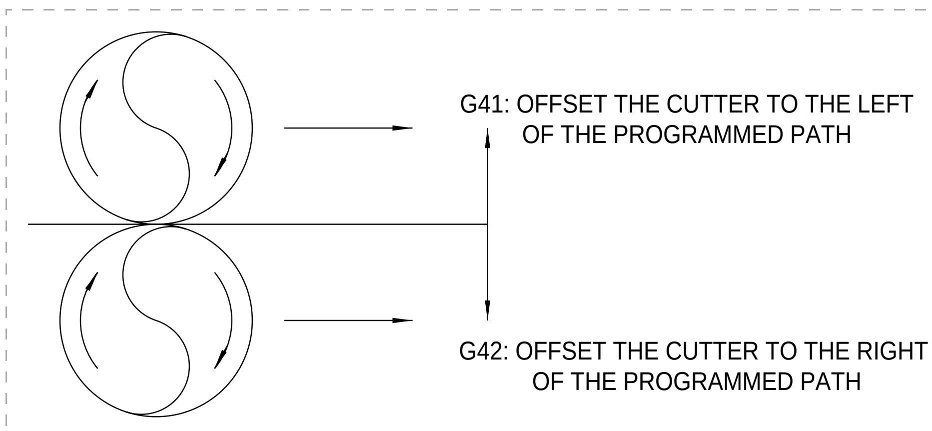


Figure 9-2: Conventional Cut

9.5 CLIMB AND CONVENTIONAL CUTTING

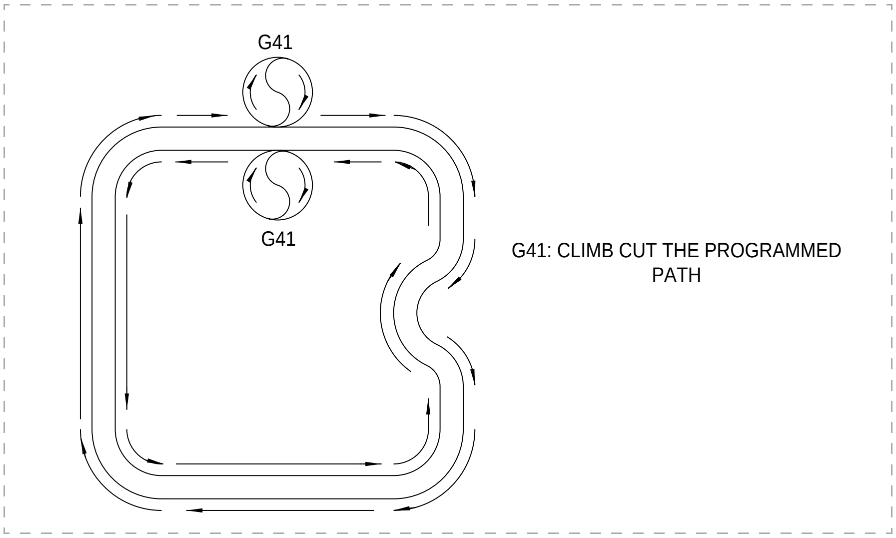


Figure 9-3: **Climb Cutting**

Climb cut the inside of a part by following a general counterclockwise path. Climb cut the outside of a part by following a general clockwise path.

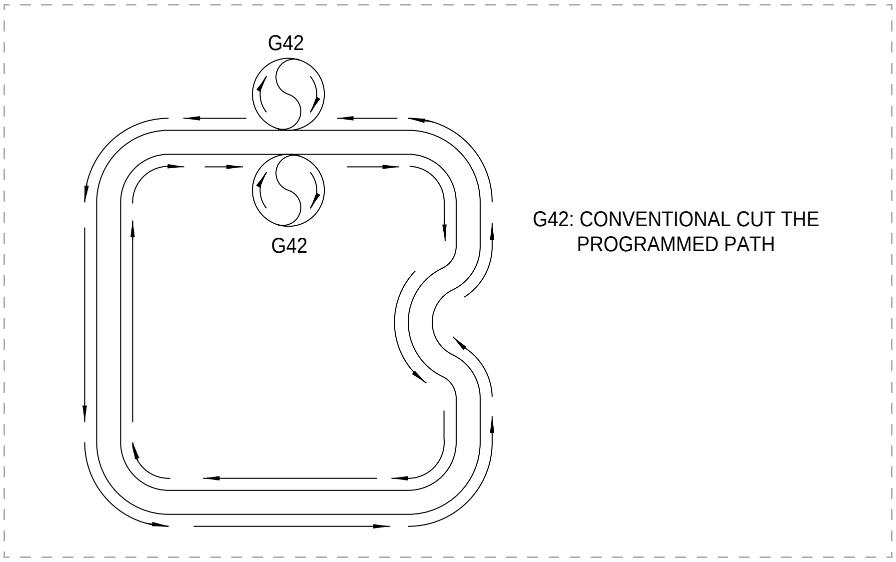


Figure 9-4: **Conventional Cutting**

Conventional cut the inside of a part by following a general clockwise path. Conventional cut the outside of a part by following a general counter clockwise path.

9.6 THE H AND THE D WORD WITH CRC

9.6.1 H AND D WORD USE

Cutter Radius Compensation must be called with offset call. In Format 1 mode, the offset number is selected by the use of an H word. In Format 2, the D word is used. The amount applied will be 1/2 the diameter specified in the table. Positive or negative diameters are allowed. A negative diameter will cause the displacement in the opposite direction. In Format 2 mode, the cutter offset specification in the tool data table may be changed to register diameter or radius

(See SETP section 8.0 COMMANDS, MAN-0131). The D word in the program (Format 2) indicated which offset to use. In Format 1, the use of the D word will override any diameter offset selected by the last H word. The next H word will override the previous D word.

The use of the H99 with the Q word, in Format 1, may be used to specify a specific tool diameter. This diameter value given to the Q word is placed into the tool table location 99. This new value will be used until a new H word or D word is specified.

EXAMPLE:

M6 T1 (.506 E.M.
G0G90S200 M3 E1 X0Y0
H1 Z.1
G1 Z-1.
G41X3.
Y2.
H99 Q.5 This specified a tool diameter of .5
To reverse direction of cut the G41 or G42 mode can be switched without canceling the other code. This can be used when kellering (cutting in both direction to remove material).

9.7 ADVANTAGES OF CLIMB CUTTING

When using climb cuts, the cutter will bend away from the wall being cut. This will automatically leave stock on the wall. Because it will leave stock, this will eliminate the need to program a different roughing pass around the part outline. A roughing pass and a finish pass will result from using the same path two times around the outline. After material has been removed on the first pass, tool pressure is reduced and the tool will not bend on the second pass. Sometimes more than two passes are required depending on the length and type of cutter and on the type of material being cut. For a second pass use the copy command to copy the program, or use subroutines or subprograms.

Simply increasing the diameter of the tool in the tool table will not always work for a roughing pass. Inside radii could be a limiting factor. If the tool diameter is increased to a size larger than an inside radius on the part outline, a “TOOL DIAMETER TOO LARGE” message will appear.

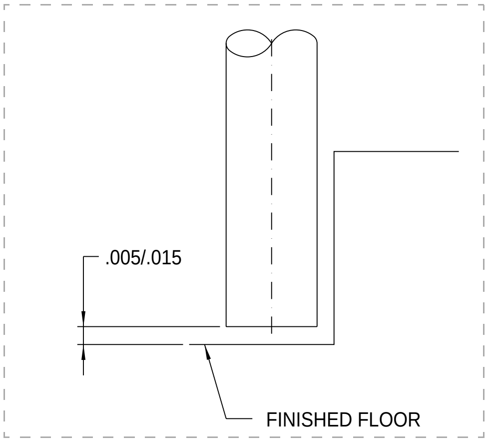


Figure 9-5:

> **NOTE**
> The tool is bending the first time around, so program the tool to be .005 to .015 above any floors (the longer the tool the more it will bend). On the second or final pass, program the tool to cut the floor. The second pass may also use a different feedrate and spindle RPM.
>
> **NOTE**
> Because of the rigidity of a bed style vertical machining center, it is not necessary to program a conventional cut around the part outline and then finish the part with a climb cut. Although this is true for most materials, plastics and hardened materials are the exception.

In most cases climb cutting will allow the tool to last longer. It is not recommended to use climb cutting through a hardened surface, as in a welded, flame hardened, flame cut, hard anodized, or a hot rolled surface.

Climb cutting allows a higher RPM to be used, and along with a higher RPM, higher feed rates (as compared to conventional cutting).

The heat created at the tool and work piece is less with a climb cut than with a conventional cut. With tools as short as possible and a high RPM, the chips will take the heat away from the tool and the work piece.

> NOTE
> When using a higher RPM, use cutters with good chip clearance. Use two or three fluted end mills in place of cutters with four or more flutes.

9.7.1 ADVANTAGES OF CONVENTIONAL CUTTING

Use conventional cutting to “scoop out” or cut through hardened material. If the cutter is deep enough to start the chip in soft material, the chip will continue to form up through the hard material. This will break down the cutter, but it will last longer than using a climb cut.

The heat between the cutter and the work piece is greater with a conventional cut, so a this cut is used for a finish cut for some plastics. It will leave a smoother finish than a climb cut.

In some cases a conventional cut is used to push a flimsy part up against a fixture. If a climb cut was used, the part would not have any support and would bend because of the cutting action.

> WARNING
> Climb cutting causes the cutter to bend away from the wall being cut, and conventional cutting will cause the cutter to bend into the wall being cut.

9.8 GUIDELINES FOR USING CRC

Use CRC (G41, G42) only when cutting a part outline (part path program). Program motion of the center of the tool when cutting areas other than walls, using G40 (CRC cancel). When programming the motion of the center of the cutter (tool path program) using G40, the control will not alter the path.

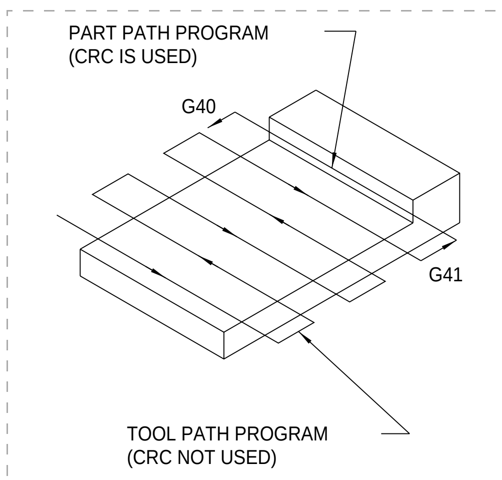


Figure 9-6: Using CRC

CRC (G41, G42) should only be used when cutting a part outline. Program the outline of the part (using the print dimensions). The control will compensate for the radius of the cutter being used and cut to leave the programmed part outline.

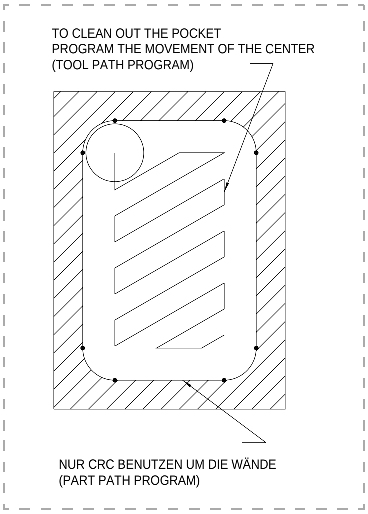


Figure 9-7:

> **NOTE**
> Cutter Radius Compensation must be canceled prior to locating the center line of the tool to a specific point.

9.9 GENERAL RULES

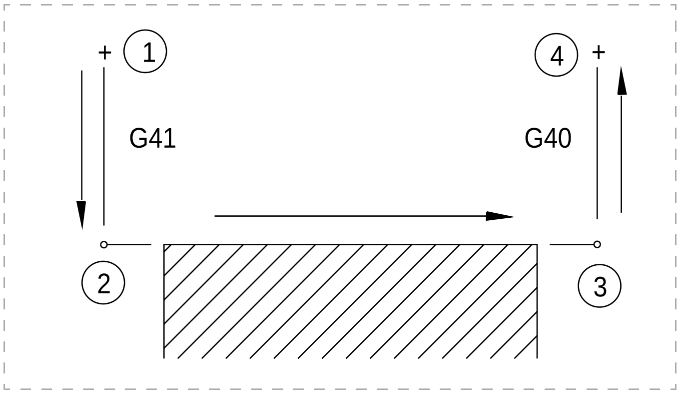


Figure 9-8: General Rules

1. G90 X-.4 Y.4 Position the tool, at least the tool radius away from the wall to be cut
2. Y0 G41 G1 F10. With the move to the wall, use G41 to apply CRC
3. X3.01 Move along the wall

4. Y.4 G0 G40 With the move away from the wall being cut, use a G40 to cancel comp

> **NOTE**
> The distance of the move up to the wall and away from the wall must be greater than or equal to the radius of the cutter.

9.9.1 EXAMPLES OF APPLYING AND CANCELING CRC

These examples represent part outlines and are intended to give general ideas on applying and canceling cutter radius compensation.

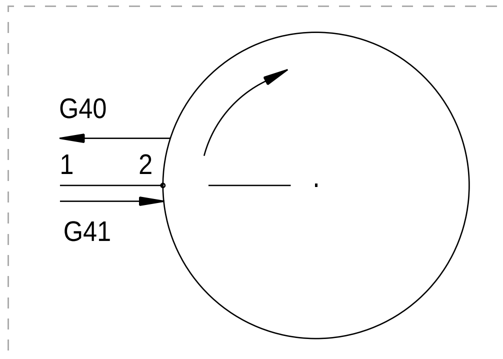


Figure 9-9: Applying & Canceling CRC (1)

1. Position the tool, at least the tool radius away from the boss.
2. Apply CRC along with the move up to the boss.
3. Program the circular move.
4. Cancel CRC along with the move back to the first position.

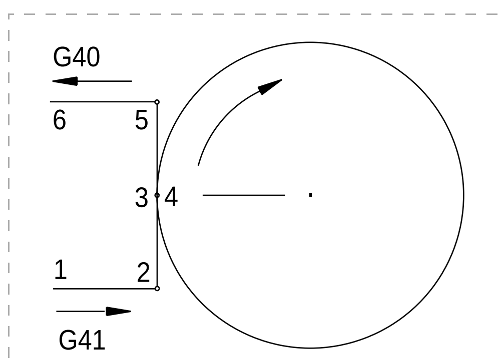


Figure 9-10: Applying & Canceling CRC (2)

1. Position the tool, at least the tool radius away from the line from 2 to 3.
2. Apply CRC along with the move from position 1 to 2.
3. Complete the entire path.
4. Cancel CRC along with the move from position 5 to 6.

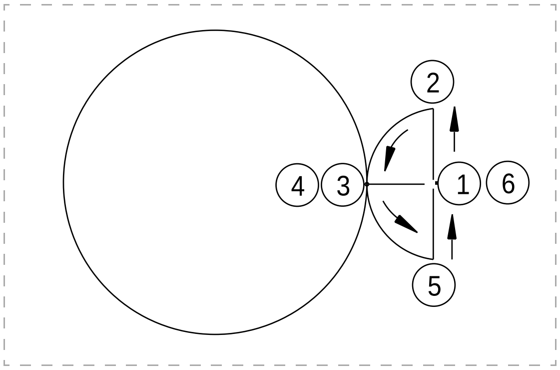


Figure 9-11: Applying & Canceling CRC (3)

1. The radius of the lead in lead out radius must be larger than the radius of the tool to be used.
2. Apply CRC along with the move from position 1 to 2.
3. Complete moving around the path.
4. Cancel CRC along with the move from position 5 to 6.

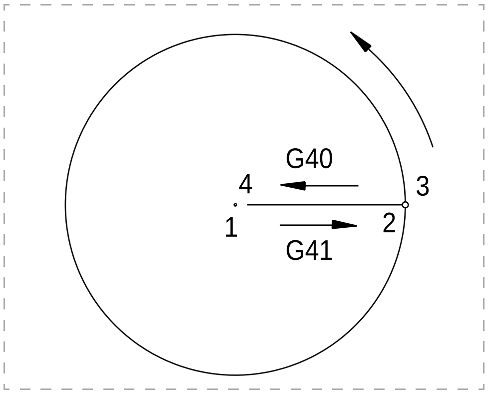


Figure 9-12: Applying & Canceling CRC (3)

1. Apply CRC with the move from position 1 to 2. Position 1 must be at least the radius of the tool to be used away from the edge of the circle to be cut. Position 1 does not need be at the center of the circle.
2. Cut counter clockwise around the circle.
3. Cancel CRC along with the move away from the circle from position 3 to 4.

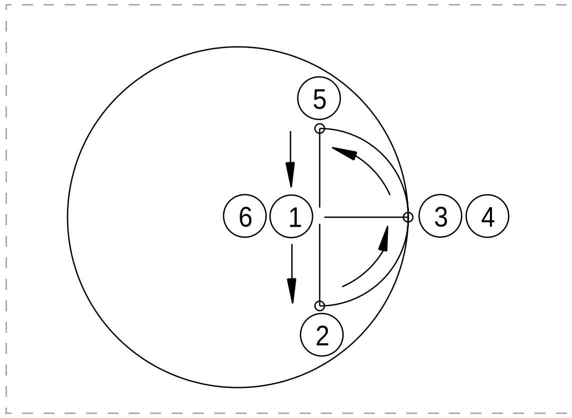


Figure 9-13: Applying & Canceling CRC (4)

1. The radius of the lead in lead out radius must be larger than the radius of the tool to be used.
2. Apply CRC along with the move from position 1 to 2.
3. Complete moving around the path.
4. Cancel CRC along with the move from position 5 to 6.

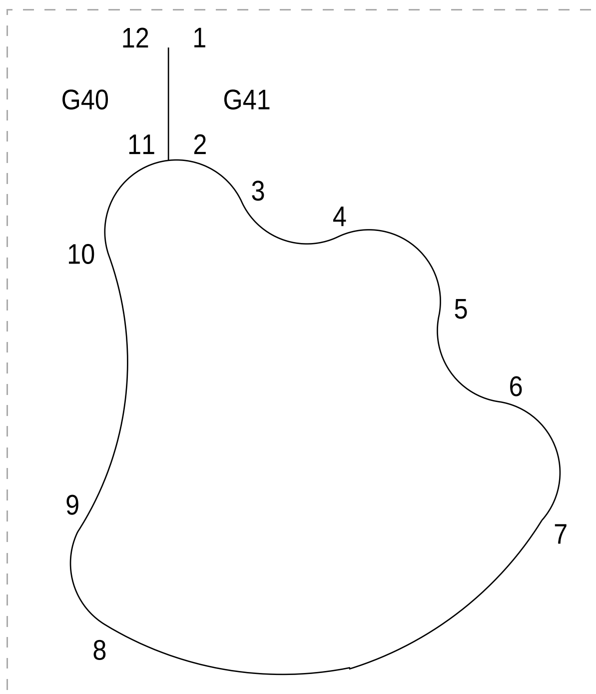


Figure 9-14: Applying & Canceling CRC (5)

1. Apply CRC along with the move up to the part wall (from position 1 to 2).

2. Move around the part.
3. Cancel CRC along with the move away from the part (from position 11 to 12).

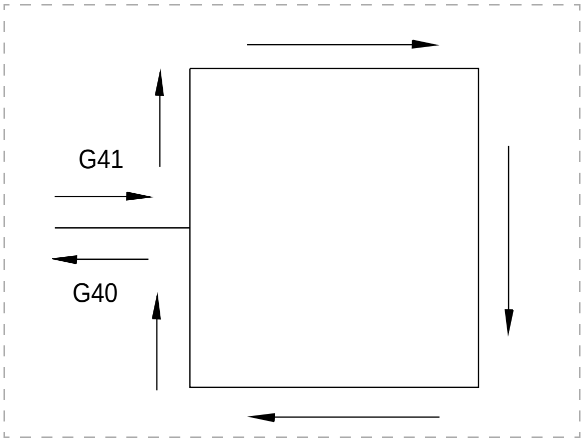


Figure 9-15: Applying & Canceling CRC (6)

1. Apply CRC along with the move up to the part wall. This move must be at least the radius of the tool to be used away from the part.
2. Move in a general clockwise direction around the outside of the part (for climb cut).
3. Cancel CRC along with the move away from the part. Again, this move must be at least the radius of the tool to be used away from the part.

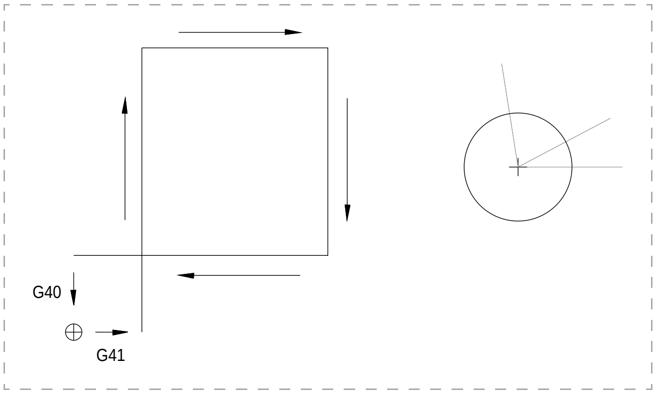


Figure 9-16: Applying & Canceling CRC (7)

1. Apply CRC along with a move up to an extended wall from the part. This move must be at least the radius of the tool to be used away from the part.
2. Move in a general clockwise direction around the outside of the part (for climb cut) to an extended line off the wall of the part.
3. Cancel CRC along with the move away from the extended line. Again, this move must be at least the radius of the tool to be used away from the extended line.

9.10 O-RING GROOVE EXAMPLE

In an O-ring groove, consider both circles of the groove as separate walls.

1. Start at the midpoint between the walls.
2. Apply CRC (comp) moving to the first circle.
3. Cut the circle.
4. Cancel comp moving back to the midpoint.

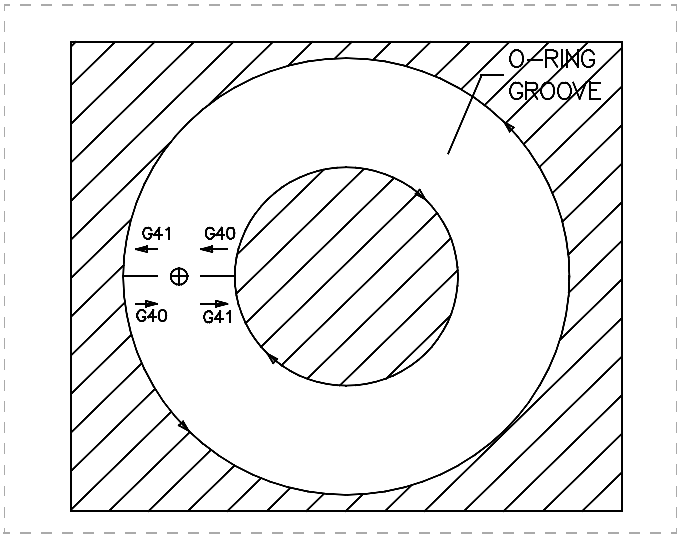


Figure 9-17: O-Ring Groove

Following the general rule, always cancel CRC when leaving a wall. When going from one wall to the other, cancel comp moving to the midpoint, then apply comp again moving to the other wall.

5. Apply comp moving to the second circle.
6. Cut the circle.
7. Cancel comp moving back to the midpoint.

9.11 PERPENDICULAR RULE

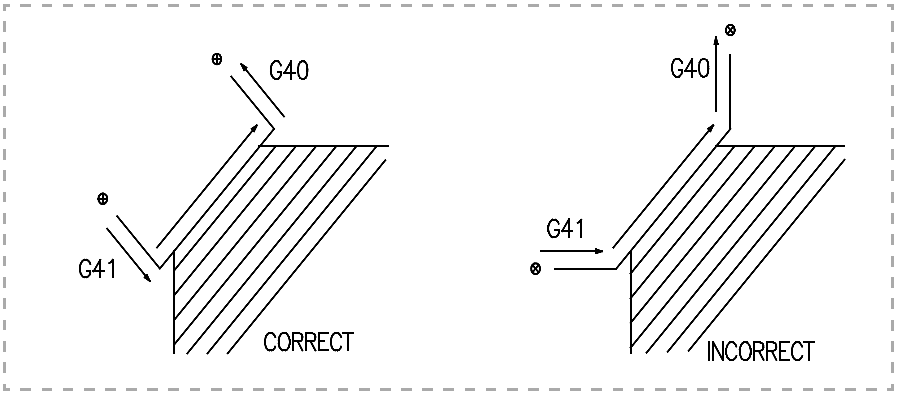


Figure 9-18: Perpendicular Rule

It is best to apply and cancel comp using moves perpendicular to the wall. When perpendicular moves are used, the cutter will follow the programmed move. Otherwise, the cutter will not follow the move to or away from the wall.

Moving perpendicular to and from a wall is best. When comp is on, the tool will remain perpendicular (tangent) to the wall it is touching. Beginning and ending the move perpendicular will keep the tool perpendicular for the entire programmed path.

Comp can be applied with a move that is not perpendicular; however, the actual move the tool will make when it is moving up to and away from the wall cannot be predicted. Even though it is possible to turn comp on, with a move that is not perpendicular the wall, this method is not recommended.

Any line, at any angle, that extends through the center of a circle is considered perpendicular to the circle. When starting comp such a line is recommended for contoured walls.

9.12 FILLET RADII AND STEP DOWNS

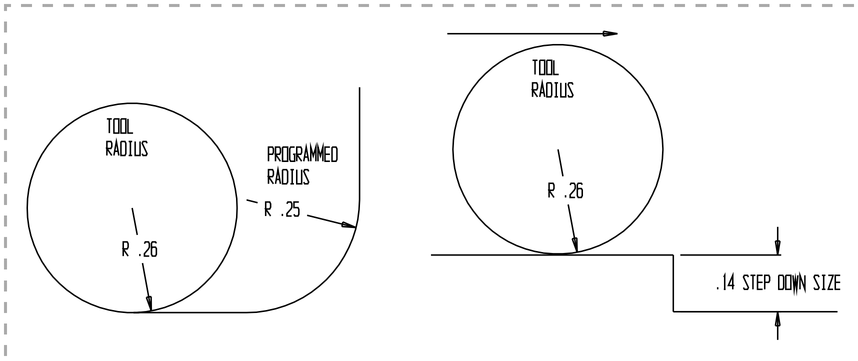


Figure 9-19: Fillet Radii and Step Downs

All features on a print have tolerances. Inside corner radii (“fillet” or “blend” radii) usually have more tolerance than other features. Take advantage of this and program inside corner radii larger than the radius of the tool to be used for cutting the part. Also remember, when a larger tool diameter is going to be used for a roughing pass, program fillet radii larger than the intended tool radius. In the drawing above, the tool radius in the tool table is too large for the programmed fillet radius. In the drawing below, the programmed fillet radius is larger than the tool radius in the tool table. The operator can increase the tool radius in the table up to the smallest inside fillet radius.

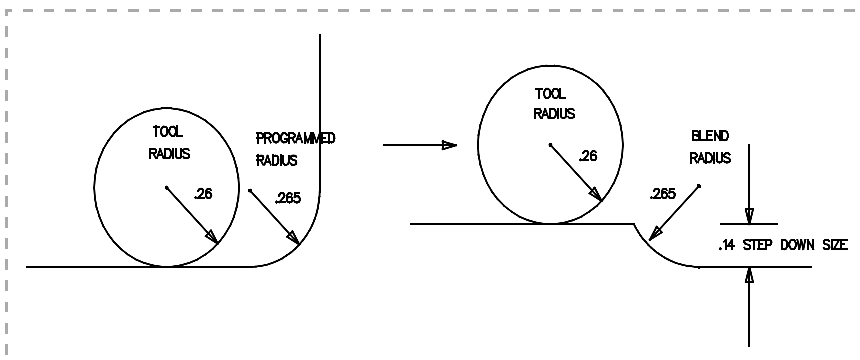


Figure 9-20: Fillet Radius Larger than Tool Radius

In the drawing above, the step down distance is smaller than the radius of the tool. The cutter will not be able to step down, and a TOOL DIAMETER TOO LARGE message will appear. The picture below shows how to program a fillet radius from the corner of the step and tangent to the bottom wall. Again make the fillet radius large enough to accept a radius for a roughing pass.

> **NOTE**
> It is always better to program a fillet radius between two intersecting features, rather than to leave them as steps and intersections.

9.12.1 PROGRAM EXAMPLE

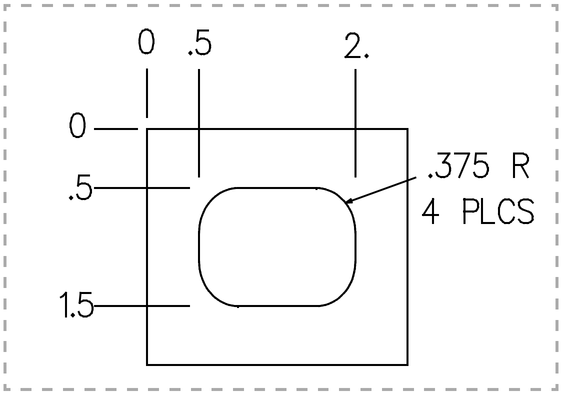


Figure 9-21: **Program Example**

The print above shows the inside radii as .375 in four places. A .75 end mill could be used to form the inside radii; however, if the end mill is used to form the inside radii, the cutter will bend into the corners and chatter. By programming an inside corner radius larger than the radius of the tool to be used, the problem of bending into the corner and chattering will be eliminated. Drilling out the corners, slowing down, speeding up or dwelling in the corners will not work!

Sometimes the tolerances do not allow the programmer to program an inside corner radius larger than a common sized end mill. For example, a print may allow the corner to be no larger than .250. Using a .5 end mill would cause the cutter to form the inside radius. When forming an inside radius with the radius of the cutter is not desirable, use a .375 cutter or reground .5 end mill and contour the .250 fillet radius.

NOTE
Never use a cutter to form an inside corner radius and expect good results on the part!

NOTE
Always contour an inside corner radius!

EXAMPLE:

```
N1 O1 (CRC RECTANGULAR WINDOW EXAMPLE
N2 L100 (POCKET
N3 Y-1.12
N4 X.875 Y-1.5 I.375 J0 G3
N5 X1.625
N6 X2. Y-1.125 I0 J.375
N7 Y-.875
N8 X1.625 Y-.5 I-.375 J0
N9 X.875
```

N10 X.5 Y-.875 I0 J-.375
N11 M17
N12 M30
N13 (TOOL #1, 3/4 2FL EM (CRC)
N14 G0 G90 S5000 M3 E1 X.875 Y-.875
N15 H1 D1 M7 Z.1
N16 G1 Z-.4 F10. G8
N17 X.5 G41
N18 F35.
N19 L101
N20 F45.
N21 L101
N22 X.875 G40
N23 M5 M9
N24 G0 H0 G90 Z0
N25 M6 T2

The subroutine includes only the moves around the wall of the window. Programming in this manner allows the subroutine to be called up two times for two passes around the wall. The move up to the wall at N17 turns comp on, and is using the feedrate from N16. A new feed at N18 is programmed and then the sub is called one time at N19. A feedrate change is made at N20, then the sub is called again at line N21. When running this program, the moves will look as if they are continuous from the first time around to the second time around. This is because all of these moves are preprocessed. The move away from the wall, while turning comp off, is made at line N22.

9.13 CORNER ROUNDING

When cutter radius compensation is applied, it can use two different methods to move around a corner: rolling and intersectional.

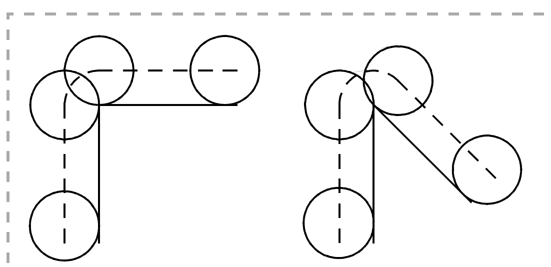


Figure 9-22: Corner Rounding

9.14 M96 CANCEL INTERSECTIONAL CRC (ROLLING)

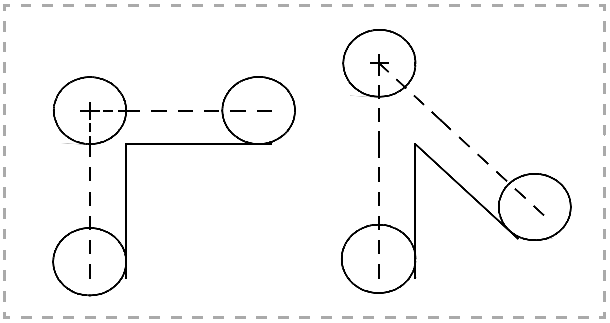


Figure 9-23: Rolling

9.15 M97 INTERSECTIONAL CRC

The M96 or M97, as default codes, can be selected when using the SETP command.

The M96 mode of CRC is more commonly used. In most cases it is also the safer mode. Using this mode insures that the tool will always be touching the programmed walls. An M97 allows the tool to move away from the programmed walls, where it might gouge some other programmed feature. These two codes are named by the way they move around any corner: tangentially for M96 and intersectionally for M97.

9.16 WHEN TO USE M96 AND M97

Each code produces a different type of corner on the floor; however, both will cut a mitered corner on all top edges.

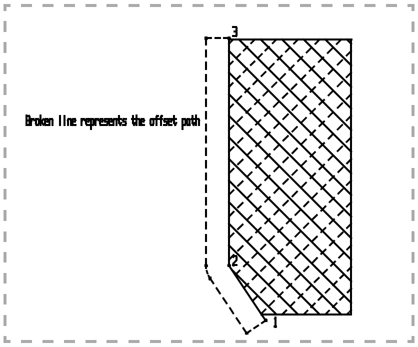


Figure 9-24: Using M96 and M97

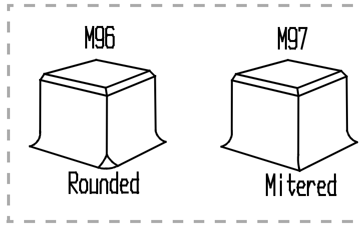


Figure 9-25: Rounded/Mitered Corners

M96 produces a rounded corner on the floor. M97 produces a mitered corner on the floor.

> NOTE
> Some prints will specify the type of corner needed on the part. In most cases the type of rounding used will only affect the part visually, not functionally.

M96 and M97 are default codes, and can be established by using the SETP command. M96 is more commonly used as default because M97 causes the tool to bypass a corner which may cause the tool to bump into another portion of the part.

If a part has steps, and blend radii were not used to move up and down the steps, use M97. The control will accept the program better than in the M96 mode. As mentioned earlier in regards to fillet radii and step downs, it is better to always program a fillet radius between two intersecting features, rather than to leave these features as steps and intersections. If fillet radii are programmed, use the M96 mode.

Grooves, slots, and O-ring grooves should use M96. This is most important when the diameter of the tool is almost equal to the width of the slot or groove.

9.17 CRC & Z AXIS MOVES

While CRC is in effect, a Z move can be made to move to a new Z level or to execute a helical move. The control will “look ahead” of Z moves and compensate the cutter for the next X or Y move.

M6 T1 (TOOL #1, 1/2 DIA. FINISH EM (CRC)
G0 G90 S6000 M3 E1 X24.5 Y.3
H1 M7 Z-.5
X24.2 G41
Y-1.02 G1 F35.
Z-.2 Here is a move to a new Z level with CRC still in effect
Y-1.9 G5
Y-2.02
X24.5 G0 G40

9.18 APPLYING COMPENSATION WITH A Z MOVE

CRC can be applied along with a Z move. When comp is applied with a Z move, the cutter will offset tangent to the next X or Y axis move. The next line for the control to move tangent to can be a linear move or a circular move.

> NOTE
> Format 1 and Format 2 apply the offset differently.

EXAMPLE:

FORMAT 1

A G41/G42 in the same line with a Z move will offset the cutter before making a Z minus move and after a Z plus move.

M6 T1 (TOOL #1, 3/4 DIA. EM (CRC)
G0 G90 S1500 M3 E1 X.6 Y-4.6
H1 Z.1 M7
Z-.52 G41   Here the cutter offsets first, then the Z move is made
X0 Y-3.7 G1 F15.
Y.02
Z.1 G0 G40 Here the Z move is made first, then comp is canceled

> NOTE
> The example program above will operate differently in Format 2. See the next example program for Format 2.

EXAMPLE:

FORMAT 2

In Format 2 the Z move and the offset move will be made together.

M6 T1 (TOOL #1, 3/4 DIA. EM (CRC)
G0 G90 S1500 M3 E1 X.6 Y-4.6

H1 Z.1 M7
Z-.52 G41 Here the offset and Z axis move at the same time
X0 Y-3.7 G1 F15.
Y.02
Z.1 G0 G40 Here the Z axis moves and the offset is canceled at the same time

EXAMPLE: FORMAT 1 & 2

The sample program below will operate in the same way for Format 1 and Format 2.

M6 T1 (TOOL #1, 3/4 DIA. EM (CRC)
G0 G90 S1500 M3 E1 X.6 Y-4.6
H1 Z.1 M7
G41 Here the cutter offsets
Z-.52   Then the Z move is made
X0 Y-3.7 G1 F15.
Y.02
Z.1 G0   Here the Z move is made
G40 Then comp is canceled

9.19 APPLYING COMPENSATION WITH A Z MOVE ON A CIRCLE

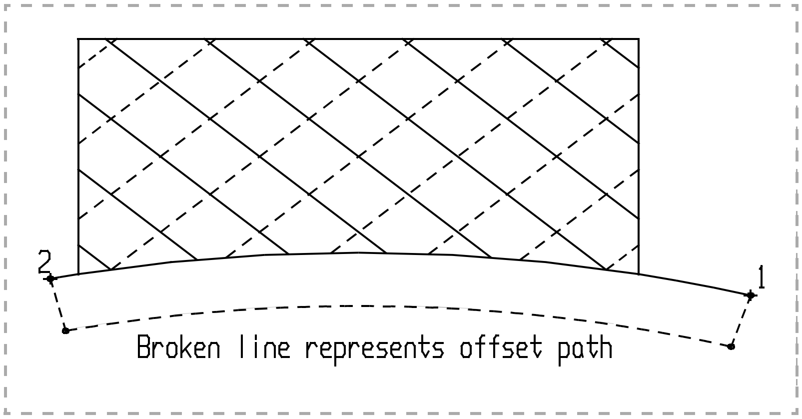


Figure 9-26: **Cutter Offset**

EXAMPLE:

FORMAT 1 & 2

The sample program below will operate in the same way for Format 1 and Format 2.

M6 T1 (TOOL #1, 3/4 DIA. EM (CRC)
G0 G90 S1500 M3 E1 X2.4 Y-1.2
H1 Z.1 M7
G41 Here the cutter offsets perpendicular to the circle
Z-.52 Then the Z move is made
X-1. Y-1.1225 I-1.4 J-4.8 G3 Here the circle cut is made
Z.1 G0 Here the Z move is made
G40 Then comp is canceled

9.19.1 COMPENSATION EXAMPLE

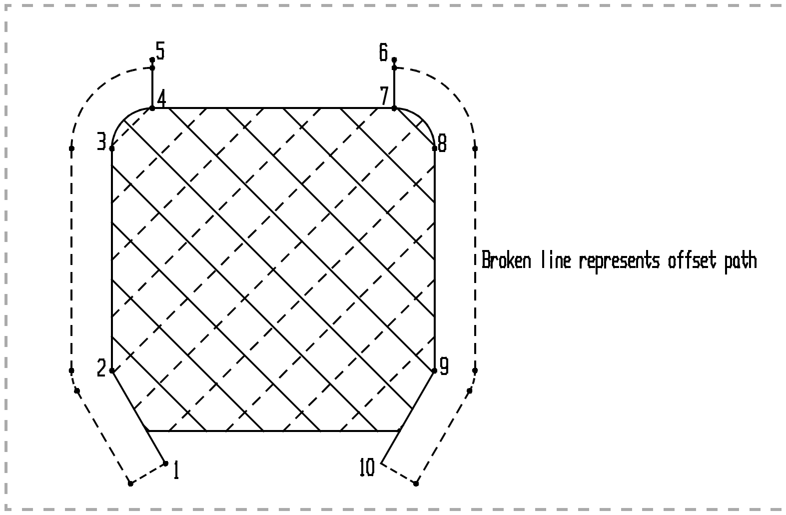


Figure 9-27: Compensation Example

EXAMPLE:

FORMAT 1 & 2

The sample program below will operate in the same way for Format 1 and Format 2.

M6 T1 (TOOL #1, 1/2 DIA. EM (CRC)
G0 G90 S5000 M3 E1 X.332 Y-2.2 This moves to point one
H1 Z.1 M7
G41 Comp is applied perpendicular to the next move
Z-.27 Z moves down
X0 Y-1.625 G1 F30.
Y-.25
X.25 Y0 I.25 G3
Y.3 G40 G0 Comp is canceled along with the move away from the wall being cut
Z.1
X1.75 This moves to point six
Z-.27
Y0 G41 Comp is applied with the move up to the wall to be cut
X2. Y-.25 J-.25 G3
Y-1.625
X1.668 Y-2.2
Z.1 G0 Z moves up perpendicular to position ten
G40 Comp is canceled and positions over point ten

10.0 ERROR MESSAGES

10.1 ERROR MESSAGES

10.1.1 ABSOLUTE MODE REQUIRED AT N =

The CNC must be in the G90 (absolute) mode during all probe functions. Probe function L9101 is programmed in the G91 (incremental) mode.

10.1.2 AMPLIFIER FAULT ON AXIS

This occurs at power on if the control does not detect an axis that should be operational. This is often seen when the A axis is not in use and the dummy termination plug has been installed. If the dummy termination plug is installed, this message can be disregarded.

If this message appears for any of the other axes (X, Y, or Z), then power off to check the motor overload relays. Power on and if the message appears again, call the service department, noting the error message number. The motor overload relays can be recognized by their white reset buttons. They are located in both control enclosures. The white button must be pressed to reset a relay.

This message also appears when the EMERGENCY STOP button has been pressed. The operator must release the EMERGENCY STOP button and press the JOG button to reset the amplifiers. (Depressing the EMERGENCY STOP button disconnects the axis amplifiers. This is why this message appears.)

10.1.3 ARM MUST BE LEFT

Before a pallet change can occur, the arm must be completely to the left with the pallet on the table. The feed back on the arm is being read at this time.

10.1.4 ARRAY EXCEEDS ALLOWABLE NUMBER

This message will appear when the number on the array exceeds the expected amount. This amount varies from variable to variable. For example, H(99) is acceptable, H(102) is not.

10.1.5 ATC FAILURE

Check to see if the Z axis alignment was properly set at cold start. The automatic tool changer has failed to respond to an M6 or the TC,1 command. Try a power off and power on. If this does not help check the fuses F5, F6, F7, and F8 to see if they are burned out. Circuit breakers 1 and 2 (if they are present) should be pressed to see if they have been tripped.

> WARNING
> Power off the machine before checking fuses, relays, and/or circuit breakers.

If this does not help, call the service department.

If this message appears with the turret in the extended position, it may be an indication that the sensor for turret extension is faulty; call the service department.

10.1.6 ATC TURRET FAILURE

Check to see if the Z axis alignment was properly set at cold start. Then check to see if anything is caught between the sheet metal guard and the turret. If not, this could be an indication that the sensor for turret rotation is faulty.

The fuses F5 and F6 should be checked to see if they are burned out. Circuit breaker 2 (if it is present) should be pressed to see if it has been tripped.

> WARNING
> Power off the machine before checking fuses, relays, and/or circuit breakers.

10.1.7 ATC WILL NOT MOVE TO POSITION

Check to see if the Z axis alignment was properly set at cold start. An attempt is made to make a tool change and something prevents the turret from going all the way out to the spindle. Check the tracks that allow the turret to slide out to the spindle. Sometimes chips stick to the track and prevent the rollers from rolling on the track.

When this message appears use the command SETCS then the command HO which will bring the tool to the Cold Start position. It is important to visually check the axis indicator markers to see if they are aligned; if not, jog the table until the markers are aligned. Use the command CS, then answer Yes (Y) to the “move to home?” question. This will bring the tool to the part’s home position and enable the operator to begin running the program in AUTO again. It is also possible that the turret extended sensor is faulty.

Fuses F5, F6, F7, and F8 should be checked to see if they are burned out. Circuit breakers 1 and 2 (if present) should be pressed to see if they have been tripped.

> WARNING
> Power off the machine before checking fuses, relays, and/or circuit breakers.

10.1.8 ATC AMBIGUOUS TURRET POSITION, VERIFY POCKET # WITH SETTO,# OR, TURRET LOCATION LOST, RESET TURRET LOCATIONS WITH SETTO,#

The Tool Count sensor indicates the Turret is located between Buckets, or has been interrupted. Move Turret to next Bucket position, and apply SETTO,# where # is the Bucket number in the Bucket ready position.

1. Another system may have faulted while Turret was moving or the ATC was communicating. Correct the fault, and reset Turret locations with SETTO,#.
2. Check the Door Interlock System.
3. Check for power interruption to the Turret Motor, fuses, and circuit breakers.
4. Check the Turret Motor for liquid contamination, or inoperative brake.
5. Check the Tool Count Sensor.

10.1.9 ATC AMBIGUOUS BUCKET POSITION, VERIFY POCKET # WITH SETTO,# OR, BUCKET NOT UP/ DOWN, RESET TURRET LOCATIONS WITH SETTO,#

The Tool Up or Tool Down sensor indicates that Bucket is out of position, and the Turret Locations may be lost. Correct Bucket Up/Down problem, and apply SETTO,# where # is the Bucket number in the Bucket Ready position.

1. Check Tool Up/Tool Down reed switches on Bucket Cylinder.
2. Check for both air valves on at the same time.
3. Check for inadequate air supply.

10.1.10 ATC ARM IS NOT IN HOME POSITION

The Stopping Sensor or the Arm at Home Sensor indicates that the Arm is not in its Home Position, and must be before the machine can continue.

1. Check Door Interlock system.
2. Check Head Position Sensor and Head Position.
3. Check for power interruption to Arm Motor, fuses, and circuit breakers.
4. Arm Motor may need to be moved manually.

10.1.11 ATC ARM SOLENOID SENSOR/HOME SENSOR FAILURE OR, ARM DID NOT MOVE FROM HOME

The Arm at Home Sensor indicates that the Arm has not moved away from ATC home position.

1. Check Door Interlock system.
2. Check Head Position Sensor and Head Position.
3. Check for power interruption to Arm Motor, fuses, and circuit breakers.

10.1.12 ATC BUCKET UP SOLENOID/SENSOR FAILURE OR, BUCKET DID NOT MOVE UP

The Tool Up Sensor has not reported a successful move up of the Bucket in preparation for rotating the Turret.

1. Check the lower reed switch on the Bucket Cylinder.
2. Check for inadequate air supply.
3. Check air valve.

10.1.13 ATC BUCKET DOWN SOLENOID/ SENSOR FAILURE OR, BUCKET DID NOT MOVE DOWN	The Tool Down Sensor has not reported a successful move down of the Bucket in preparation for exchanging the tools. 1. Check the upper reed switch on the Bucket Cylinder. 2. Check for inadequate air supply. 3. Check air valve.
10.1.14 ATC COMMUNICATION ERROR WITH 1330 CARD	A communication timeout has occurred between the CNC CPU and the DATC Controller (1330). 1. Route ribbon cable connecting the 1470 to 1330 away from fans. 2. If error occurs on power-up, there may not be an actual problem.
10.1.15 ATC INTERLOCK IS ON	An inhibited ATC function has been attempted while interlocked.
10.1.16 ATC TOOL CLAMPING FAILURE OR, ARM DID NOT ARRIVE AT SPINDLE	The arm at Spindle Sensor or the Stopping Sensor has indicated that the Arm has not arrived at the Spindle. 1. Check for Arm Plunger latching problem. 2. Check Door Interlock system. 3. Check Head Position Sensor and Head position. 4. Check for power interruption to Arm Motor, fuses, and circuit breakers.
10.1.17 ATC TOOL UNCLAMPING FAILURE OR, ARM DID NOT LEAVE SPINDLE	The Arm at Spindle Sensor or the Stopping Sensor has indicated that the Arm did not leave the Spindle. 1. Check Door Interlock system. 2. Check Head Position Sensor and Head position. 3. Check for power interruption to Arm Motor, fuses, and circuit breakers. 4. Check for tool release failure.
10.1.18 ATC TOOL ARM OBSTRUCTED OR, ARM DID NOT ARRIVE AT SPINDLE	The Arm travel was interrupted before arriving at the Spindle. 1. Check Door Interlock system. 2. Check Head Position Sensor and Head position. 3. Check for power interruption to Arm Motor, fuses, and circuit breakers.

10.1.19 ATC TOOL SENSOR FAILURE OR, ARM NOT AT SPINDLE

The Stopping Sensor and the Arm at Spindle Sensor both indicate that the Arm has not fully positioned at the Spindle.

1. Check Door Interlock system.
2. Check Head position Sensor and Head position.
3. Check for power interruption to Arm Motor, fuses and circuit breakers.
4. Check for proper Spindle orientation.
5. Check for Tool alignment in ATC grippers.

10.1.20 CAROUSEL SOLENOID FAILURE (OR SENSOR) OR, TURRET HAS NOT ROTATED

The Tool Count Sensor indicates that the Turret Motor has not rotated the Turret as expected within the allotted time.

1. Check Door Interlock system.
2. Check for power interruption to Turret Motor, fuses, and circuit breakers.
3. Check Tool Count Sensor and alignment.

10.1.21 DRAWBAR SENSOR FAILURE OR, DRAWBAR NOT CLAMPING TOOL

The Drawbar Cylinder Sensor indicates that the Drawbar is still in release position, and has not retracted from the Spindle.

1. Check Drawbar mechanism for binding or no lubrication.
2. Check Drawbar Cylinder Sensor alignment.

10.1.22 DRAWBAR SOLENOID FAILURE (OR NO AIR) OR, DRAWBAR NOT RELEASING TOOL

The Drawbar Cylinder Sensor indicates that the Drawbar could not fully release the tool from the Spindle and the Drawbar Cylinder Piston has not moved down full stroke.

1. Check for inadequate air supply or inoperative air valve.
2. Check Drawbar Cylinder Sensor alignment.
3. Check for broken Drawbar Cylinder Piston.

10.1.23 ATTEMPTED DIVISION BY ZERO

In a macro statement, a division by zero was attempted.

10.1.24 ATTEMPTED SQR OF A NEGATIVE NUMBER

In a macro statement, a square of a negative number was attempted.

10.1.25 ATTEMPT TO CHANGE CRC SIDE WITHOUT G40 AT N =	This message will occur when attempting to switch from G41 to G42, or vice versa, on a linear move. However, switching from one to the other is permitted during circular interpolation.
10.1.26 ATTEMPT TO CHANGE TOOL WHILE IN CRC MODE, N =	This message occurs if a G41 or G42 is used in the program and a G40 code does not appear before an M6 code. The G40 cannot be on the same line with M6. An M6 will not cancel CRC.
10.1.27 ATTEMPT TO SWITCH PLANE DURING CRC	G40 must be in effect (CRC must not be on) when changing planes. The default plane selection is G17. If CRC is in effect and a code to change to another plane (G17, G18, G19) is used, this message will appear in the block where the plane selection was called. G40 must be in effect (CRC must not be on) when changing planes.
10.1.28 AXIS CONTROLLER DOES NOT RESPOND DURING POWER UP SEQUENCE	This an indication that there is a communication problem between the CPU (1400) board and the axis controller (1010). Note error number and call the service department.
10.1.29 AXIS CONTROLLER DOES NOT RESPOND TO NC	This an indication that there is a communication problem between the CPU (1400) board and the axis controller (1010). Note error number and call the service department.
10.1.30 AXIS DATA TRANSFER IS STALLED	This message is the result of a data transfer problem between the 1030 card and a 1010 card. This message does not represent a serious problem and should only be reported to the service department if it is persistent. Please note error number.
10.1.31 AXIS DOES NOT RESPOND	This could be a problem with the 1010 card for the stated axis. Try a power off and a power on. If this does not help, call the service department.
10.1.32 AXIS FAULT(S) AS FOLLOWS	This appears when the EMERGENCY STOP button is pressed, a motor overload relay is tripped, or the machine places itself into the Emergency Stop mode. If any message or error number is presented on the screen, record this message and number and report them, if necessary, to the service department. Release the EMERGENCY STOP button if necessary, then press the JOG button which will reset the machine. If the machine does not reset, call the service department. If the EMERGENCY STOP button is not depressed, then power off the machine and press the reset buttons of the motor overload relays to see if they are tripped.

The motor overload relays can be recognized by their white reset buttons. They are located in both control enclosures. The white button must be pressed to reset a relay.

If the machine stalls for one reason or another, the control places itself into the Emergency Stop mode. Press the JOG button and jog the tool out of the stalled position.

An axis fault also occurs during an emergency stop because the control removes the power to each of the axis servo amplifiers as a safety feature. The control then analyzes the situation and, as a part of the report, shows a servo amplifier fault, a result of having removed power to the amplifiers.

10.1.33 AXIS OVERFLOW

The axis controller detected an overflow situation. Power off the machine at the main power switch, wait ten seconds, and turn the power on again. If this problem is persistent, call the service department.

10.1.34 BAD CIRCLE OR MISSING G AT N

This can indicate an incorrect starting position for the circle, an incorrect ending position, or an incorrect description of the arc center location relative to the starting position of the circle. This can also indicate that the circle is programmed in absolute or incremental and the G90 or G91 codes are incorrect.

10.1.35 BAD DATA OR NO TOOL DIA.

This occurs when an H word (in format 1) or the D word (in format 2) is missing in the program. When a program uses cutter radius compensation (CRC) or a fixed subroutine, an H word or D word must be specified. This also occurs when no diameter value is placed in the tool table when using Fixed Subroutines L94NN, L95NN, L9601, L9701, L9801, L9901. A DIAMETER VALUE MUST BE USED.

When the fixed subroutine for engraving (L9201) is used, the Z depth and the R2 variable must be included in the line. If the R0 variable is not specified, it is assumed as R0+0. If the R1 variable is not specified, it is assumed as R1+1.

10.1.36 BAD FIXTURE NO.

Fixtures are numbered E1-E48. This message is displayed when a fixture offset is programmed with a value greater than 48. Format 2: G54-G59 and E6-E48 are used.

10.1.37 BAD INTERPOLATION TYPE AT N

Check to see what type of move is intended for this block. G1, G2, and G3 are the interpolation codes; G1 is for linear moves and G2 and G3 are for circular moves.

10.1.38 BAD R FIELD

This occurs during input from the keyboard or through the RS-232 port (from an off line computer) when the R0 was typed incorrectly. Example: If RO is typed (R with the letter O) or R with no zero (0) is typed (R-.05 is accepted but not R.05 or R+.05).

	Correct the line in the CNC; or correct the line in the text editor and send the program again.
10.1.39 BAD R1 USING L91 AT N =	The R1 parameter is not present in the block containing L9101, or it is present with an improper value. L9101 has 9 possible functions that are defined by the R1 parameter; R1+1. through R1+9.
10.1.40 BAD T WORD AT N=	This message occurs during a SUM command. It indicates that the T word value is larger than the tool changer capacity.
10.1.41 BAD Z OR R0 IN CANNED CYCLE CALL, N =	This is a programming error due to the position of the R plane relative to the Z axis position when initializing a fixed (canned) cycle. PROBLEM 1. The R plane is described to be above the I plane. 2. The R plane is described to be below the final Z depth. 3. The final Z depth is described to end above the I or R planes. 4. Canned cycle not cancelled before returning to tool change position. 5. Canned cycle not stated in absolute as required. SOLUTION Check to see if the program has the correct positioning mode (G90 or G91), and check for correct signs and decimal point placement. NOTE The H code might not be in the program.
10.1.42 CALCULATED RADIUS ERROR OF	This is a programming error that could be due to a G90 or G91 code missing or being in the wrong place (e.g.: giving absolute positions while still in incremental). Also, check the end point description, the beginning position, and the arc center description (I, J, and K), or radius designation. This message does not appear during the Auto mode. It appears when using the SUM command (SU).
10.1.43 CANCEL CRC BEFORE G50.1 OR G51.1 AT N =	Before using CRC (G41, G42), mirror the desired axes (G51.1). Also, turn CRC off (G40) before canceling mirror image (G50.1).

10.1.44 CANCEL CRC BEFORE N =

The control requires the G40 code to cancel cutter radius compensation before the line number listed. (See section 9.0 CUTTER RADIUS COMPENSATION, MAN-0131.)

10.1.45 CANCEL Z AXIS MIRROR BEFORE TOOL CHANGE AT N

The G50.1 code must appear on a separate line before the M6.

10.1.46 CANNOT RETURN FROM SUBPROGRAM OR SUBROUTINE CALLING LINE CANNOT BE FOUND

This is a background editing error message. The line that called the subprogram or subroutine was deleted or cannot be found.

10.1.47 CANNOT START DIRECTLY IN A SUBPROG.

A mid program start must begin from the main program. If it is necessary to start the program in a subroutine, the COPY command can be used to copy the subroutine to the main program as many times as the subroutine will be repeated. The programmer or operator can now do a mid program start to where the subroutine was copied into the main program. After the part is complete, delete the copied portion and run the program as before.

Another method to start the program in a subroutine is to use the AUTO command. Enter AU,# (see AUTO command in the Operators Manual), where # is a sequence number in the subroutine. The control will process the program from the beginning of the program up to the number specified then start execution of the program.

10.1.48 CHAR. SIZE OR DEPTH EXCEEDS 2.5 IN.

Characters that are higher than 2.5 inches cannot be engraved using the Engraving mode (L9201).

10.1.49 CHECK EMERGENCY STOP SWITCH AND MOTOR OVERLOAD RELAYS

If the EMERGENCY STOP button (switch) is not pressed, it is necessary to check the motor overload relays.

> WARNING
> Power off the machine before checking fuses, relays, and/or circuit breakers.

The motor overload relays can be recognized by their white reset buttons. They are located in both control enclosures. The white button must be pressed to reset a relay.

Also, check the brake relay fuse (F17) to see if it is burned out. It is advised to power off the machine when checking the fuses or motor overload relays.

10.1.50 CHECK SPINDLE LUBE, WAY LUBE AND AIR PRESSURE

When this message is displayed, it is necessary to check the way lube level, spindle oiler level, and the air pressure gauge. It is advised to check these oil levels at least once a day.

10.1.51 CLEAR AN EMERGENCY STOP CONDITION

An emergency stop condition was invoked. To clear, gently rotate the red EMERGENCY STOP button (if depressed) in the direction of the arrows on the button (clockwise), then press the JOG key. Wait 2 seconds for the amplifiers to be reset, then continue.

10.1.52 COMMAND ERROR

This occurs because a SET (parameter) command was typed in improperly. It is suggested to look in the Users Manual or use the menu of the control to help with any command format. The following is a list of available SET commands; SETH, SETX, SETY, SETZ, SETA, SETB, SETCS, SETTO, SETP.

10.1.53 COMMAND PROHIBITED BY THE KEY LOCK

With the key lock in the horizontal position, editing commands (CH, CO, DE, IN) cannot be used.

10.1.54 CRC CALLED WITH NO TOOL ASSIGNED AT N =

This occurs when the programmer has turned CRC on without an H word in the program (or in format 2 no D word).

Table 10-1: CRC Called w/ No Tool Assigned at N=

FORMAT 1	FORMAT 2
N1 G41	N1 H1
N2 H1 This is incorrect	N2 G41 This is incorrect
N1 H1	N1 H1 D1 Diameter must be specified for Format 2
N2 G41 This is correct for Format 1 style only	N2 G41 Format 2 style programming & acceptable for Format 1 style

This is a common error in one tool programs when the Z axis is set with the tool at the gauge point. Assign an H word with no tool length offset (only a tool diameter) in the tool table.

10.1.55 D OR H TOO LARGE

The control maintains a table of 99 tool diameters and 99 length offsets. The control will only accept H words and D words up to 99.

10.1.56 DATA TRANSFER FAULT

For this error, it is important to note which axis the transfer fault is on. Note the error message number and report it to the service department.There is a data transfer problem between the 1030 card (slot 8) and the 1010 card for the stated axis.

10.1.57 DNC MODE

This message appears after DNC is typed at the control. The VMC is then ready for DNC operation and is waiting for CNC code from the RS-232 port.

10.1.58 DOOR BEGAN TO CLOSE WHILE ARM WAS MOVING

This message will occur if the door begins to close during a pallet change. The feedback of the door may need adjustment or there may have been a drop in air pressure.

10.1.59 DO YOU WANT TO MOVE TO THE LAST HOME POSITION?

This is not an error message, but this message comes up each time the operator uses the CS command and a home position is established (SETH) in memory.

10.1.60 DO YOU WANT TO ZERO TOOL TABLE?

This prompt is accessed by reinitializing the memory via the RI command. A ‘Y’ response zeroes the tool offset table.

> WARNING
> All tool diameter and length offsets are deleted from memory!

10.1.61 DO YOU WANT TO ZERO FIXTURE OFFSETS?

This prompt is accessed by reinitializing the memory via the RI command. A ‘Y’ response zeroes the fixture offset table.

> WARNING
> A ‘Y’ response deletes all fixture offsets from memory!

10.1.62 DO YOU WANT REINITIALIZE MEMORY?

This prompt is accessed by reinitializing the memory via the RI command. A ‘Y’ response zeroes the memory of the CNC.

> WARNING
> A ‘Y’ response deletes all program data from memory! After memory is reinitialized, a cold start must be performed.

10.1.63 DRY RUN OPTION __ IS IN EFFECT

The selected dry run option (1, 2, or 3) is displayed prior to automatic execution. See the Run Time Menu for the appropriate option definition.

10.1.64 DUPLICATE NAME

No two programs in the library can be identified by the same number. When copying an existing program assign an unused number to the new, or copied program. Use option

2 (display program numbers) of the PR menu to determine which numbers have been used.

10.1.65 DUPLICATE OR BAD PROG. NAME

This will occur if a program is being input through the RS-232 port and the program has an O word in the first line that is a duplicate of an existing number in the program library. Use option 2 (display program numbers) of the PR menu to determine which numbers have been used. Change the O word to one not being used, or remove it. Then send the program out to the control again.

If the program currently active in memory has an O word in the first line, the program will be moved into the program library when the new program is transferred through the RS-232 port. If the program currently active does not have an O word in the first line, the program will be deleted when the new program is transferred.

10.1.66 DWELL, HIT START TO INTERRUPT

This message is displayed when a G04 (dwell) is executed by the CNC. You can interrupt the dwell time and continue program execution by pressing the START key.

10.1.67 E WORD MAY ONLY BE USED WITH G0 OR G1 N=

Fixture offsets cannot be on the same line with a circular move (G2, G3).

EMERGENCY STOP - TAKE APPROPRIATE ACTION:

The appropriate action to take:

1. Release the EMERGENCY STOP button if it has been pressed.
2. Press the JOG button which resets the amplifiers if the problem has been resolved.
3. Type HO and press ENTER, then after the waiting message appears press START (see note).
4. The operator may now resume the running of the part.

> NOTE
> It is not necessary to move the axes to their Cold Start position or establish the part home again unless the message, JOG AXES TO HOME POSITIONS, THEN ENTER THE CS COMMAND appears.

10.1.68 EMERGENCY STOP-TAKE APPROPRIATE ACTION

An emergency stop has occurred. If the emergency stop button is depressed, turn it clockwise until it returns to its normal position. Press the JOG button and the message WAIT 2 SECONDS, THE AXES ARE BEING RESET appears. If this message persists, call the service department.

10.1.69 ENTER AGAIN OR HIT MANUAL TO EXIT

This message comes up when inserting program coding at the keyboard (using the IN command) or when in Manual Data (MD) and Change (CH) mode. It is referring to the line just typed. Something was not acceptable, for example the letter O for the number zero (0) or anything else not correct. The control is still on the line with the problem, so retype the entire line.

10.1.70 ENTER COMMAND SETP AND SET THE MACHINE PARAMETERS

The SETP procedure will ask questions regarding the machine and certain preferences. If any of the questions are not clear, call the service department for help.

10.1.71 ENTER COMMAND MU TO SEE THE MENU

This happens when a command is typed incorrectly. Either it is an unrecognizable command or a comma was omitted.

This message is only a suggestion. It is not necessary to go to the menu to use a command. The menu is there to remind the operator of the command format.

10.1.72 ERROR

This message is a program input error. It is displayed when a program block containing an error is transmitted to the CNC, via the RS-232. It appears at the time the error is received. Therefore, list the program at the control to display the last block of code received. The following block of code in the program, which was not received, is in error. This normally is caused by back to back alpha characters, which must be separated by numeric characters. Also, a 0 could have been typed as an O(s).

10.1.73 ERROR IN USER PROGRAM, ‘SUM’ TO SEE MESSAGE

See the Operators Manual for instructions on using the SUM (SU) command. This message will occur when the control lists an error message in the Auto mode, and the operator persists in running the program by pressing the START or AUTO button. This message will appear when the line in which the error exists is at the top of the stack of lines to be executed.

10.1.74 ERROR N WORD EXCEEDS 99999

This error occurs when the sequence number is larger than 99999. The sequence number must be between .001 and 99999.

10.1.75 ERROR WHILE PROCESSING BLOCK

This is an indication that there may be a problem with the CPU board.

1. Send the machine back to cold start alignment markers.
2. Power off the machines main power, wait 15 seconds and power on the machine.
3. Cold start, then run the program again.

If this error repeats:

1. Save all the programs and offset information currently in memory, write down the present home position
2. Send the machine back to cold start alignment markers.
3. Use the command RI and answer Yes (Y) to all the questions. If the above does not work, please call the Service representative in your area.

10.1.76 ERROR(S) READING TAPE

While the control is inputting a tape or program with the TA command, error option 2, this message appears after input is complete. If the TA command and error option 1 is used (TA,0,1 or TA,1,1), the input of the program would have been stopped if an error in the program is detected.

If errors are detected, list the program and look for missing line numbers to find which lines the control did not accept. Use the IN command to insert the missing lines where needed.

10.1.77 FILE OVERFLOW

This message could mean that the program has too many G codes and M codes on the same line. It could also mean that during CRC, there were too many blocks that the control had to look beyond for the next compensated move (comment lines, Z moves, G code and M code lines, and dwells).

This message may also appear if a fixed cycle is in effect when attempting to initiate cutter radius compensation.

10.1.78 FIXTURE OFFSET OUT OF RANGE

This message occurs when an E word value in the program exceeds 48 (or in Format 2 exceeds G54 - G59) or exceeds E1 - E48.

10.1.79 FIXTURE OFFSET MUST BE APPLIED WITH G0 OR G1

Fixture offsets can not be applied on lines with circular interpolation (G2 or G3). Fixture offset must be applied prior to commencing arc command line (at G0 or G1) or unused fixture offset removed if not applicable to arc.

10.1.80 G28 AND G29 USED WITH CUTTER RADIUS COMP AT SEQ

Before the control can execute the codes G28 or G29, the program must turn CRC off (G40).

10.1.81 G31 USED WITH AN INCOMPATIBLE WORD OR MODE

Only G1, P, and F are allowed with a G31. No other codes are supported. CRC, mirror image, rotation, and drill cycles are not allowed during the execution of a G31 code.

10.1.82 G45-G48 & G52 ARE NOT ALLOWED WITH ROTATION, N =	Codes G45-G48 and G52 are incremental offsets. Rotation can only be used in the Absolute mode. Therefore, these codes are not allowed.
10.1.83 G91.2 IS NOT ALLOWED IN FORMAT 1	The machine is in format 1 with the G91.2 code in the program. The G91.2 code can only be used in format 2. In format 1 you must remain in the Incremental mode when G91.1 is coded.
10.1.84 G92 CANNOT BE USED IN CRC MODE, N =	The G92 code should be used in a line before CRC is turned on. The code G92 can only be used after a G40 (G40 is a default code).
10.1.85 G92 MUST BE ONLY G CODE IN BLOCK	There can be no other G codes in the line with a G92. The program does not need to be in absolute (G90) to use the G92 code.
10.1.86 GNN IS AN UNSUPPORTED G CODE AT N	The G code displayed is not a supported code of the FADAL CNC 88.
10.1.87 HELICAL MOVE TOO SHORT, N =	See error message HELICAL RISE TOO STEEP.
10.1.88 HELICAL RADIUS TOO SMALL, N =	See error message HELICAL RISE TOO STEEP.
10.1.89 HELICAL RISE TOO STEEP, N =	The radius of the circle and the helical rise are radically different in length (usually the rise is much longer in comparison to the radius). Also, depending on the programmed feed rate, the control may or may not be able to handle the situation. Reducing the feed rate in the program can sometimes correct this problem.
10.1.90 I, J, OR K MUST BE SPECIFIED AT N =	This message is displayed when an X, Y, or Z is used with an R0 without a G2 or G3. Full circles must use either the I, J, or K and the direction around the circle. (See Circular Interpolation, section 13.0 INTERPOLATION, MAN-0131.)
10.1.91 ILLEGAL O WORD	This error occurs during program input via the RS-232. The O word must have a value of 1 through 9999. No other character or symbol is allowed.
10.1.92 ILLEGAL G CODE DURING G91.1 MODE AT N =	This message appears when an illegal G code is programmed while in G91.1. (See section 3.0 G CODES, MAN-0131.)

10.1.93 IMPROPER USE OF CANNED SUBR.

This message may appear for the following reasons:

1. No tool diameter in the tool table or No D word was specified.
2. If the programmer omits a required R word from the subroutine.
3. If the programmer does not cancel a fixed cycle with a G80, G28, G29, M6, G49, or H0 before using a fixed subroutine, except for L93 (bolt hole).
4. Using L96 - L99, if R1 variable is equal to the radius of the tool. The R1 variable is used for the radius on the corner of the tool (fillet radius), not the tool radius. For example, for a 1.0 diameter end mill with a .125 corner radius, the R1 variable will be R1+.125, and the tool diameter entered in the tool table will be 1.0.
5. The G41 or the G42 code has not been canceled with a G40 before using a fixed subroutine.

10.1.94 INCHES MODE REQUIRED - OPERATOR MUST SET

The CNC is in the Metric mode (SETME), and the program has a G20 or G70 code to verify that the control is in the Inches mode. To run this program, the operator must set the CNC to the Inches mode by entering the SETIN command.

10.1.95 INCOMPATIBLE G CODES AT SEQ

Some codes, even though they are modal and from different groups (families), are incompatible with one another. Break up the grouping of G codes on the line by moving some of them to the line just before or after the line they are currently on. This messages is also displayed when any G code is on the line with a G53.

EXAMPLE:

N20 G1 F20.
N21 G8 G41 X.5   The G41 and the G8 are incompatible
Move the G8 to the line with the G1
N20 G1 G8 F20.   This is correct
N21 G41 X.5

10.1.96 INCREMENT DIVIDED BY TWO

If renumbering causes a line in the program to be greater than 99999, the increment for renumbering is divided by two and renumbering is resumed.

All of the lines or program blocks must have sequence numbers. The NU command is used to renumber the lines in the program by the increment specified in the first parameter. For example, entering NU,5 will renumber the lines, incremented by five. If program is large, renumber by 1.

10.1.97 INCREMENT TOO LARGE

This message occurs when the increment parameter of the program input (IN) command causes the sequence numbers to exceed 99999 or a typing error in the program has been given a value that is too large. Example: X-123456789.

10.1.98 INCREMENT TOO SMALL	This message occurs when the increment parameter of the program input (IN) command is less than .001. Example: IN, 0.00001.
10.1.99 INPUT XMODEM TRANSMIT BLOCK MISSED ERROR	A packet has been missed or came out of order during Xmodem transmission.
10.1.100 JOG AXES TO HOME POSITIONS, THEN ENTER THE CS COMMAND	This message appears at power on. If the axes are at the Cold Start position (machine home), enter the CS command. The operator needs only to jog the axes if they are not at the Cold Start position. This message also appears when the tool changer crashes and the AUTO button is pushed or the MD command is used. However, the operator does not have to jog to Cold Start. Use the SETCS command to reference the Cold Start position, then enter HO to return the axes to home, and when the axes are back at the Cold Start position, visually check, and then enter the CS command. SETP will also make this message appear. If the operator used the SETP command at the part home, handle the situation in the same way as suggested for a tool changer crash (see above).
10.1.101 LOOK AHEAD WAS CANCELED BY OPERATOR	This indicates that the NO LOOK AHEAD buffer option of the Run Time Menu has been selected. The CNC processes only one block at a time, while displaying only two blocks of the program on the screen. This option can be selected or canceled using the Run Time Menu by typing MU in the Automatic mode.
10.1.102 M,S,T LOCKOUT IS IN EFFECT	This message indicates the M function, Spindle, and Tool change lockout option of the Run Time Menu has been selected. In this mode, M function, Spindle or Tool Change commands will not be performed during Dry Run. This option can be selected or canceled using the Run Time Menu by typing MU in the Automatic mode.
10.1.103 M FUNCTION TOO LARGE AT N =	M functions cannot exceed a 2 digit value. (See section 8.0 COMMANDS, MAN-0131 or the menu (MU) for a list of the accepted M functions.)
10.1.104 MAIN PROG. NOT FOUND	This is a programming error. There are subroutines written in the beginning of the program without an M30. M30 is used to separate the subroutines from the main program. Insert the M30 in the proper place. (See section 8.0 Commands, MAN-0131.) The M30 code on the first line of the program and a subroutine heading (L100) on the next line will also cause this error message to appear. *N1 O1 (START OF PROGRAM* *N2 (ACCEPTABLE PROGRAM EXAMPLE* *N3 L100 (SUB #1*

N4 X1.
*N5 **
N6 L200   SUB #2
N7 L105
N8 M46
N9 X-5.Y-1.
N10 M47
*N11 **
N12 L300 (SUB #3
N13 G91
N14 L204
N15 L104
N16 G90
*N17 **
N18 M17 This ends the last subroutine
N19 M30 This separates the sub section from
*N20 * The main program*
N21 (MAIN PROGRAM

10.1.105 MEMORY ERROR, RELOAD PROGRAM

This is a memory error, turn the machine's main power off. Then power on and reload the program. If this does not work, save all the programs in the memory (PU,3) and use the RI command, answering Yes (Y) to all the questions. After using the RI command, load the programs back into the control.

10.1.106 MEMORY ERROR, RESPOND WITH Y TO DELETE BAD BLOCKS

There is a possible hardware problem when this message appears. If the operator answers Yes (Y) to this question, the control may delete some of the program in memory. Answer Yes (Y) only if the program in memory can be replaced (if the program is stored on paper tape or on computer disk). Answer No (N) if the program needs to be saved.

Turn the machine's main power off, then power on and reload the program. If this does not work, save all the programs in the memory (PU,3) and use the RI command, answering Yes (Y) to all the questions. After using the RI command, load the programs back into the control. If this does not help, call the service department.

10.1.107 METRIC MODE REQUIRED - OPERATOR MUST SET

At power on, the control is in the Inches mode. The Metric mode must be turned on by using the SETME command.

This message will appear if the G21 or G71 code is present in the program to verify that the Metric mode has been turned on by the operator.

10.1.108 MOTOR OVERLOAD	This is an indication that something has stalled or overheated a motor, and is associated with an emergency stop (see EMERGENCY STOP).
10.1.109 MOVE EXCEEDS AXIS LIMIT AT N = CHECK PROGRAM AND TOOL OR FIXTURE OFFSETS	The most common reason this message appears is because of either a missing or an extra G90 or G91 code. This message will appear only when the control is in the Auto mode or Manual Data (MD) mode. This message will not appear in the Sum mode (SU). When the control is executing the program, processing of the program is ahead of the current line that is being executed. If the control detects an over travel situation while processing the program, then execution of the program will stop immediately. The setup person may have to consider moving the fixture to another location on the table. The program may be correct, but the fixture may be set up incorrectly. The programmer can provide information to the setup person, indicating how close to the axis limits the fixture can be located.
10.1.110 MOVE TURRET TO TOOL 1 AND ENTER SETTO COMMAND	When this occurs, the head will be above the tools in the turret with the turret ready for loading. Move the turret using the turret CW or CCW buttons so that the tool designated to be tool #1 is under the spindle, then remove tool #1 from the turret. Press the JOG button. The head will stay where it is (4 above the Z axis CS position), and the turret will move back to its home position. Now use the command SETTO to establish this turret position as #1. Use the HO command or jog the head down to the Z axis CS position. Next use the command TC,1 to reopen the turret. Replace tool #1 in the turret and press the MANUAL button to bring the head down over tool #1.
10.1.111 NEXT ENTRY IS OUT OF RANGE	This message is displayed by entering pitch error compensation for the ball screw when an entry exceeds the maximum allowable difference of 9 (see Survey command in the Maintenance Manual). For example, if an entry is 15 and the next consecutive entry is 0, the difference is greater than 9.
10.1.112 NO ANGLE, THE POINTS ARE THE SAME	The calculated angle of either function 3 or 4 of L9101 is the same as the expected angle.
10.1.113 NO MEMORY AVAILABLE FOR NEW PROGRAM	When this occurs, the operator will have to remove (delete) as many programs as necessary from the program library in order to input another program from the RS-232 port. The standard memory capacity is 38K. FADAL distributors have a memory expansion board available that increases the memory capacity to 16 MEG.
10.1.114 NO OTHER WORDS ALLOWED WITH O WORD	If program coding is used on the first line of the program along with the O word, this message appears. A comment is allowed with an O word and is used to identify each program in the library. The first sixteen characters of this comment are shown when the program library menu displays the programs in memory.

EXAMPLE: *N1 O1 G0 G90 (P/N 1234 This line is incorrect*
N1 O1 (P/N 1234 This line is correct
N2 G0 G90

10.1.115 NO PARAMETERS OR PARAMETERS CORRUPTED MACHINE DEFAULTS USED

This message occurs most commonly after the memory has been zeroed from the DI diagnostics mode. This will also occur if there is a memory error. Enter the machine parameters with the SETP command. The parameters to use are usually found recorded on a check off sheet on the inside door of the pendant.

10.1.116 NO RESPONSE FROM AXIS

This happens when powering on the machine and the CPU (1400) does not get a response from the axis controller (1010).

If this message appears for the X, Y, Z, A, or B axes, power the machine's main power off and then on again. If this does not help, call the service department.

10.1.117 NO TOUCH OR INCOMPLETED POINT AT N =

The move that causes the probe to touch the part was completed without a touch. Increase the length of the move so that the probe touches. Also, you must activate the selected probe by coding an M64 for an MP8 probe, or M65 for a TS20.

10.1.118 NOTE ERROR THEN HIT MANUAL

When the programmer is inputting a program through the RS-232 port (using the command TA,1,0 or TA,1,1), the input will be terminated if the control detects a bad programming word (EXAMPLE: O for 0, or double letters (XX1.5), etc.). The programmer will have to edit the program at the computer or Teletype before inputting the program again.

The control will ignore all errors in the program and continue input by using the error option 2, with the TA command (EXAMPLE: TA,1,2). An error count is given at the end of transmission. If there were errors detected, list the program to find missing line numbers. If the lines are numbered in sequence before transmission of the program, look for the numbers that are out of sequence. After the numbers have been located, use the IN command to insert the missing lines.

10.1.119 NUMBER OUT OF RANGE

This message appears when a number of twelve digits or greater is typed in. This error may be detected by the machine after making a calculation. The calculated number may be out of range, for example, the macro calculation is less than .0001.

10.1.120 OFFSET ENTRY ERROR

This message is displayed using the Utility (UT) command, by pressing the MANUAL key when the CNC prompts you for a tool diameter. It is also displayed by entering a diameter value that is too large.

10.1.121 ONLY BLOCK SKIP ALLOWED WITH MACRO	The only non macro command allowed with a macro line is the block skip character.
10.1.122 ONLY M3, M4 & M5 ARE ALLOWED WITH M6 AT N =	This message appears when the M6 is coded with an M function other than an M3, M4, or M5. The unacceptable M function must be moved to another line.
10.1.123 ONLY Z, L, R & F WORDS ALLOWED	This message appears when the programmer uses the fixed subroutine L9201 for engraving, and has programmed other words in the line with the L9201 other than Z, L, R, and F.
10.1.124 ORIENTATION FAILURE	Visually check the air pressure, it must be between 80 and 90 PSI. If this problem persists call the service department.
10.1.125 OUT OF FILE SPACE, A FILE COMPRESSION IS BEING TRIED	This is a message that informs the programmer that the memory of the control is almost full. It is usually displayed when editing the active program in memory, or during program input via the RS-232. This does not destroy any portion of the program. A file compression organizes the memory so that unused portions are made available. The NE command will perform a file compression before prompting you with a (Y or N) to delete the current program. Also, a file compression is performed by deleting the current program using option 5 from the PR menu. It is always best to retry the command that caused this message to appear. After the compression, the command will often work the second time. If unwanted programs are stored in the program library, remove them to create more room in memory. If there is only one program in memory, try to reduce the size of the program by removing unnecessary comments and making full use of all modal codes. Create subroutines for repetitive portions of the program wherever possible. The program may have to be broken up into separate operations. The standard memory capacity is 38K. FADAL distributors have a memory expansion board available that increases the memory capacity to 16 Megabytes.
10.1.126 O WORD OUT OF RANGE	This message occurs using the program input (IN) command when typing an O word greater than 4 digits. O words must be an integer of 1 through 9999.
10.1.127 PALLET DISABLED IN PARAMETERS	This message occurs when a pallet changer command or code is used and the SETP parameters indicate no pallet changer exists on the machine.
10.1.128 PALLET IN STORAGE	This message occurs when there is an attempt to store a pallet where a pallet is already stored.

10.1.129 PALLET NOT FULLY STORED

This message occurs when the time to store a pallet exceeds the allotted time. The feedback on the arm may need to be adjusted. Call the service department for assistance.

10.1.130 PALLET MUST BE CLAMPED

This message occurs when both pallets are in storage and a command is given to make a pallet change. One of the pallets must be loaded and clamped on the table before a pallet change can occur. The feedbacks for pallet loaded and pallet stored are being read at this time.

10.1.131 PARAMETER ERROR

After a command is entered and a comma is typed, the control is expecting a parameter. If a parameter is not typed before pressing the ENTER button, this error message appears.

EXAMPLE: *AU, This is a parameter error*

> NOTE
> If the operator is not using a parameter, for instance, the second and third out of a possible four, then the parameters can be omitted when the fourth parameter is typed in.

EXAMPLE: *AU,50,,,1 This is acceptable*

Also if the DE command is used to delete a line that does not exist, or the CO command is used to copy lines that do not exist, this message appears.

10.1.132 PARITY ERROR

This message appears during input through the RS-232 port and is caused by several factors:

1. Unshielded cable is being used, and the cable is subject to electrical noise.
2. The baud rate is too fast for the length of cable used.
3. There is a failure in the RS-232 port (hardware problem).
4. There may be a problem with the computer that is sending the program.

10.1.133 PARITY ERROR DURING DNC

This occurs during DNC transmission of data (see the error message PARITY ERROR).

10.2 PLEASE PUT AN O WORD AT THE FIRST OF THE CURRENT PROGRAM THE FOLLOWING PROGRAMS ARE IN MEMORY

To use the PR command, the active program must have an O word and a number in the first line. The number must be different than the other numbers in the program library. This is why it lists the other programs in memory.

EXAMPLE:

N.1 O43 (P/N 34-765)
Here, a line was inserted before N1 using the command IN,.1
N1 G0 G90 S10000 M3

A comment can be added to the line with the O word. These comments help identify the program.

10.2.1 POINTS ARE ON SAME LINE AT N =

The 3 touch points of L9101 function 1 are on the same line. Check the program for positioning errors.

10.2.2 POSSIBLE PROBE OVER TRAVEL

During the UT command, this error occurs for two cases:

1. when the Z axis home position is set below the Cold Start position; and the tool change is made to the Probe.
2. when there is a positive offset value used, and the tool change is made to the Probe.

10.2.3 POSITION LIMIT

This message will occur when the program has instructed the tool to move out beyond the axis limits. When the tool moves beyond the axis limit the CNC issues an emergency stop. The machine will stop at the line that caused the over travel.

10.2.4 PRESS Y TO KEEP THIS POSITION PRESS N TO RETURN TO LAST POSITION

These messages appear after jogging while in Slide Hold or Single Step modes. For the proper application of the Jog Away feature see section 8.0 COMMANDS, MAN--0131.

10.2.5 PROBE TEST = FAILURE

Testing the probe by using the Utility (UT) command failed. Verify that the probe was properly interfaced. If an MP8 or MP9 is being tested, be sure the 9- volt battery has a charge. Also remove any obstructions between the probe (transmitter) and the collector unit (receiver).

10.2.6 PROBLEM POSITIONING SLIDES TO ZERO

This error only occurs during Cold Start, when the Glass Slides are not aligned close enough to the indicators during Cold Start. Place the machine in Jog and manually align the Cold Start indicators more accurately.

10.2.7 PROGRAM DOES NOT EXIST RETRY OR HIT MANUAL TO EXIT

This happens when the selected program number does not exist in the program library. Enter a program number that does exist, or press the MANUAL button, and then use the PR command again to display the programs that are stored in memory (option 2).

The programmer may have put an O-1 or an O.1 as a program number. The number will show up when the programs are displayed in the library but the control will not be able to retrieve the program. If the operator uses the PU,3 command and presses the EMERGENCY STOP button when the lost program is being displayed on the screen, that program is now active in memory and can be edited. Delete the bad O word, from O-1 or O.1 (which appears as 1000 when it is listed in PR) to O1.

10.2.8 PROGRAM NOT FOUND

This message is displayed when the PA command is used to display the active program, and no program is active. Use the PR command to activate the desired program stored in memory.

10.2.9 PROGRAM BLOCK NOT FOUND

This message is displayed when the LI command is used to list selected blocks of the active program, and the selected blocks are not found or the program is not active. View the program by using the PA command.

10.2.10 P WORD TOO LARGE

The P word for referencing a line number using M99 is too large of a value. Line numbers can not exceed 99,999.

10.2.11 RESET THE EMERGENCY STOP SWITCH

The EMERGENCY STOP button has been pushed. Turn the button clockwise until it is released and then push JOG.

10.2.12 RESOLVER FAULT OR SCALE ERROR

This message may occur in three cases as follows:

1. A failure of the resolver.
2. An axis runaway during power on.
3. A large Scale or Motor error, indicating a possible Scale failure.

Call the service department when this message appears.

> NOTE
> Do not run the machine when this message appears.

10.2.13 RAILS NOT ALIGNED

This message occurs when the rails for the palette hanger are not aligned or the feedback switches need adjustment. Call the service department for adjustment.

10.2.14 RETURN PALLET TO THE LOAD POSITION

This message occurs when a pallet change is attempted and the other pallet is away from the load position. The operator must slide the pallet into the load position before the other pallet on the table can be changed. The feedback from the pallet loaded and pallet stored are being read at this time.

10.2.15 ROTARY AXIS MOVE TOO LONG. N =

The maximum incremental rotary move for one block is 1080.00 degrees. If it is necessary to go further, break up the move into several blocks.

10.2.16 RS-232 ERROR DURING DNC

This error only occurs during DNC operations. It may occur while using 9600 baud to DNC. Slow the baud rate. It may also indicate line noise or the possibility of dropped characters. These would indicate a possible problem with the RS-232 cable or the communications port. Check the cable and communications port and retry DNC.

Also, failure of the CPU may cause this error. If the above corrections are unsuccessful, contact the maintenance department.

10.2.17 SCALE ERROR

A scale error has been detected. The LEDs in the scale interface box in the back of the machine will indicate which axis gave the error. Call the service department.

10.2.18 SEE MENU FOR NEW CD FORMAT

The CD command is different for some versions of software. See the menu (MU) for these changes.

10.2.19 SEQUENCE NUMBER TOO LARGE

This error occurs using the Program Input (IN) command when the ‘From’ parameter is specified greater than the maximum allowable number. Sequence numbers can not exceed 99,999.

10.2.20 SEQUENCE NUMBER TOO SMALL

This error occurs using the Program Input (IN) command when the ‘From’ parameter is less than the minimum allowable number. Sequence numbers can not be less than .001.

10.2.21 SERVO AMPLIFIER FAULT

This message appears when the EMERGENCY STOP button is pressed. The operator must pull the button out (for ‘85 and older models) or turn it clockwise (for ‘86 to present models) and then press the JOG button to reset the amplifiers. If the amplifiers do not reset, call the service department.

10.2.22 SINGLE STEP

This message indicates that the control is in the Single Step mode. If the green START button is pressed when in this mode, the following program block is executed. The feed rate potentiometer affects all axis motion.

At the end of each block, the operator can press the JOG button and jog away from the current position without aborting the operation. Program execution can be continued at the new location, or the axes can be returned to the position at which the Jog function was initiated. To exit single step, press the AUTO button for continuous block execution.

10.2.23 SLIDE HOLD

Pressing the SLIDE HOLD button stops all axis motion. The distance to finish the move is presented under the blinking SLIDE HOLD message.

It is possible to press the JOG button and jog away from the current position while in the Slide Hold mode without aborting the operation. Program execution can be continued at the new location, or the axes can be returned to the position at which the Jog function was initiated. To exit from the Slide Hold mode, the operator must press the START button if in single step, or press the AUTO button for continuous block execution.

10.2.24 SPINDLE CONTROLLER DOES NOT RESPOND

Note the error number and see the list at the end of this chapter. This simply could be a problem that occurs when the operator is powering on the machine. Turn the machine's main power off. Press the motor overload relay button, and then power on again. If this does not help call the service department with the error message and number.

The motor overload relays can be recognized by their white reset buttons. They are located in both control enclosures. The white button must be pressed to reset a relay. Call the service department first.

10.2.25 SPINDLE CONTROLLER OR DRIVER FAULT

Note the error number and see the list at the end of this chapter. This message is displayed if the spindle was commanded to turn on after an emergency stop without pressing the JOG button to reset the amplifiers.

If this message appears when not in the Emergency Stop mode, take these steps to help the service department analyze the problem:

1. Look at the air pressure gauge and write down the current air pressure (Do not change it at the machine!)
2. Check to see if the belts are on the pulleys.
3. Do not turn the power off.
4. Call the service department and report this problem (while the machine is on).

10.2.26 SPINDLE CONTROLLER SOFTWARE UPDATE IS REQUIRED	The current software module (1610) detected spindle software that was not compatible. Call the service department.
10.2.27 SPINDLE DRIVER FAULT	This is the result of the air pressure being too low while the spindle is on. Air pressure should be set between 80 and 90 PSI. Visually check the air pressure and start the program over again. If the air pressure is not the problem, power off the machine and check the motor overload relays to see if they have been tripped. The motor overload relays can be recognized by their white reset buttons. They are located in both control enclosures. The white button must be pressed to reset a relay.
10.2.28 SPINDLE FAILURE WHILE TAPPING	If the spindle stops while in G74, G75, or G84 modes, this message will appear. Problems such as a dull tap, undersized hole, poor tapping lubricant, or a shallow hole must be eliminated. The programmer may want to consider thread milling as an alternative to tapping. Call the service department if this is persistent.
10.2.29 SPINDLE FAILURE DURING REVERSAL	This will occur when an M3 and an M4 are in the same line. If the spindle stops while in G74, G75 or G84 modes, this message will appear. Problems such as a dull tap, undersized hole, poor tapping lubricant, or a shallow hole must be eliminated. The programmer may want to consider thread milling as an alternative to tapping. Call the service department if this is persistent.
10.2.30 SPINDLE HAS FAILED TO TURN ON	Either the Hall Effect Switch on top of the spindle pulley has not recognized that the spindle is turning, or the spindle actually has not turned on and there is another problem. Call the service department.
10.2.31 SPINDLE FAULT LINE	The spindle inverter has detected a fault. See the fault number on the inverter and report this to the service department.
10.2.32 SPINDLE MOTOR TEMPERATURE FAULT	Check to see if the spindle fan is on. If it is not, turn machine's main power off and check the fuse for the spindle fan (F24). Allow time for the spindle motor to cool down and attempt to run the machine again. If this message persists, call the service department.
10.2.33 SPINDLE WOULD NOT STOP	The spindle would not stop in the designated amount of time (12 seconds). If this message is persistent, call the service department.
10.2.34 STACK OVERFLOW	This message should be reported to the service department. It is caused by excessive stacking of keyboard commands.

10.2.35 SUBROUTINE CALL IS NOT ALLOWED IN MDI	Subroutines can be executed in AUTO mode ONLY.
10.2.36 SUBR. DOES NOT EXIST	This message indicates that a subroutine was called that is not in the subroutine section of the program. Check to make sure that the subroutines were numbered properly. This message also appears if the G66 code is used on a line that does not have a subroutine call on it.
10.2.37 SUBR. NESTING ERROR	This is a message that occurs when a subroutine heading (L0100) is entered after the M30 code.This message will also occur if a Fixed Cycle is still active (use a G80 to cancel) or G68 (rotation) is still active when a tool change or end of program is coded (use a G69 to cancel). *N20 M17* *N21 M30* *N22 L200 This is not permitted after an M30*
10.2.38 TAPE INPUT TERMINATED	This message reports to the operator that the tape, or program input process is complete.
10.2.39 TAPE IS GOOD	When verifying a punched tape, this message appears if the tape is accepted. This message also appears if no errors are detected in transmission when the TA command is used.
10.2.40 TEMPERATURE FAULT	For any temperature fault, the operator must check the cooling fans; if they are not operational, turn the machine's main power off and check the fan fuses (F23, F24, F27, and F28). If this message is persistent, call the service department.
10.2.41 TEMPORARY CONFLICT WITH AUTO	This message occurs when background editing is being used. The edit just made is in the area where AUTO is currently executing. The change will be made when AUTO is out of that area.
10.2.42 THE AXIS CONTROLLER IS NOT A 1010-4	See Survey in the Maintenance Manual.

10.2.43 THE SURVEY CONTAINS ERRORS, PLEASE REVIEW	See Survey in the Maintenance Manual.
10.2.44 THE SURVEY WAS NOT WRITTEN TO THE AXIS CONTROLLER	See Survey in the Maintenance Manual.
10.2.45 THERE IS NO SURVEY	See Survey in the Maintenance Manual.
10.2.46 THREAD LEAD NOT SPECIFIED AT N =	Fixed cycles G74, G75, and G84 need a Q word in the line that represents the decimal thread lead of the tap. If the Q word is omitted then this message will appear. If the programmer uses the fixed cycles G74, G75, or G84 and does not cancel the cycle with any of the following codes: M6, G28, G29, G49, G80, H0, then this message will appear at the next Z move.
10.2.47 TOO MANY BLOCKS FOR GAP	This happens when the CO (copy) command is used and there are too many lines being copied between existing lines in the program. Use the NU (renumber) command and copy again, now that the program is renumbered.
10.2.48 TOO MANY M FUNCTIONS AT N=	This error occurs when another M function is coded with the M17 or M30. These two codes must be the only M function on the line.
10.2.49 TOO MANY CONSECUTIVE NON MOTION BLOCKS	This error message appears if a program contains over sixteen consecutive comment blocks (with no program coding in the lines). Put coding between the comment lines or put comments on lines with program coding, so that the program can have comments on every line. Comment blocks in the beginning of the program can be placed in an unused subroutine. This way the control never reads the lines when executing the program, but the operator can list the program and read the comment lines.
10.2.50 TOO MANY PARAMETERS	Commands have associated with them a certain number of parameters. If more parameters are typed in than allowed, this message appears.
EXAMPLE:	*AU,50,,,1, or AU,50,,,3,1*
	Refer to Menu (MU) on the control for the correct format of each command.

10.2.51 TOO MANY SUBR. CALLS

Nesting can only be seven deep. If the program goes further than seven deep, then this message appears.

10.2.52 TOOL BREAKAGE DETECTION = FAILURE AT N =

When using the TS-20 touch probe is used for tool breakage detection, this error message indicates the program line where the broken tool is detected. Replace the tool and continue the program.

10.2.53 TOOL DIAMETER TOO LARGE AT N=

This error message is displayed in cutter radius compensation when the programmed move is smaller than the radius of the cutter. To correct this problem:

1. Check the tool diameter entered in the tool table to see if it was entered correctly.
2. Check the program to see if it contains any typographical errors.
3. If the intention was to make a rough pass using a larger tool diameter than the finish pass tool diameter, the smallest programmed inside radius, on the part, should be as big as the tool radius (half the tool diameter) entered for the roughing pass.
4. Check the math work to see that it is correct, and was transferred correctly to the program.

When debugging a program that uses cutter radius compensation, use a zero diameter in the tool table when running the program or when using the SUM command for the first time. Then put the tool diameter in the tool table and try it again. This will make the debugging process easier.

10.2.54 TOOL OFFSET NOT ALLOWED DURING Z MIRROR

If mirroring the Z axis is desired, first call up the tool length offset, then code the G51.1 Z0. The H word must be on a line in the program before the G51.1 Z0.

10.2.55 TOOL NN IS IN THE SPINDLE

Identifies the current tool that is in the spindle. This is displayed when entering the program execution (AU) and the Manual Data Input (MD) modes, and is also listed in the tool table display (DT).

10.2.56 TOOL NUMBER TOO HIGH

The starting tool number of the utility (UT) tool setting cycle must not exceed that of the tool changer capacity.

10.2.57 TOOL TURRET LOCATION IS NOT SET

Rotate the tool turret so that tool #1 is adjacent to the spindle and then type SETTO.

10.2.58 TRANSFER ERROR, PLEASE RETRY	See Survey command in Maintenance Manual.
10.2.59 TROUBLE READING THE EXTERNAL SLIDE HOLD SWITCH	The feedback from the external slide hold switches is erratic or inconsistent. This is an indication that the external slide hold switch system is faulty. Call the service department.
10.2.60 TYPE A Y TO IGNORE, OR TURN POWER OFF AND THEN ON AGAIN	This error message appears only after the message AXIS CONTROLLER DOES NOT RESPOND DURING POWER UP SEQUENCE. This is an indication that there is a communication problem between the CPU and the axis controller. Call the service department.
10.2.61 UNDEFINED MACRO ERROR	This message will appear when situations during a macro execution cause the system to fail, but there is no specific error or message associated to the situation.
10.2.62 WAIT 2 SEC., THE AXIS DRIVERS ARE BEING RESET	This message occurs when the JOG key is pressed after an axis fault. The control returns to the Command mode after 2 seconds. The axis drivers are reset and the machine can continue normal operation.
10.2.63 WAITING	In the WAITING mode, the operator has the choice of pressing either the START or AUTO button to execute automatic motion. Or, the operator can press the MANUAL button to exit to the Command mode.
10.2.64 WAITING ON AIR VALVE	When this occurs, the control is waiting on a signal from one of the air valves indicating that the valve has completed its job; usually air pressure to the machine has dropped below 80 PSI. If this message persists, call the service department.
10.2.65 WARNING POSSIBLE GOUGE AT N =	Generally caused by math errors in the program. The CNC has processed a move that possibly could cause the cutting tool to damage the part. The error usually is not large enough to make any noticeable difference on the machined part. Therefore, only a warning is displayed. If there is a noticeable difference, review your program math.
10.2.66 WAY LUBE PRESSURE SWITCH FAILURE	Check the way lube oil level if this message appears.
10.2.67 WRITING ERROR, WAIT 2 MIN. AND RETRY	See Survey in the Maintenance Manual.

10.2.68 X,Y OR Z MOVE MUST BE SPECIFIED AT N =	Probe function #1 (L9101) has been programmed without an axis specified for the given plane. For example, if probing is to be used in the XY plane (G17), X and/or Y must be programmed in the same block as L9101 R1+1. Z can be used if in the ZX (G18) or YZ (G19) plane.
10.2.69 YOU CANNOT DELETE THE PROGRAM THAT IS CURRENTLY ACTIVE	To delete a program in memory, it must be in the library. Use the PR command and switch the program currently in memory with a program in the library. The program can then be deleted. If there are not any programs in the library, use option 3 to start a new program. This will put the currently active program in the library and then the program can be deleted. Or just simply use the DE command and delete starting from the first block in the program to the last.
10.2.70 YOU HAVE A VERSION UPGRADE OR MEMORY HAS BEEN CORRUPTED... MEMORY NEEDS TO BE ZEROED	This occurs when the control detects that a new module has been inserted. See the Maintenance Manual for proper zeroing procedures. This may also occur because of a memory error and the memory needs zeroing.
10.2.71 YOU MUST ENTER THE BACKLASH TABLE	The CNC does not have a valid backlash table. Enter the backlash for each axis by using the BL command. The backlash set at the factory is documented on the inside surface of the pendant door. The backlash information is retained by the battery backed memory card. If this message appears again after the BL command has been used, call the service department.
10.2.72 YOUR VERSION OF CNC MAIN NEEDS TO BE UPDATED	The control has detected an incompatibility with the CNC memory manager software on the 32MP pendant. Call the service department for an update.
10.2.73 Z, Q, OR F MISSING IN CANNED CYCLE CALL, N =	For the proper format for each fixed cycle see Section 4.0 FIXED CYCLES, MAN-0131. .
10.2.74 Z & M6 LOCKOUT IS IN EFFECT	This message indicates the Z axis and Tool Change Lockout option of the Run Time Menu has been selected. In this mode, Z axis movement and tool change commands will not be performed during Dry Run. This option can be selected or canceled by using the Run Time Menu.

10.2.75 Z AXIS MUST BE AT COLD START FOR TC,1

Check to see if the Z axis alignment was properly set at cold start. The operator cannot use the TC,1 command unless the Z axis is at the Z axis CS position. Use the HO command or jog the axis to the Cold Start (Z0).

If the SETZ command has been used, first use the HO command to bring all axes home, then enter the SETCS command. Jog the Z axis to Z0 and then use the command SETH so the X and Y home position is not lost. The TC,1 command can now be used to open the tool changer. If necessary return the Z axis to the previous setting and use the SETZ command. (See the error message JOG AXES TO HOME POSITIONS if the tool changer crashes.)

10.3 AXIS/SPINDLE CONTROLLER ERROR MESSAGES

10.3.1 DESCRIPTIONS

Error numbers help discern the source of the problem the axis or spindle is reporting. The error number and text will follow the message CALL THE SERVICE DEPARTMENT AND REPORT THE MESSAGE AND NUMBER ON THE SCREEN TO CORRECT THESE PROBLEMS.

Error numbers from the AXIS or SPINDLE CONTROLLER

10.3.2 1. NO MOTOR FEEDBACK

This message appears on power up if no motor feedback (resolver or encoder) or no motor motion is detected by the axis controller.

10.3.3 2. ENCODER NOT RESPONDING PROPERLY

This message appears during the rigid tap operation if the encoder feedback is interrupted during the spindle operation for any period longer than 3 seconds. The spindle operation is halted and this error message is displayed.

10.3.4 3. LOGIC JUMPER INCORRECT OR COMMAND SIGNAL MISSING

This message appears on power up. An illogical motor-tach jumper configuration or failed component may cause the axis to run away in the opposite direction of the command signal. To prevent runaway, axis operation is halted and this error message is displayed.

10.3.5 4. ENCODER IS NOT 1024 LINE

This message appears on power up, only for axes with a 1024-line encoder as motor feedback, if the count of the encoder lines per revolution of the screw is not 1024 lines per turn.

10.3.6 5. NO INDEX MARK DETECTED

This message appears during the cold start process, if the index mark of the axis' primary feedback device, encoder, or scale is not detected.

10.3.7 6. SWITCH 1 DISABLED	This message is not used.
10.3.8 7. RIGID TAP PRECYCLE ENCODER COUNT PROBLEM (NOT ENOUGH COUNT)	This message indicates that the spindle feedback is not functioning properly. Prior to the rigid tap cycle, the spindle feedback is checked for operational accuracy. Malfunction of the feedback results in termination of the cycle and display of this error message.
10.3.9 8. SPINDLE MAGNET NOT DETECTED OR SPINDLE NOT RUNNING	This message appears during spindle operation if the spindle magnet is not detected for any period longer than 3 seconds. This problem could be related to either command signal, spindle not turning, or spindle magnet malfunction.
10.3.10 9. SPINDLE FAULT LINE DOWN	The fault line signal down to the inverter should stay high during spindle operation. If this signal is held low when the spindle is running then this error message is displayed.
10.3.11 10. ENCODER AND MAGNET NOT RESPONDING OR SPINDLE NOT RUNNING	This is a spindle operation error message. During spindle operation, the encoder feedback and spindle magnet are monitored. If no proper response from either of them is detected for any period longer than 3 seconds, then this error message is displayed. This error could also be caused by the command signal, i.e., if the spindle is not turning.
10.3.12 11. MOTOR OVERLOAD. EXCESSIVE FOLLOWING ERROR	This message is not used.
10.3.13 12. MOTOR OVERLOAD. PULSE COMMAND STEP COMMAND CHECKSUM ERROR	This message appears at the completion of a move if the number of pulses required to complete the move does not match the number of pulses received from the CPU by the axis controller, usually indicating a hardware-related problem.
10.3.14 13. MOTOR OVERLOAD. FOLLOWING ERROR GREATER THAN THE OVERLOAD FACTOR	This message is not used.

10.3.15 14. MOTOR OVERLOAD. MISSING 0 OR 1 COMMAND	This message appears if an axis move command is not initialized properly (missing 0 or 1 command), usually indicating a hardware-related problem.
10.3.16 15. TIME OUT ON RIGID TAP PRE-CYCLE TEST	This message appears during rigid tap precycle test if the spindle rotation is interrupted or the spindle magnet is not detected for a certain length of time.
10.3.17 16. TIME OUT ON RESOLVER RESPONSE	This message appears any time the resolver zero-crossing pulse is not detected within the allowable time (4-10 msec). The zero-crossing pulse is hardware-generated every 1 msec. This can also be caused by motor over temperature or by severe motor vibrations.
10.3.18 17. BAD READING ON RESOLVER PORT	This message appears if the value of resolver “counts”, read from the resolver port when the “zero-crossing” pulse is detected is too large, indicating possible failure of resolver cables, 1010-4(-5) card, or the resolver itself. This can also be caused by motor over temperature or by severe motor vibrations.
10.3.19 18. SERVO AMPLIFIER FAULT LINE DOWN	This message appears if the axis card cannot detect the fault line signal from the amplifier, or the fault line signal strength drops below 10V, indicating a possible amplifier fault.
10.3.20 19. STACK OVERFLOW. OVERLOADED WITH STEP COMMAND	This message appears if the stack on the axis card (used to store CPU commands until they have been serviced) overflows.
10.3.21 20. MOVE TRANSFER FAULT. INCOMPLETE DATA FOR MOVE COMMAND	This message appears when an axis move command is improperly transferred to the axis card, as indicated by a checksum comparison between what the CPU sent and what the axis card received.
10.3.22 21. SPURIOUS INTERRUPTS. NOT SERVICEABLE	This message appears if the interrupt currently being processed is an undefined interrupt, and thus cannot be serviced, indicating possible failure of the 1010-4(-5), 1030, or main CPU. This could also indicate a software problem.
10.3.23 22. BAD SCALE READING	This message appears if the feedback from the scale port is outside allowable limits, indicating possible failure of the EXE box, scales, cables, or 1010-4(-5) card. This can also be caused by severe motor vibrations.

10.3.24 23. MOTOR OVERLOAD. ERROR > OVERLOAD FACTOR IN STANDBY MODE	This message appears (in stand-by mode) when the following error exceeds the user-defined overload factor, which sets, in motor turns, the maximum following error for the axis operation. MESSAGES 23, 24, 25, AND 26 ARE NO LONGER RELATED TO ERROR 13.
10.3.25 24. MOTOR OVERLOAD. ERROR > OVERLOAD FACTOR IN POINT TO POINT MODE	See ERROR 23.
10.3.26 25. MOTOR OVERLOAD. ERROR > OVERLOAD FACTOR IN CONTOURING MODE	See ERROR 23.
10.3.27 26. MOTOR OVERLOAD. ERROR > OVERLOAD FACTOR IN JOB MODE	See ERROR 23.
10.3.28 27. RETURN TO MAGNET TIME-OUT	This message appears at the completion of the rigid tap cycle if the spindle controller could not detect the magnet within 6 seconds while returning to home position.
10.3.29 28. ORIENTATION TIME-OUT OR TAPPING CALIBRATION FAILURE	This message appears if the spindle controller could not detect the magnet within 10 seconds, either during spindle orientation (M19, M6) or on the last rotation during rigid tap precycle.
10.3.30 29. RIGID TAP PRECYCLE ENCODER FAILURE (TOO MANY COUNTS)	See ERROR 7.
10.3.31 30. SURVEY CLEARED DUE TO BLANK OR CORRUPT SURVEY	This message appears during cold start if the checksum, calculated from reading the survey table, is not equal to the checksum stored in the survey table. The survey will be zeroed.

10.3.32 31. TIME-OUTON ORIENTATION

If the spindle magnet does not align within 10 seconds of an orient spindle then this error message is displayed.

10.3.33 32. SPINDLE WILL NOT STOP. CHECK INVERTER ZERO SPEED

If the spindle does not stop in 3 seconds from entering a spindle stop command then this error message is displayed.

10.3.34 33. ENCODER CHANNELS ARE REVERSED (RIGID TAP)

This message will display if the encoder connection for spindle motor is reversed at spindle control card.

11.0 COORDINATE SYSTEMS

11.1 MACHINE COORDINATE SYSTEM

The zero of the Machine Coordinate System (MCS) is the Cold Start Position. The Z axis cold start position is the tool change position. The coding of a G53 moves the machine to the programmed location in reference to the MCS. A G53 X0 Y0 always returns the machine to the MCS position. If a Tooling Coordinate System is not created, the CNC uses the MCS for the execution of G28, tool length offsets and the HO command. The tool change function (M6) automatically positions the Z axis to the zero of the MCS before executing the tool exchange.

11.2 TOOLING COORDINATE SYSTEM - HOME

The Tooling Coordinate System (TCS) is established as the X, Y, Z, A, B zero position. The SET(axis) or SETH command is used to establish this position (see section 8.0 COMMANDS, MAN-0131). The Z axis is generally positioned at the cold start position for safety. After entering the SETH command, the axes positions are displayed on the video screen as the ZERO. Tool length offsets are measured relative to the TCS. The TCS allows the operator to adjust all offsets simultaneously. The operator may jog the Z axis up or down and perform a SETZ to adjust all offsets. Fixture offsets are relative to the TCS.

The axes are positioned to the zero of this coordinate system when the programmer issues a G28. The default Tooling Coordinate System is the MCS (Cold Start).

Enter the command: SETCS, and HO to switch from the TCS to the MCS.

11.3 FIXTURE OFFSETS

Fixture offsets are used to establish a part home position different from the TCS. This is most commonly used to locate the home positions for multiple part operations. This may also be used to establish a part change position, with the TCS, and a part home position, with the fixture offset. The fixture offset gives the user the capability to individually adjust each fixture. After the TCS is set during the job setup procedure, the locations (X, Y and Z) of up to 24 fixtures may be set in the Fixture Offset Table, each offset being relative to the TCS (Home). To enter a fixture offset, see FO command section 8.0 COMMANDS, MAN-0131. The programmer references each offset, by use of the E word; E1 for offset #1; E2 for offset #2 etc. Fixture offsets 1-6 may also be referenced by programming a G54-G59; G54 for offset #1; G55 for offset #2 etc.

Programming a fixture offset in Format 1, without motion words, causes the axes to move to the current absolute position on the specified fixture. In programming Format 2, the move to the fixture is not made until an X, Y, or Z word is specified. The axis move can be specified in the same block or after the block containing the fixture offset code.

EXAMPLE:

Fixture offset 1 = X1. Y1. Z1.
Fixture offset 2 = X2. Y2. Z2.

```
N1 O1
N2 G90 G0 X0 Y0 Z0
N3 E1 X0 Y0
N4 G0 X1. Y-1.
N5 E2 X2.5 Y2.5
N6 Z2.5
N7 E0 X0 Y0
N8 Z0
```

Format 1 results of running the example above:

N2.X0.Y0.Z0.
Block 2 the machine is already at X0 Y0 Z0 so no motion occurs.

N3.X1.Y1.Z1.E1.G0
Block 3 causes movement in all three axes because with format 1 the motion for the fixture offset is applied immediately for all axes.

N4.X1.Y-1.G0
Block 4 will move to the X and Y value relative to the fixture offset 1 that has already been applied for each axis.

N5.X2.5Y4.5Z1.E2.G0
Block 5 causes movement in all three axes because with format 1 the motion for the fixture offset is applied immediately for all axes.

N6.Z2.5G0
Block 6 will move to the Z value relative to the fixture offset 1 that has already been applied for each axis.

N7.X-4.5Y-4.5Z-2.G0
Block 7 will cancel the fixture offset currently in effect. Both format 1 and 2 cancel the same way and cause motion immediately for all axes.

N8.Z-2.5G0
Block 8 no fixture offsets are in effect, the Z moves as normal.

Format 2 results of running the example above:

N2.X0.Y0.Z0.
Block 2 the machine is already at X0 Y0 Z0 so no motion occurs.

N3.X1.Y1.E1.G0
Block 3 causes movement only in X and Y because with format 2 the motion for the fixture offset is applied only in moves that have the axes called out.

N4.X1.Y-1.E1.G0
Block 4 causes movement only in X and Y because with format 2 the motion for the fixture offset is applied only in moves that have the axes called out. Note that the Z axis fixture offset has yet to be applied and moved.

N5.X2.5Y4.5E2.G0
Block 5 causes movement only in X and Y because with format 2 the motion for the fixture offset is applied only in moves that have the axes called out.

N6.Z4.5E2.G0
Block 6 finally the Z causes movement because the block has a Z axis motion called out.

N7.X-4.5Y-4.5Z-2.G0
Block 7 will cancel the fixture offset currently in effect. Both format 1 and 2 cancel the same way and cause motion immediately for all axes.

N8.Z-2.5G0
Block 8 no fixture offsets are in effect, the Z moves as normal.

11.4 PROGRAM COORDINATE SYSTEM

The Program Coordinate System (PCS) is established by the programmer at some convenient point relative to the work piece and fixture. The default PCS is the Tooling Coordinate System. The PCS is set by the use of the G92 (Absolute Preset) or G52 (Coordinate System Shift) code in the part program. If the programmer knows the location of the Tooling Coordinate System, the operator can use G92 with the X, Y, Z, etc. dimension words to set the PCS. The G92 is used to establish the location of the part home from the current location. When using a G92, after a G28 return to home, the G92 should be canceled. This is accomplished by moving to the TCS zero and coding a G92 X0 Y0. The G52 is used to shift the current home position by a specified absolute amount.

EXAMPLE:

The material is 12" X 12", one inch thick. The Tooling Coordinate System (TCS) is set at the X minus Y plus corner of the material. The desired Program Coordinate System (PCS) is at the center of the material.

N1 G0 G90 S2000 M3 X0 Y0   Locate to the TCS
N2 H1 M7 Z.1
N3 G92 X-6. Y6. Establishes the PCS at the center
N4 X2. Y3.

Block #1 establishes that the current position is at the TCS.

Block #3 establishes the PCS at the center of the material. No machine motion occurs.

Block #4 moves the X axis to 2. from the PCS and 8. from the TCS (this is the same position). The Y axis is moved to 3. from the PCS and -3. from the TCS (this is the same position). Once the PCS is established, the programmer may switch between absolute(G90) and incremental(G91) modes without affecting the PCS. The G28 causes the machine to return to the TCS zero position.

EXAMPLE: The material is 12" X 12", one inch thick. The Tooling Coordinate System (TCS) is set at the X minus Y plus corner of the material. The desired Program Coordinate System (PCS) is at the center of the material.

N1 G0 G90 S2000 M3 X0 Y0 Locate to the TCS
N2 H1 M7 Z.1
N3 G52 X6. Y-6. Establishes the PCS at the center
N4 X2. Y3.

Block #1 establishes that the current position is at the TCS.

Block #3 establishes the PCS at the center of the material. No machine motion occurs.

Block #4 moves the X axis to 2. from the PCS and 8. from the TCS (this is the same position). The Y axis is moved to 3. from the PCS and -3. from the TCS (this is the same position). Once the PCS is established the programmer may switch between absolute (G90) and incremental (G91) modes without affecting the PCS. The G52 is canceled by coding a G52 X0 Y0. This returns the machine to the TCS on the next motion word line.

12.0 ROTARY AXES

12.1 A AXIS

12.1.1 DIRECTION OF MOTION

Direction of motion is defined as per ANSI/EIA RS-274-D. Since the VMC rotates the work piece, the rotary head must rotate clockwise to achieve a counterclockwise tool motion and visa versa.

The positive sign is assumed. The Negative symbol must precede the angular amount. A-90. is correct, -A90. is incorrect.

A+ = Counterclockwise work piece rotation (viewing in the X+ direction).

A- = Clockwise work piece rotation (viewing in the X+ direction).

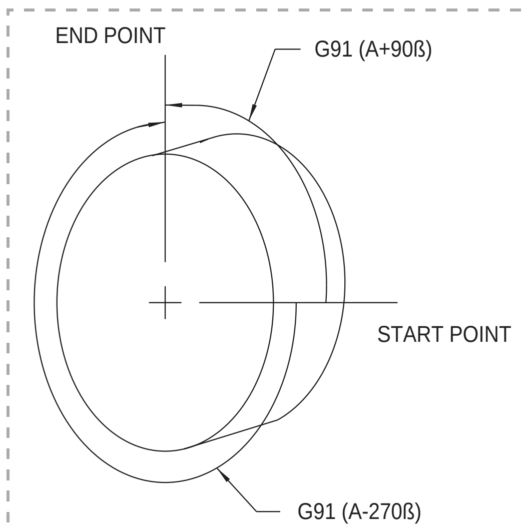


Figure 12-1: Direction of Motion

12.1.2 G90 ABSOLUTE MODE

In the absolute mode (G90) the A word value defines the final position between 0 and 360. The + or - signs define the direction of rotation. The + sign causes counterclockwise work piece rotation, the - sign causes clockwise rotation to the indicated degree. There are two possible ways to get to a position on the rotary table in absolute: by rotating either from the positive or negative direction.

EXAMPLE:

1. G90: In absolute terms A+90. and A-90. will position to the same location; however, from different directions. An A+90. will rotate the work piece in the positive (CCW)

direction to the absolute 90 degree location. An A-90. will move the table in the negative (CW) direction to the absolute 90 degree location.

N1 G0 G90 A90.000
N2 A-0

Block N1 rotates the work piece counterclockwise to the 90th degree position.

Block N2 rotates the work piece clockwise to the zero degree position. Changing N2 to a value of A+0 would cause counterclockwise rotation to the zero degree.

12.1.3 G91 INCREMENTAL MODE

In the incremental mode (G91) the A axis value defines the direction and number of degrees to rotate. An A+90. causes the work piece to rotate counterclockwise 90 degrees. If the next move were an A-90. the work piece would rotate clockwise to the original position.

EXAMPLE:

1. G91: In incremental terms A+90. and A-270. will position to the same location; however, from different directions. An A+90. will move the table in the positive (CCW) direction 90 degrees from its current position. An A-270. will move the table in the negative (CW) direction 270 degrees from its current position.

N1 G0 G91 A+90.
N2 A-90.
Block N1 rotates the work piece counterclockwise 90 degrees. Block N2 will rotate the work piece clockwise 90 degrees.

A AXIS COLD START

The A axis markers must be aligned during the cold start procedure. The mark on the face plate and the mark on the body of the A axis must be in line before using the CS command.

A AXIS HOME POSITION

The A axis can be set to zero at any angle by using the SETA command. The SETA command will store the current A axis position into memory as the A axis home position. When the CS command is used and the move to home question appears the A axis position appears, at the bottom of the screen, along with the XYZ and B axis positions.

EXAMPLE:

1. Jog the fixture or part until it is level.
2. From the command mode, type SETA, then press ENTER.
3. The A axis should now be set to zero.

12.1.4 MOVE TO HOME

The G28 code in format one and the HO command return the rotary head to the set point along with the X and Y axes. In format two, a G90 G28 X0 Y0 A0 needs to be coded so that the A axis will move home with the X and Y axes.

1. If the current Z axis position is above (+) the HOME position, the X, Y and A axis moves to zero first, then the Z axis moves in the negative direction.
2. If the current Z axis position is below (-) the HOME position, the Z axis moves in a positive direction (to zero) first, then the X, Y, and A axis move to zero.

After the HO 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.

> **WARNING**
> This move will always be in the direction that is the shortest distance back to A0.

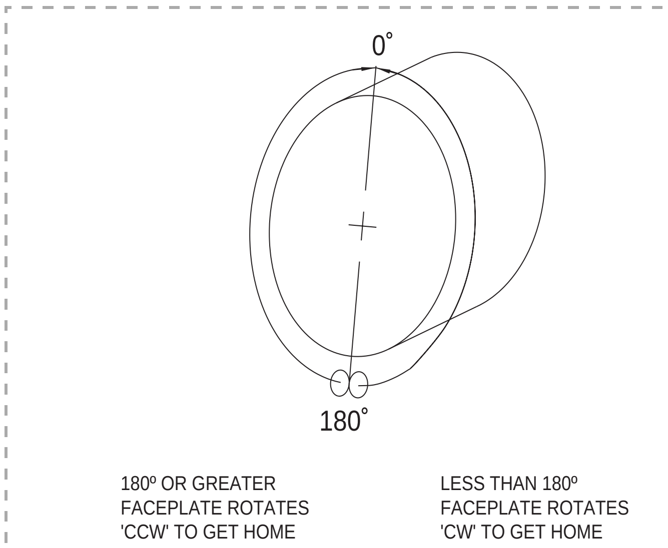


Figure 12-2: Move to Home

12.1.5 JOG MODE

Jogging the A axis in the positive direction causes the work piece to rotate counterclockwise when viewing in the X+ direction. Relative tool motion would be clockwise. Jogging the A axis in the negative direction causes the work piece to rotate clockwise when viewing in the X+ direction. Relative tool motion would be counterclockwise.

A AXIS BRAKE

Use the M60 code to activate the A axis brake. The brake will remain activated until the M61 is coded or another A axis move is made, either from the program or jog.

EXAMPLE: *N1 G90 G0 M61 Y-2.3 X1.3 Brake off*

N2 Z1.6754 A90. M60 Z and A axis move then brake on
An M-60 is used with an A move when using fixed cycles so that the cycle will not execute until the brake is applied.

EXAMPLE:

N10 G82 G99 R0+.1 Z-.25 F45. P130
N11 A30. M-60
N12 A60. M-60
On line N11 and N12 the A axis moves into position, the brake is applied, then the cycle is executed.

DECIMAL DEGREES

A axis values are given in decimal degrees.

EXAMPLE:

d° = Degrees
m’ = Minutes
s" = Seconds
DD = Decimal degrees
40°30’13" = 40.50361
d° m’ s" = DD
DD= d° + ((m’ + (s" / 60)) / 60))

12.1.6 NUMERICAL FORMAT

No more than seven numbers are allowed.

A##.#####

A###.####

A####.###

> NOTE
> The decimal point is required for all angles except for A0.

12.1.7 MAXIMUM & MINIMUM ANGULAR LIMITS

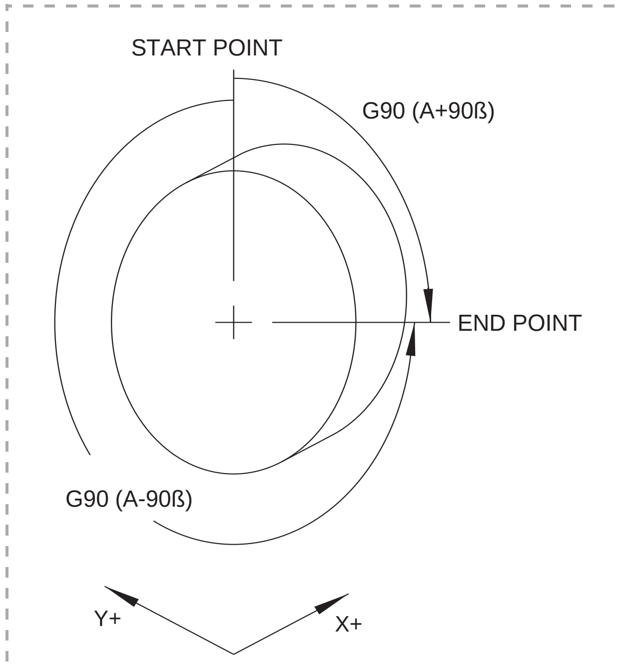


Figure 12-3: Max. & Min. Angular Limits

Smallest: A.002

Largest: A1080.

Programming a G91 G1 X2. A1080. will cause an interpolated move where the X axis moves 2. and the A axis moves 1080. degrees. When a fixed cycle is used, A axis motion will cause execution of the cycle.

EXAMPLE:

```
N1 M6 T1
N2 (TOOL #1, DRILL
N3 G0 G90 S10000 M3 E1 X0 Y0 A30.
N4 H1 D1 M7 Z.1
N5 G81 G99 R0.1 Z-.5 F40. X.5
N6 X2.5
N7 A90.
N8 X.5
N9 A150.
N10 X2.5
N11 A210.
N12 X.5
N13 A270.
N14 X2.5
N15 A330.
N16 X.5
N17 G80
N18 M5 M9
N19 G0 G90 H0 Z0
```

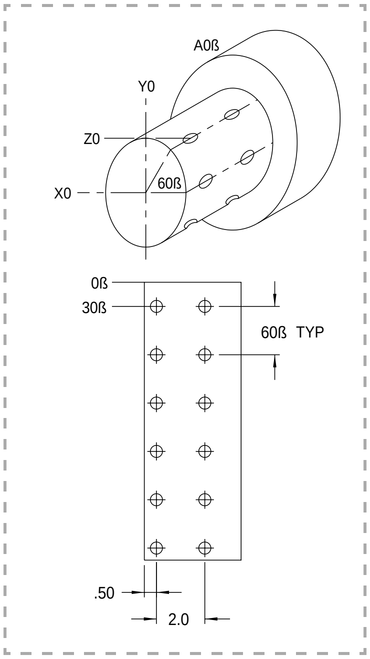


Figure 12-4:

The A axis can be interpolated along with any other axis. For example, an X and A axis move can be programmed on one line using the G1 code. The X and A axis moves will both end at the same time.

> **NOTE**
> A G2 or G3 will not accept an A axis move in the same line. (See Flat Cam Programming).

EXAMPLE:

N18 M6 T3
N19 (TOOL #3, 3/8 (.375) 2FL EM
N20 G0 G90 S8000 M3 E1 X3.5 Y0 A0
N21 H3 D3 M7 Z.1
N22 G1 Z-.27 F10.
N23 G91 F50.
N24 X1.
N25 X-1.A-30.
N26 X-1.A30.
N27 A90.
N28 X2.
N29 A-90.
N30 X1.
N31 Z.1 G0
N32 M5 M9

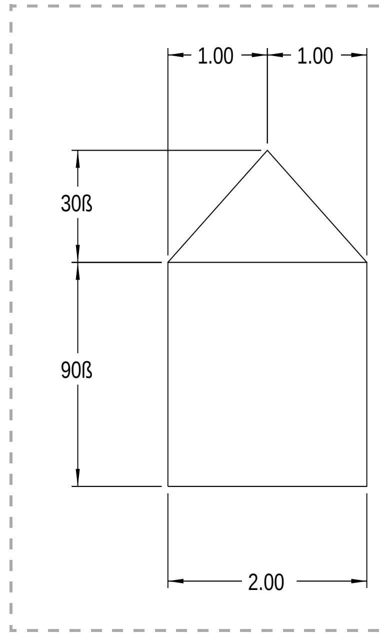


Figure 12-5:

12.1.8 DEGREE FEEDRATE CALCULATION

The actual move distance must be calculated before the feed rate can be determined. This distance may be estimated; however, for best results use the equation below.

The following example is programmed for a part with a diameter of 4.0. The desired cut is 90 degrees interpolated with an X axis move of 3.0 (see line N6 of the example program below). The feed rate used for calculation was 25 ipm.

Actual move distance = Sqrt (((Dia. of surface to be cut * 3.14159) / (360 / Degrees of rotation))2 + X2)

EXAMPLE:

Actual move distance $= \text{Sqrt}(((4 * 3.14159) / (360 / 90))^2 + 3.02)$
$= \text{Sqrt}((12.56636) / 4)^2 + 9)$
$= \text{Sqrt}((3.14159)^2 + 9)$
$= \text{Sqrt}(9.8696 + 9)$
$= \text{Sqrt}(18.8696)$
$= 4.3439$

(G94) Feed Rate = Degrees of Rotation / (Actual Move Distance / Desired Feed Rate)

EXAMPLE: *Feed rate = 90 / (4.3439 / 25)*
= 90 / .173756
= 517.97

EXAMPLE: G94

N1 M6 T1
N2 (TOOL #1, 1/2 END MILL
N3 G0 G90 S5000 M3 E1 X0 Y0 A0
N4 H1 M8 Z.1
N5 G1 Z0 F25.
N6 X3. A90. F517.97
N7 X1. 25 Inches per Minute move
N8 X3. A90. 517.97 Degrees per Minute
N9 G90 G0 H0 Z0

NOTE
The machine control default is G94, therefore it is not required to code the G94 into the program if degrees per minute is used.

NOTE
An F word, on a line with an A axis move only, represents degrees per minute.

12.1.9 FEED RATE SPECIFICATION IN DEGREES PER MINUTE

Table 12-1: Feed Rate Specification

AXIS RATIO	MAXIMUM RAPID TRAVERSE	MAXIMUM PROGRAMMABLE FEED RATE
360:1	2000	1250
180:1	4000	2500
120:1	6000	3750
90:1	8000	5000
72:1	10000	6250

12.1.10 G93 - 1/T (INVERSE TIME)

The G94 code is used more commonly than the G93. G93 was used by controls that did not interpolate XA or YA moves. The advantage of using G94 (the default code) is to allow the programmer to switch between ipm and dpm with no code cancellation. Except for code cancellation, G94 and G93 effectively cut the A axis interpolated move in the same way.

The following example is programmed for a part with a diameter of 4.0. The desired cut is 90 degrees interpolated with an X axis move of 3.0 (see line N6 of the example program below). The feed rate used for calculation was 25 ipm.

Actual move distance = Sqrt (((Dia. of surface to be cut * 3.14159) / (360 / Degrees of rotation))2 + X2)

EXAMPLE: *Actual move distance = Sqrt (((4 * 3.14159) / (360 / 90))2 + 3.02)*
= Sqrt ((12.56636) / 4)2 + 9)
= Sqrt ((3.14159)2 + 9)
= Sqrt (9.8696 + 9)
= Sqrt (18.8696)
= 4.3439

(G93) Feed Rate = 1 / (Actual Move Distance / Desired Feed Rate)

EXAMPLE: *Feed rate = 1 / (4.3439 / 25)*
= 1 / .17376
= 5.755

EXAMPLE: G93

```
N1 M6 T1
N2 (TOOL #1, 1/2 END MILL
N3 G0 G90 S5000 M3 E1 X0 Y0 A0
N4 H1 M8 Z.1
N5 G1 Z0 F25.
N6 G93 X3. A90. F5.76
N7 G94
N8 X1.
N9 G93 X2. A30. F11.07
N10 G94
N11 G90 G0 H0 Z0
```

All moves in this example from N6 to N9 are equivalent to a feed rate of 25 ipm linear interpolation.

12.1.11 G15 - YZA CIRCULAR INTERPOLATION

This code is used to interpolate Y, Z arcs while making simultaneous A axis movement. This code is used when it is necessary to cut an arc with the bottom of the end mill. G15 may also be used with a ball nose end mill.

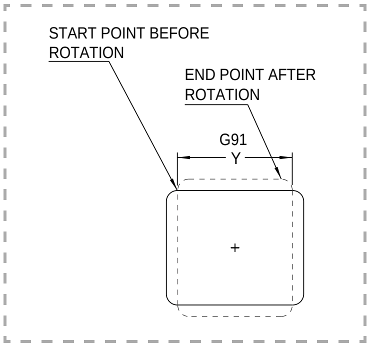


Figure 12-6:

> **NOTE**
> When the center of the arc to be cut is not the same as the center of rotation on the A axis, G15 must be used to cut the radii (see picture below).

The following diagram shows the information required to program G15.

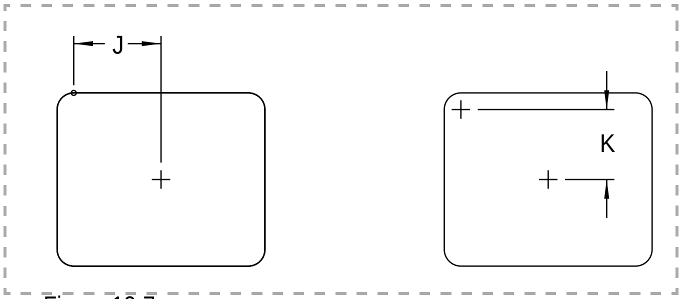


Figure 12-7:

The G15 arc requires four descriptors: end point, center description, A axis rotation, and the G2 or G3 code. These four elements are determined as follows:

1. End point description:

   a. Incrementally Y and Z axis moves are described as the direction and distance from the start point, prior to A axis rotation, to the end point, after the A axis rotation.
   b. In absolute terms the Y and Z axis end positions are given as the absolute locations after rotation.

2. Center description:

   a. The J is the incremental Y direction and distance from the Y axis start point to the center of A axis rotation.

b. The K is the incremental Z direction and distance from the center of the arc to be cut to the center of A axis rotation.

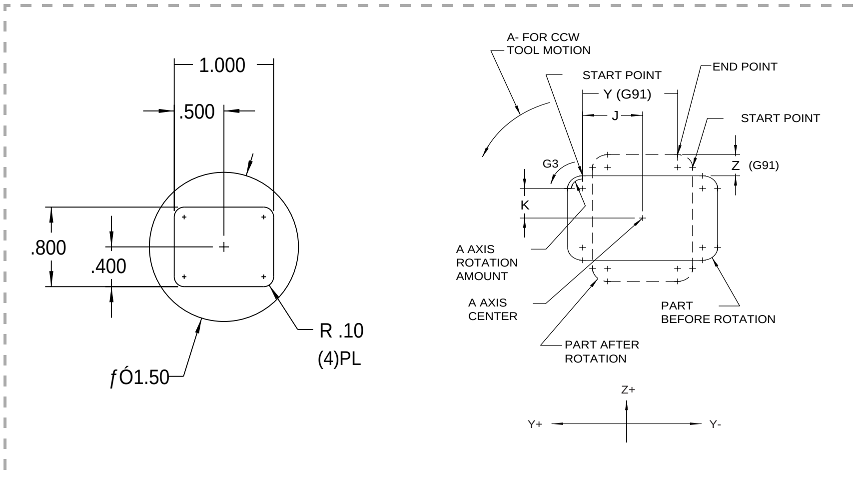


Figure 12-8:

3. A axis description:

   a. Incrementally the A axis rotation is given as the angle between the end point at the beginning of the move and the end point at the end of the move (See picture below).
   b. In absolute terms this would be given as the absolute angle of the part at the end of rotation.

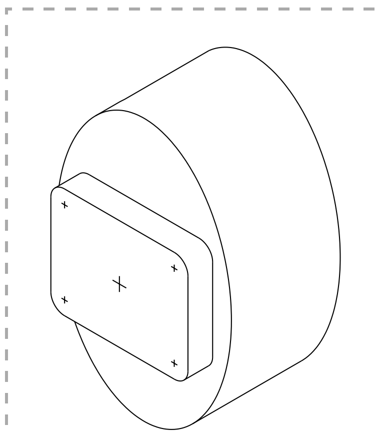


Figure 12-9:

4) G2 or G3 description:

   a. The arc direction coding is based on viewing the part looking in the X+ direction. A clockwise arc uses a G2 code and a counterclockwise arc uses a G3.

EXAMPLE:

```
N1 (G15 RECTANGLE
N2 (X0 IS END OF PART, Y0 IS CENTER, Z0 IS TOP OF 1.5 DIA PART
N3 G90 G0 X.3 Y-1.5 E1 A0 S5000 M3
N4 H1 Z-.35 M8
N5 G91 G8 G1 Y1.9F10.
N6 G15
N7 Y-.7 Z.1 J-.4 K-.3 A-90. G3
N8 G90 Y.6
N9 G91 Y-.7 Z-.1 J-.3 K-.4 A-90. G3
N10 G90 Y.8
N11 G91 Y-.7 Z.1 J-.4 K-.3 A-90. G3
N12 G90 Y.6
N13 G91 Y-.7 Z-.1 J-.3 K-.4 A-90. G3
N14 M5 M9
N15 G90 G0 H0 Z0
N16 E0 X0 Y0 A0
N17 M2
```

EXAMPLE:

```
N1 (G15 CRANKSHAFT
N2 (X0 IS END OF PART, Y0 IS THE CENTER, Z0 IS THE TOP OF 1.5 DIA
N3 (TOOL IS A .5 DIA CENTER CUTTING 2FL EM
N4 G90 G0 S5000 M3 E1 X.45 Y0 A0
N5 H1 Z.1 M8
N6 Z-.05 G1 F10.
N7 G15
N8 G91
N9 Y0 Z0 J0 K-.45 A-360. G3
N10 G90
N11 Z.1 G0
N12 X1.15 A180.
N13 Z-.05 G1
N14 G91
N15 Y0 Z0 J0 K-.45 A-360. G3
N16 G90 G0 Z.1
N17 M5 M9
N18 G90 G0 H0 Z0
N19 E0 X0 Y0 A0
N20 M2
```

12.1.12 FLAT CAM (CAM WRAPPING) PROGRAMMING

Flat cam programming is used when an XY program needs to be “wrapped” around the circumference of the part. This function is designed to convert Y axis motion into A axis motion. XA conversion is used when the A axis is the rotary table, YB conversion is used when the B axis is the rotary table.

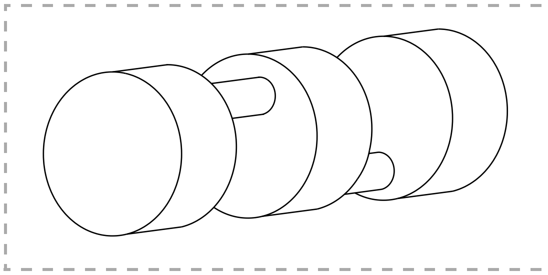


Figure 12-10: Flat Cam

The conversion from Y to A axis moves is defined in the program by using a G17 and Q word in the same line. The Q word represents a number used by the control for converting the Y or X axis moves to A or B axis moves.

WRAPPING X ON B AXIS

Y axis wrapping is assumed. Use G17 Q word P1 on the same line in the program for YB wrapping. All information for Y to A wrapping applies to X to B wrapping.

CAM DIAMETER

The cam diameter to be used is usually given on the blueprint. If it is not given, use the outside diameter of the part for the cam diameter. Y axis moves, when wrapped, are measured along the circumference of the cam diameter.

> **NOTE**
> The angular move for one inch of Y axis motion for CAM DIA. 1, is less than the angular motion for one inch of CAM DIA. 2 (see the picture above).

A AXIS RATIO

Each rotary table has an axis ratio. Not all manufactures keep the same ratio. This number is affected by the gear ratio of the axis. The manual for each rotary head contains the A axis ratio number to use for calculating the Q word.

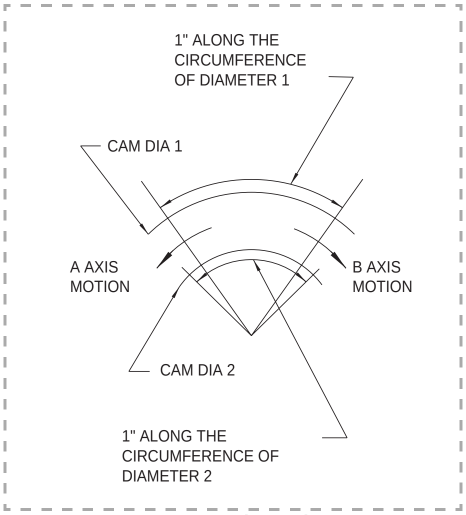


Figure 12-11: A Axis Ratio

Q WORD

A axis ratio = 90 to 1, cam diameter = 4., pi = 3

.14159

(Inches) Q = A axis ratio / (5 * pi * (cam diameter in inches)) 5= counts constant

Q = 90 / 5 * 4 * 3.14159

Q = 90 / 20 * 3.14159

Q = 90 / 62.8318

Q = 1.4324

(Metric) Q = A axis ratio / ((5/25.4) * pi * (cam diameter in millimeters))

UNWRAPPING

Prior to canceling the G17 Q word code, the Y axis must be “unwrapped” so that the A axis can return to its original position.

1. The position of the Y axis, when the G17 Q word is coded, establishes the original position of the A axis.

2. Returning the Y axis to its original position should return the A axis to its original position. The A axis position read out on the screen must be the same position that it started from to be fully unwrapped. The Q word can be altered to attain this.
3. An absolute or incremental Y axis move can be used to return the Y axis to its place of origin.

G90

If the program moves the Y axis Y+5., then the unwrapping move would be Y0.

G91

If the program moves the Y axis Y+5., then the unwrapping move would be Y-5.

CANCELING CAM WRAPPING

Directly after the unwrapping move, cancel the flat cam programming function by coding a G17 on a line by itself.

EXAMPLE:

N1 O1 (FLAT CAM PROGRAM EXAMPLE
N2 M6 T1
N3 (TOOL #1, 1/2 2 FL E.M. USE .5 IN THE TOOL TABLE
N4 G0 G90 S2000 M3 X0 Y0 A0
N4.5 G51.1 Y0 It is important to mirror the Y axis
N5 H1 D1 M8 Z.1
N6 G17 Q1.4324 This line starts Flat cam conversion (see Q word)
N7 X1.125 Y-2.125
N8 G1 Z-.27 F25.
N9 G1 G42 X2.125 To maintain a climb cut on a mirrored path, use G42
N10 Y-.25
N11 X1.875 Y0 I-.25 G3
N12 X.25
N13 X0 Y-.25 J-.25 G3
N14 Y-4.5
N15 X.322 Y-5.2437 I1.02 G3
N16 X.625 Y-5.9437 I-.657 J-.7 G2
N17 Y-6.5
N18 X2.125 I.75 G3
N19 Y-3.375
N20 X1.125 G40
N21 Z.1 G0
N22 Y0 Return to original Y position (unwrapping move)
N22.5 G50.1 Turn off mirror
N23 G17 This line cancels the Flat cam conversion
N24 M5 M9
N25 G90 G0 H0 Z0

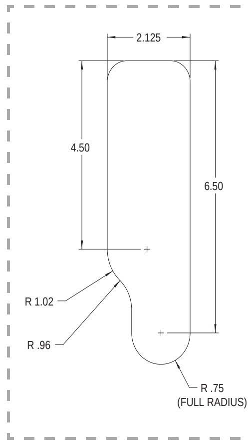


Figure 12-12:

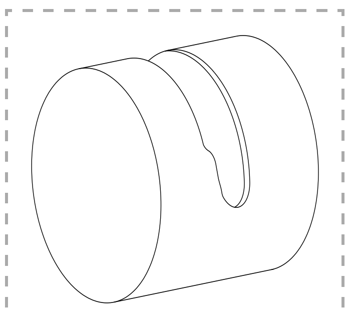


Figure 12-13:

> **NOTE**
> Rapid movements (G0 or G5) are reduced in speed during the flat cam conversion.

400 / Q word amount = new rapid rate

400 / 1.4324 = 279.25 IPM

MID PROGRAM START

Mid program starts **may not** be executed after the G17 Q word which is used to start the cam wrapping feature. The mid program start feature can be used on any line before the G17 Q word and after the G17 that is used to cancel cam wrapping.

WRITING A CAM WRAPPING PROGRAM

When writing a program to be wrapped, mirror the Y axis with a G51.1 Y0 at the beginning of the program, just before the G17 Q#, and exchange all G41 codes to G42. (See the program example above.)

12.1.13 CAM WALL ANGLES

A axis machining can produce two different cam wall angles. The cam walls may intersect or they may be parallel. These wall angles are determined by the programming methods used. The wall configuration required is established by the part blueprint.

> **NOTE**
> Flat cam conversions will produce walls that intersect (see picture below).

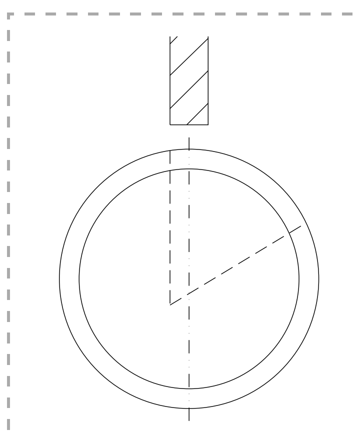


Figure 12-14:

Review the print to determine whether the walls on the print intersect or are parallel. Place a straight edge on a wall on the print to help determine if the walls intersect.

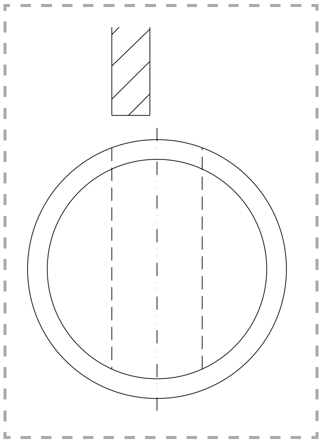


Figure 12-15:

> NOTE
> If the walls are parallel, DO NOT use flat cam (cam wrapping) conversions (see picture above).

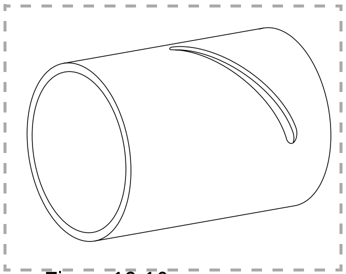


Figure 12-16:

Parallel walls are normally associated with slots or grooves. These parallel walls can be maintained by using a cutter that is the same diameter as the width of the slot. When the slot is wider than the cutter, Y axis movements must be made to maintain parallel walls.

Parts that have parallel walls, such as slots or grooves, are usually parallel because a follower pin must fit into the slot or groove.

1. Start cutting the slot by programming the center of the groove, and cut with an undersized cutter. This cutter must be undersized enough to account for cutter deflection.
2. Use the same programmed path to make a second cut with a full- sized cutter.

Parts with parallel walls can be cut by using a cam system that provides this feature. These systems allow for the Y axis to be offset to account for cutter radius compensation, if required.

12.2 TILT ROTARY TABLE

12.2.1 DIRECTION OF MOTION

Direction of motion is defined as per ANSI/EIA RS-274-D. Some tilt rotary tables are set up with the B axis as the tilt portion and others with the A axis as the tilt portion. All rotary table information for a tilt rotary table can be read in the A axis portion of this section. Since the VMC tilts the **work piece**, the tilt must be clockwise to achieve counterclockwise **tool** motion and visa versa.

> NOTE
> The positive sign is assumed. The negative symbol must precede the angular amount. B-90. is correct, -B90. is incorrect.

B TILT TABLE

B+ = Counterclockwise work piece rotation

(viewing in the Y+ direction).

B- = Clockwise work piece rotation

(viewing in Y+ direction).

A TILT TABLE

A+ = Counterclockwise work piece rotation

(viewing in the X+ direction).

A- = Clockwise work piece rotation

(viewing in X+ direction).

TILT COLD START

The tilt table axis markers must be aligned before using the CS command.

TILT HOME POSITION

Jog the tilt table to the desired degree for home position. Then use the SETB or SETA command, whichever applies, to establish the tilt home position. This position is stored in memory and when the next CS command is used, the tilt table will return to this location.

AXIS LIMITS

The tilt limits are as follows:

1. B tilt table:
   - 105 degrees - from the cold start position
   - 15 degrees + from the cold start position
2. A tilt table:
   - 105 degrees + from the cold start position
   - 15 degrees - from the cold start position

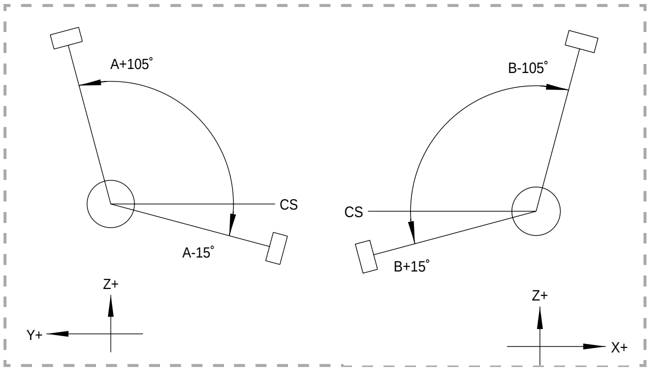


Figure 12-17:

12.2.2 TILT TABLE BRAKE

To activate the brake for the tilt table use the M62 code. When the next tilt axis move is made, the brake will automatically release. M63 can be used to release the tilt table brake.

EXAMPLE:

N1 G90 G0 M63 Y-2.3 X1.3 Brake off
N2 B+315. M62 B tilt move then brake on

12.2.3 G90 ABSOLUTE MODE

In the absolute mode, the tilt angular value defines the final position between 15 and 265 degrees. The + or - signs define the direction the table will move to get to the degree of tilt. Care must be used in selecting the proper sign for tilt to prevent over travel.

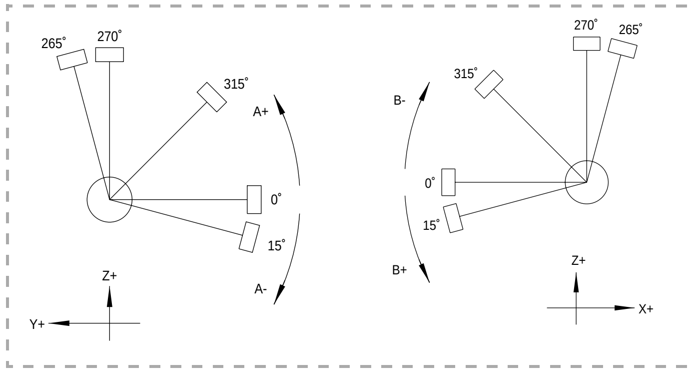


Figure 12-18: G90 Absolute Mode

EXAMPLE: In absolute terms B+10. will tilt the B axis counterclockwise to the 10th degree.

In absolute terms B-270. will tilt the B axis clockwise to the 270th degree.

In absolute terms A-10. will tilt the A axis clockwise to the 10th degree.

In absolute terms A+270. will tilt the A axis counterclockwise to the 270th degree.

12.2.4 G91 INCREMENTAL MODE

In the incremental mode (G91) the tilt value defines the direction and number of degrees for the tilt table to move. A B+10. causes the table to tilt positively (CCW) 10 degrees from its current position. A B-10. causes the table to tilt negatively (CW) 10 degrees from its current position.

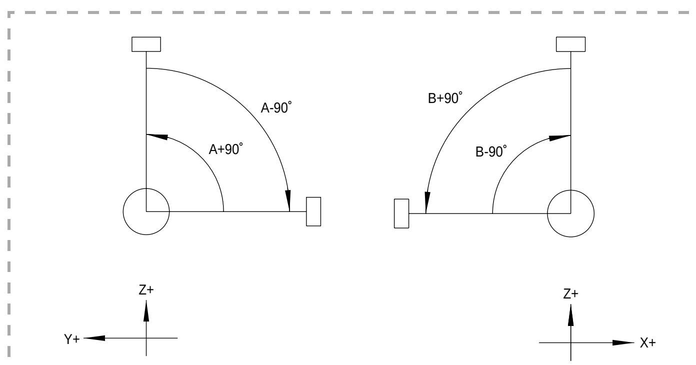


Figure 12-19: G91 Incremental Mode

EXAMPLE: In incremental terms B+90. tilts the table 90 degrees counterclockwise from its current position.

In incremental terms B-90. tilts the table 90 degrees clockwise from its current position.

In incremental terms A+90. tilts the table 90 degrees counterclockwise from its current position.

In incremental terms A-90. tilts the table 90 degrees clockwise from its current position.

FEED RATE

Feed rate is addressed by use of the F word and a G01 code. Tilt motion is programmed in degrees per minute. For example; G91 G01 B+45. F50.0 rotates the B axis 45 degrees at 50 degrees per minute. Use the following chart to find the maximum programmable feed rate for selected device.

See the degree feed rate calculation information in this section for details.

12.2.5 A & B FIXTURES OFFSETS

A & B Fixtures offsets are relative from the E0 or SETA, SETB position. This fixture offset value (in the fixture table) is an absolute value relative to zero. Within the program the direction of motion is specified by + (positive) or - (negative).

13.0 INTERPOLATION

13.1 VMC INTERPOLATION MODES

13.1.1 LINEAR INTERPOLATION

Linear Interpolation is used to generate motion along a line, at a specified feed rate. The linear mode is established by the G1 code. The VMC control can move up to 5 axes simultaneously, completing the movement at a point determined by the X, Y, Z, A, and B words.

EXAMPLE:

N1 O1
N2 M6 T1
N3 G0 G90 S2000 M3 E1 X0 Y0 A0 B0
N4 H1 Z.1 M8
N5 G1 Z-.25 F5.
N6 G1 G91 X1. F10.
N7 X1. Y1.
N8 X1. Y1. Z1. A360. B90.

According to the sample program above: Block N6 moves the X axis linearly (G1 mode) 1.0 inch at a feed rate of 10 IPM. Block N7 moves the X and Y axes together forming an angular cut. Block N8 moves all possible axes together. The G1 code will use the last feed rate established in the program with the F# word. The F# word is modal and is only canceled by another F# word. The F# will remain in effect throughout the program until another F# word is used. The F# word can appear on any line with other codes as long as the other codes have no restrictions.

G1 is modal and is only canceled by a G0 code. The G1 must be used again after using a G0 code in the program. A G2/G3 code will not cancel a G1 code. This means that if a G2 or G3 is used it is not necessary to re-state the G1 on the following line. Also, if the arc center is not described, then a straight line will be generated.

> NOTE
> Maximum program feedrate at 100% is 400 IPM for machines capable of 900 IPM rapid traverse. The maximum program feedrate at 100% is 250 IPM for machines capable of 400 IPM rapid traverse.

13.1.2 CIRCULAR INTERPOLATION

Circles and arcs are described by the start point, the end point, direction and distance from the start point to the center of the circle, and the arc direction. Any arc or circle may be programmed with two lines.

> NOTE
> The largest programmable radius is 399.9999 inches and with CRC 99.9999 inches.

1. The first line represents the move to position to the start point of the arc. This move could be a rapid move, a linear move, or the end of another circular move.

2. The second line (the circular move) includes the end point, arc center description, and arc direction. This move is made at the current feed rate.

XY (end point), IJ (arc center), G2/G3 (arc direction)

The end point, arc center, and direction around are described on the next three pages.

> NOTE
> Maximum program feedrate at 100% is 400 IPM for machines capable of 900 IPM rapid traverse. The maximum program feedrate at 100% is 250 IPM for machines capable of 400 IPM rapid traverse.

13.1.3 END POINT

The end point is described differently for G90 and G91. In both cases either the X or Y or both X and Y can be used. Similar to a linear move or rapid move, only the axis that has a difference between the start point and the end point must be included in the end point description.

EXAMPLE:

G90

In the absolute mode, the end point is described as the absolute distance from the home position of the part to the end point of the arc.

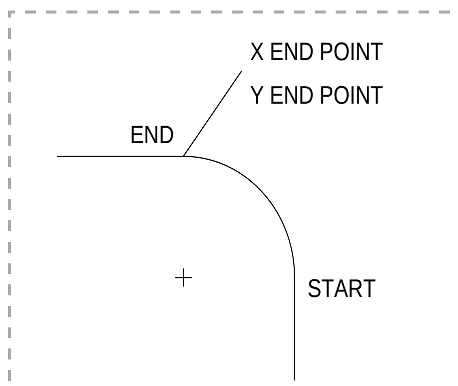


Figure 13-1: G90

G91

In the incremental mode, the end point is described as the direction and distance from the start point of the arc to the end point of the arc.

Put a pencil on the starting position of the arc and draw arrows, first horizontally along the X axis then vertically along the Y axis to the end point to help visualize the direction from start to end.

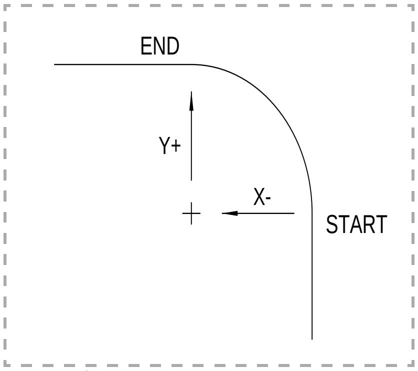


Figure 13-2: G91

13.2 ARC CENTER

13.2.1 G90 AND G91

The arc center description is the same for G90 and G91.

The arc center is the direction and distance from the start point of the arc to the center point of the arc. Put a pencil on the starting position of the arc and draw arrows, first horizontally along the X axis then vertically along the Y axis to the center point to help visualize the direction from start to center.

For the X axis direction description to the center use the letter I.

For the Y axis direction description to the center use the letter J.

If the circle begins at the start of a quadrant, only an I or J is required. If the circle begins within the quadrant, both I and J are required.

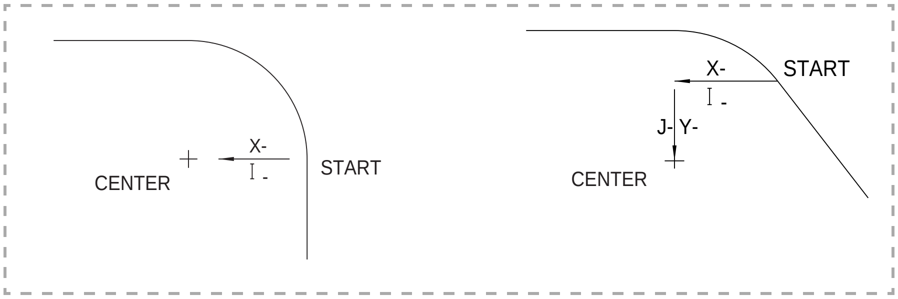


Figure 13-3:

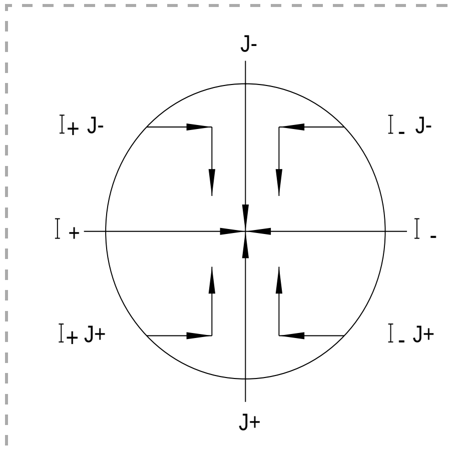


Figure 13-4: Arc Center

13.3 ARC DIRECTION

The G2 code is used when the arc direction is clockwise, G3 is used for arcs in a counterclockwise direction.

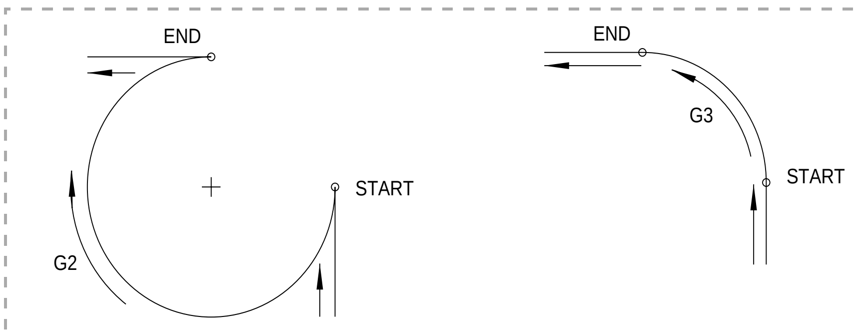


Figure 13-5: Arc Direction

13.4 CIRCLE EXAMPLES

The following are examples of various programmed arcs in absolute (G90) and incremental (G91)modes.

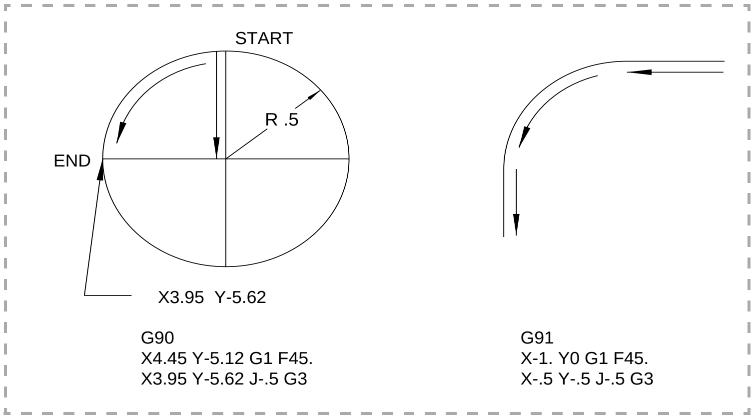


Figure 13-6: **Circle Examples (1)**

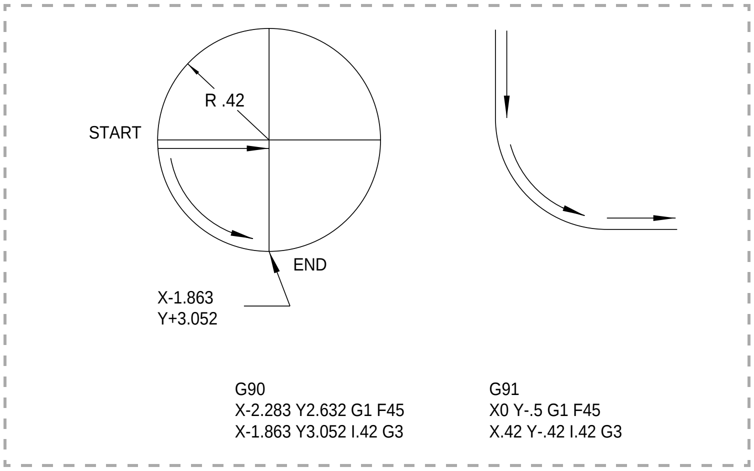


Figure 13-7: **Circle Examples (2)**

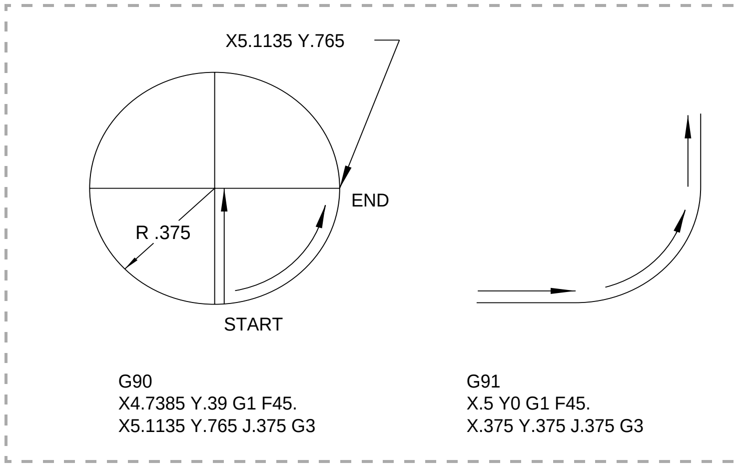


Figure 13-8: Circle Examples (3)

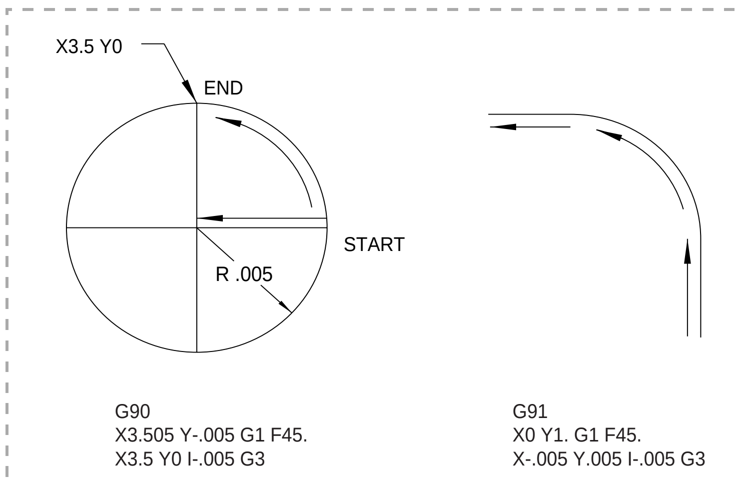


Figure 13-9: Circle Examples (4)

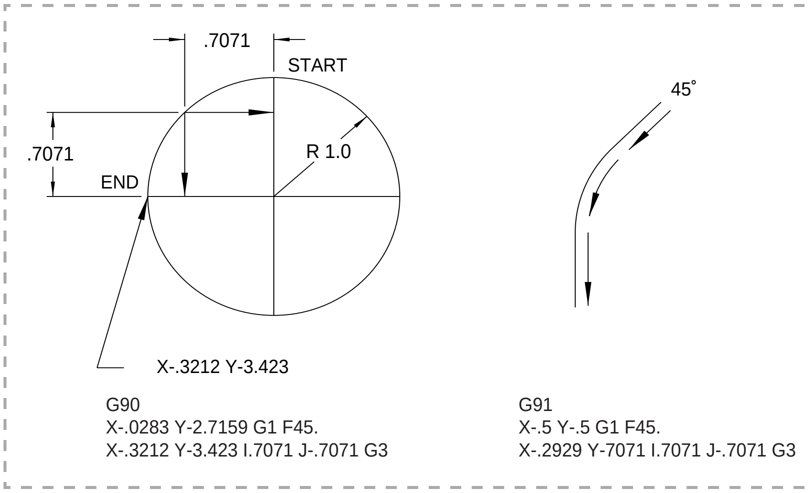


Figure 13-10: **Circle Examples (5)**

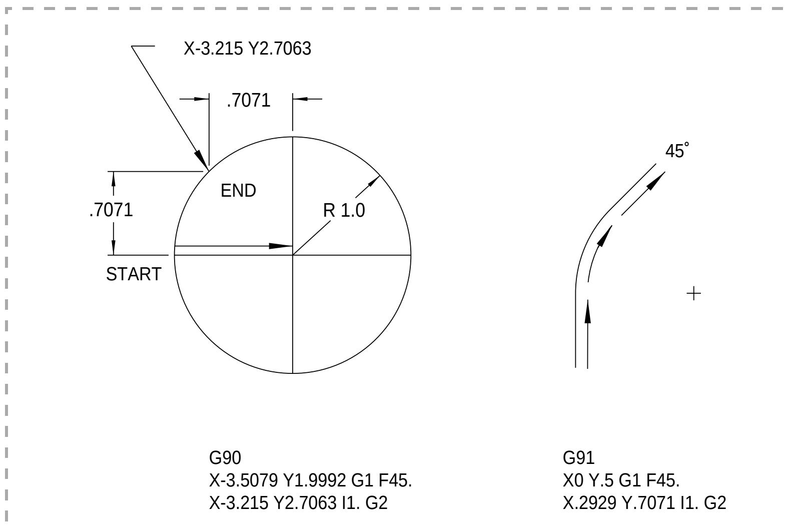


Figure 13-11: **Circle Examples (6)**

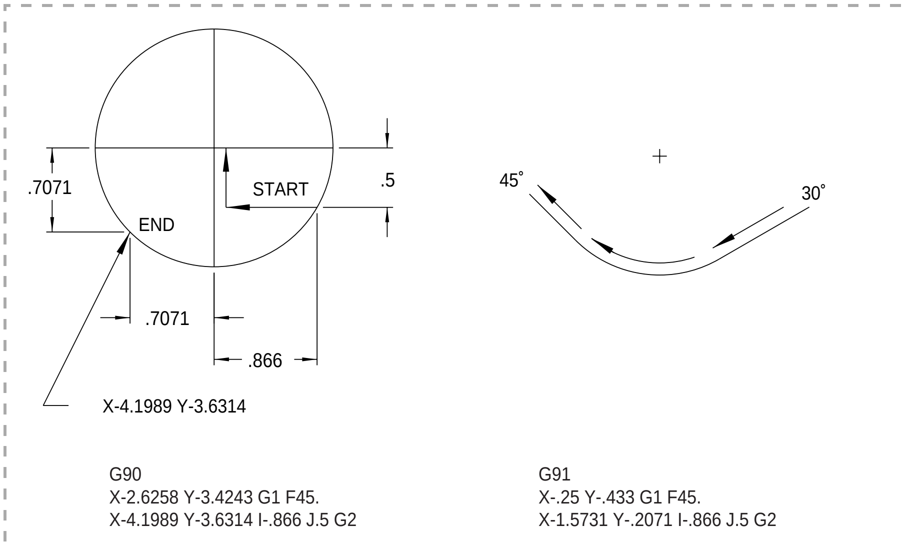


Figure 13-12: **Circle Examples (7)**

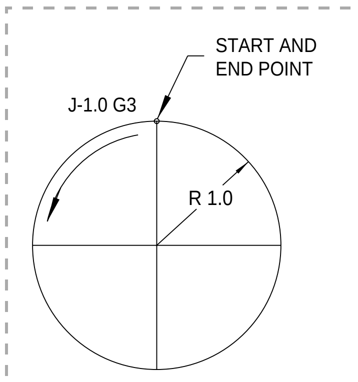


Figure 13-13: **Circle Examples (8)**

Full circles require the move to the start/end point and either the I or J, depending on the start point moved to, and the direction around.

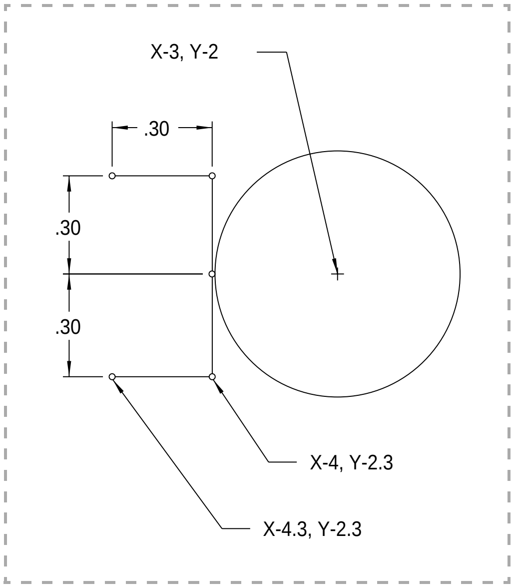


Figure 13-14: **Full Circles**

13.4.1 CIRCULAR BOSS EXAMPLE

M5 M9
G90 G0 H0 Z0
M6 T3 (TOOL #3, .5 2FL EM
G90 G0 S8000 M3 E1 X-4.3 Y-2.3
H3 D3 Z.1 M8
G8
Z-.245 G1 F10.
X-4. G41 F50.
Y-2.
I1. G2
Y-1.7
X-4.3 G40
Y-2.3 Z-.25
G91 (G91 USED FOR EXAMPLE ONLY
X.3 G41
Y.3
I1. G2
Y.3
X-.3 G40
M5 M9
G90 G0 H0 Z0

M6 T4

13.4.2 CONTOURED SLOT EXAMPLE

M5 M9
G90 G0 H0 Z0
M6 T6 (TOOL #6, 3/8 (.375) 2FL EM
G90 G0 S10000 M3 E1 X2. Y-1.5
H6 D6 Z.1 M8
Z.01 G1 F10.
G8
G91
X-1. Z-.135 F30.
X1.
X-1. Z-.145
X1.
X.19 G41 F50.
X-.19 Y.19 I-.19 G3
X-1.
Y-.38 J-.19 G3
X1.
Y.38 J.19 G3
X-1.
Y-.38 J-.19 G3
X1.
X.19 Y.19 J.19 G3
X-.19 G40
G90
Z.1 G0
G0 G90 H0 Z0

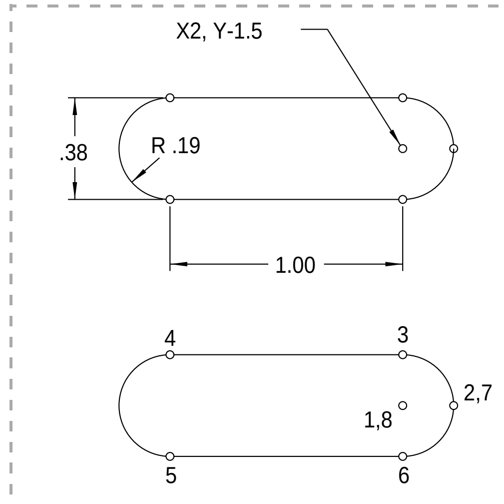


Figure 13-15: Contoured Slot Example

13.4.3 CIRCULAR INTERPOLATION USING G18 & G19

Using **G18** (XZ plane) and **G19**(YZ plane) requires the use of K to represent the direction and distance in Z from the starting point of the circle to the circle center.

13.4.4 G18

Circles in the XZ plane require either the absolute or incremental end point, the I and K circle center description, and a G2/G3 arc direction. Note that G2 and G3 are reversed. The information in this section concerning the use of I and J applies to the K word.

The G18 word must appear in the program just before the circular move. The G18 is required only for the XZ circular moves. Linear XZ moves can be programmed from any plane. When cutter radius compensation is required for linear and circular XZ moves, use the G18 just before applying compensation and the compensation will be applied to the XZ axes.

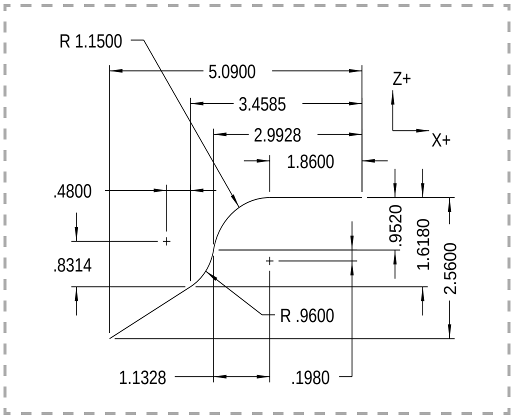


Figure 13-16:

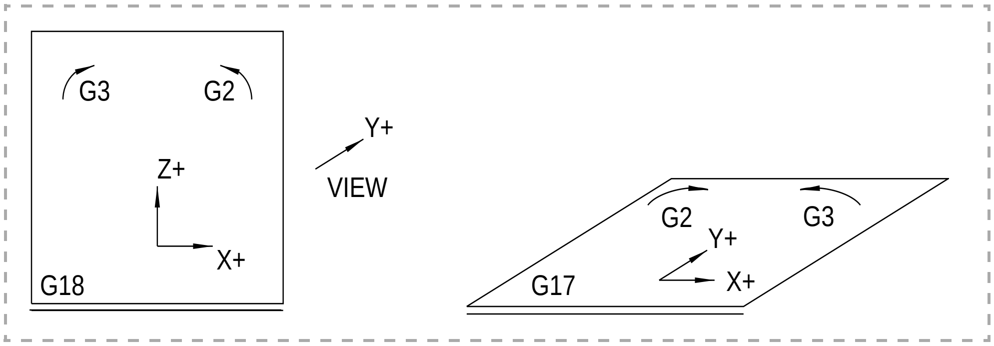


Figure 13-17:

EXAMPLE: G90

```
G90 X-5.09 Z-2. G0
G18
Z-2.56 G42 G1
X-3.4585Z-1.168
X-2.9928 Z-.952 I-.48 K.8314 G2
X-1.86 Z0 I1.1328 K-.198 G3
X0
Z.5 G0 G40
```

EXAMPLE: G91

```
G90 X-5.09 Z-2.
G18
G91
Z-.56 G41 G1
X1.6315 Z.942
X.4657 Z.666 I-.48 K.8314 G2
X1.1328 Z.952 I1.1328 K-.198 G3
X1.86
Z.5 G0 G40
```

13.4.5 G19

Circles in the YZ plane require either the absolute or incremental end point, the J and K circle center description, and the G2 or G3 description. The information in this section concerning the use of I and J applies to the K word.

The G19 word must appear in the program just before the circular move. The G19 is required only for the YZ circular moves. Linear YZ moves can be programmed from any plane. When cutter radius compensation is required for linear and circular YZ moves, use the G19 just before applying compensation and the compensation will be applied to the YZ axes.

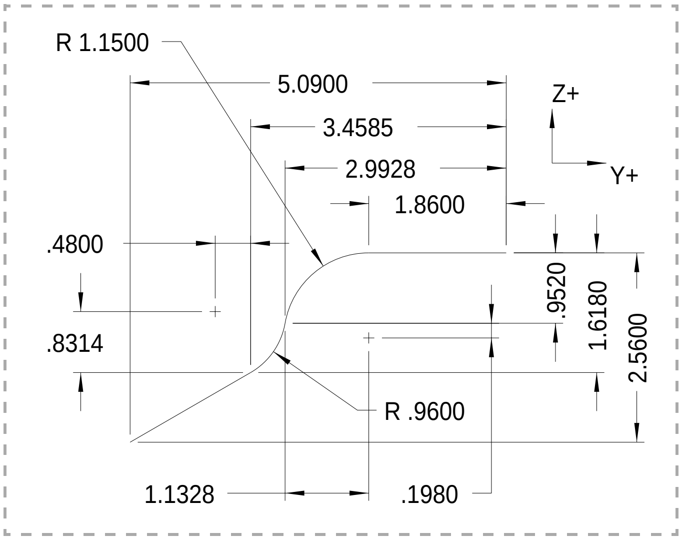


Figure 13-18:

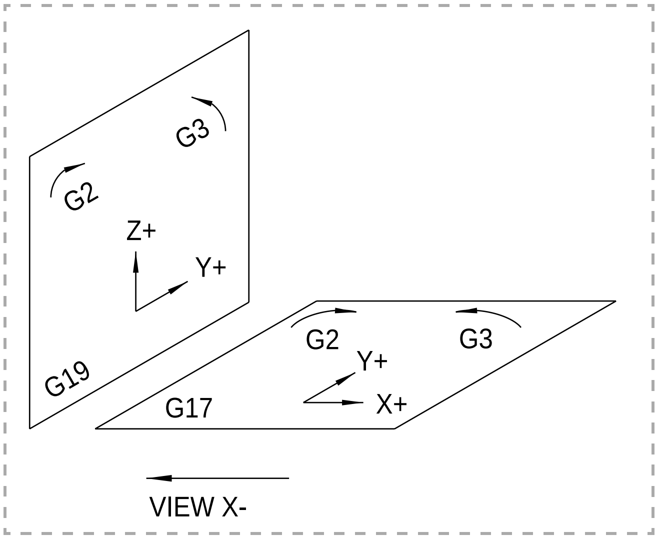


Figure 13-19:

EXAMPLE: G90

G90 Y-5.09 Z-2.
G19
Z-2.56 G41 G1
Y-3.4585 Z-1.618
Y-2.9928 Z-.952 J-.48 K.8314 G3
Y-1.86 Z0 J1.1328K-.198 G2
Y0
Z.5 G0 G40

EXAMPLE: G91

G90 X-5.09 Z-2.
G19
G91
Z-.56 G41 G1
X1.6315 Z.942
X.4657 Z.666 J-.48 K.8314 G3
X1.1328 Z.952 J1.1328 K-.198 G2
X1.86
Z.5 G0 G40

13.5 CIRCULAR INTERPOLATION USING RADIUS DESIGNATION

An R word, which represents the radius of the circle, can be used in place of the I, J, or K arc center description. This can be used in any plane (G17, G18, G19) and with absolute or incremental in effect.

An R0 variable is used to represent the radius of the arc to be programmed. The sign (+,-) of the R0 value determines the amount of angular motion. A negative value is used for motion greater than 180 degrees. A positive value for motion equal to or less than 180 degrees. For example; R0+1. defines a one inch radius, with motion equal to or less than 180 degrees; R0-1. defines a one inch radius, with the motion greater than 180 degrees.

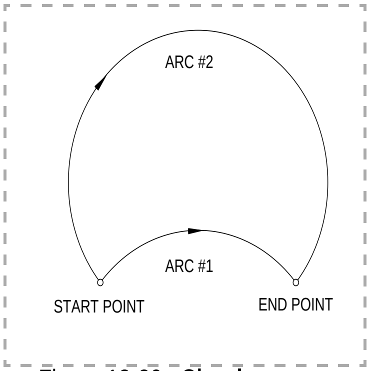


Figure 13-20: **Circular**

The drawing above shows two possible clockwise (G2) paths. Each has the same end point. One arc is greater than 180 degrees, the other less than 180 degrees. Use R0+1.0 to move along arc #1 or use R0-1.0 to move along arc #2.

13.5.1 G90

X3.623 Y-2.173 R0+.75 G2 (for arc #1)
X3.623 Y-2.173 R0-.75 G2 (for arc #2)

13.5.2 G91

X.65 Y0 R0+.75 G2 (for arc #1)
X.65 Y0 R0-.75 G2 (for arc #2)

> NOTE
> When using radius designation, the circular motion must be less than 360 degrees. Full circles are not allowed.

The radius value can be represented by a parameter:

X1. Y1. R0+R1 G3

The R0 value will be the R1 parameter value. (For more information concerning the use of parameters see section 18.0 MACROS,MAN-0131.)

13.6 HELICAL INTERPOLATION

Helical moves are used in various situations. Thread milling is one good example. Helical moves can be used for roughing the walls of a bore and for entering a pocket with a bottom cutting end mill. This type of motion can be used to replace a pilot hole drilled to enter a bore or a pocket.

When G17 (XY plane) is coded, the perpendicular axis is the Z axis. A helical move in G17 would be an XY circular move with a Z axis move coded in the same line as the circle. For G18 (XZ plane), the perpendicular axis is the Y axis. A helical move in G18 would be an XZ circular move with a Y axis move in the same line. For G19 (YZ plane), the perpendicular axis is the X axis. A helical move in G19 would be a YZ circular move with an X axis move in the same line. For any of the planes selected with G17, G18, or G19, the perpendicular axis move finishes at the same time that the circular move finishes.

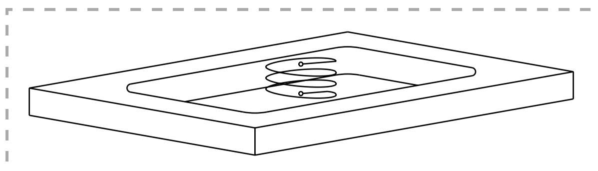


Figure 13-21:

EXAMPLE:

(TOOL #1, 3/8 2FL EM, CENTER CUTTING)
M6 T1
G90 G0 S8000 M3 E1 X2. Y-1.5 This is the move to the center of the pocket
H1 D1 Z.05 M8
X2.3 Z0 G1 F30. This is the move to the start point of the circle and to the top
G91
I-.3 G3 Z-.05 This is the first helical move
I-.3 G3 Z-.05
I-.3 G3 Z-.04 At the end of this move the tool is .14 deep
X-.3 F40. This move is back to the center of the pocket
L9601 R0+40. R1+.01 R2+2.98 R3+1.98 Rectangular pocket routine
X.3 This move is to the start of the circle
I-.3 G3 Z-.010 This last helical move is to the -.15 finished floor depth
X-.3 F50. This move is back to center
L9601 R0+50. R1+.01 R2+3. R3+2. Rectangular pocket routine
M5 M9
G90 G0 H0 Z0

13.7 REPEATING HELICAL MOVES

Helical moves are usually made by repeating subroutines, subprograms, or copying lines in the program.

13.7.1 EXAMPLES USING COPIED LINES

G90
X5.5 G1 G41
I-.5 G3 Z-.1
I-.5 G3 Z-.2
I-.5 G3 Z-.3
X5. G40

> NOTE
> The circular moves remain the same, however the absolute Z axis positions must change in each line.

G91
X.5 G1 G41
I-.5 G3 Z-.1
I-.5 G3 Z-.1
I-.5 G3 Z-.1
X-.5 G40
or
G91 X.5 G1 G41
I-.5 G3 Z-.1 L3
X-.5 G40

> NOTE
> The circular and Z axis moves remain the same in each line. The copy command can be used to copy each incremental helical move.

13.7.2 EXAMPLES USING LINE REPETITIONS

G90
X5.5 G1 G41
G91 I-.5 Z-.1 G90 L3
X5. G40

Note that the L3 is used here to repeat the helical move. This example can only be used in a main program. The L line repeat code is restricted to the main program. The G91 in the beginning of the line establishes the incremental mode for the helical move. The G90 in the end of the line establishes the absolute mode for the next line. This will work if it is typed into the control in this exact order.

13.7.3 EXAMPLES USING SUBROUTINES AND SUBPROGRAMS

EXAMPLE: USING SUBROUTINES

L100 (SUB FOR HELICAL MOVES DOWN INTO POCKET
G91 I-.5 G3 Z-.1 CRC is not applied here
M17
L200 (SUB FOR POCKET
G91 D1 The diameter of the tool must be in tool table to function properly
L104 The helical move in subroutine L100 is repeated four times here (Z-.4)
L9601 R0+10. R1+.015 R2+2.96 R3+2.96 Roughing the pocket
X1.5 G41 G1 Finishing the pocket, apply CRC here
Y.98
X-.52 Y.52 I-.52 G3
X-1.96
X-.52 Y-.52 J-.52 G3
Y-1.96
X.52 Y-.52 I.52 G3
X1.96
X.52 Y.52 J.52 G3
Y.98
X-1.5 G40 Cancel CRC
G90 Z.1 G0   Z up to move to the next pocket location
M17
M30
*********MAIN PROGRAM **********
M6 T1
G0 G90 S200 M3 E1 X0 Y0
H1 Z.1 M7
G1 Z.05 F10.
L201 Call subroutine 200 1 time
Move to next fixture and repeat

G0 G90 H0 Z0
E0 X0 Y0
M2

Note that the helical move was placed in a subroutine. The first line of sub L200 has the G91 code needed for L100 to be repeated to get the incremental Z move down.

EXAMPLE: USING SUBPROGRAMS

O1 (SUB FOR HELICAL MOVE
G91 I-.5 G3 Z-.1 CRC is not applied here
M99
O2 (SUB FOR POCKET
G91 D1 The diameter of the tool must be in tool table to function properly
M98 P1 L4 The helical move in subprogram O1 is repeated here four times (Z-.4)
L9601 R0+10. R1+.015 R2+2.96 R3+2.96 Roughing the pocket
X1.5 G41 G1 Finishing the pocket, apply CRC here
Y.98
X-.52 Y.52 I-.52 G3
X-1.56 G40 Cancel CRC
G90 Z.1 G0 Z up to move to the next pocket location
M99

Note that the helical move was placed in a subprogram. The first line of sub O2 has the G91 code needed for O1 to repeated to get the incremental Z move down.

13.8 CUTTER RADIUS COMPENSATION WITH HELICAL MOVES

Cutter radius compensation can be used with a helical move. CRC must be turned on before the helical move. Compensation is applied to the axes of the selected plane. For example in G17 compensation is applied to the X and Y axes and no compensation would be applied to the Z axis. The same is true for the G18 plane and the G19 plane.

Here is a example of applying compensation and then making the helical move. The helical move is used to rough the bore and a circular move is made at the end to finish the bore. Note that a drill was not needed as a pilot for the end mill. Because the tool is climb cutting down the wall of the bore, the tool is automatically leaving stock for the finish cut at the end.

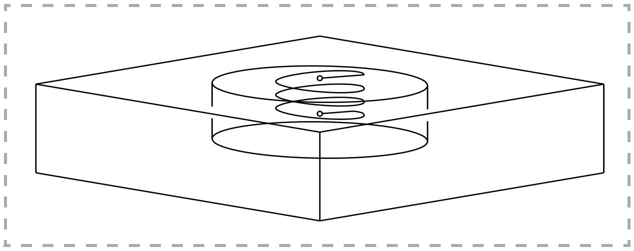


Figure 13-22: Helical Move

EXAMPLE: *(TOOL #1, 3/4 2FL EM, BOTTOM CUTTING*
M6 T1

G90 G0 S5000 M3 E1 X2. Y-2.
H1 D1 Z.05 M8
X2.5 Z0 G1 F25. Tool down to the top of the part
X3. G41 Comp on to the edge of the one two inch bore
G91 I-1. G3 Z-.3 L3 Helical down -.9 from the top
I-1. G3 Z-.13 Helical down -1.03 from the top
I-1. G3 L2 Two finishing passes
G90 X2. G40 G0 Comp off back to the center of the bore
M5 M9
G90 G0 H0 Z0

13.9 PARTIAL ARCS AND HELICAL MOVES

Helical moves can be made with any circular move, including full circles and partial arcs.

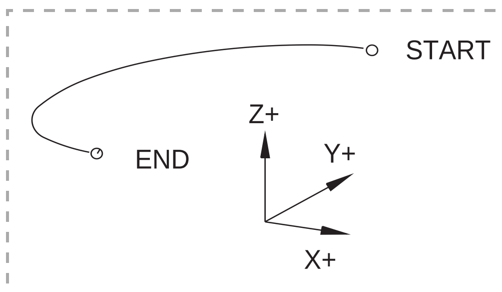


Figure 13-23: Partial Arc

EXAMPLE:

G91
Y.19 G41 G1
Y-.38 J-.19 G3 Z-.05 Helical move with a partial arc
X1. Z-.05 Linear XZ move
Y.38 J.19 Z-.05 Helical move with a partial arc
X-1. Z-.05 Linear XZ move
Y-.38 J-.19 G3 Finish moves for slot
X1.
Y.38 J.19 G3
X-1.
Y-.19 G40

13.10 THREAD MILLING

Threads can be cut using a single point thread mill or with a thread hob. A single point thread mill can be used for any lead thread, where a thread hob is ground for a specific lead. Thread hobs and single point cutters are available as carbide inserts. Inserts can also be purchased for pipe threads.

1. A boss must be cut to the major diameter of the thread for an outside diameter (OD) thread, and a bore must be cut to the minor diameter of the thread for an inside diameter (ID) thread.

2. Determine the lead of the thread to be cut. The lead represents the distance from thread to thread.

    a. Lead (inches) = 1 / threads per inch
    b. Lead (metric) =(m20X2.0) the lead is the 2.0, or 2.0mm lead. To convert to inches multiply the lead times .03937 to get the lead distance in inches.

EXAMPLE: 1 1/16 - 28 thread

Lead = 1 / 28   Lead = .0357

13.10.1 CUTTING AN OD THREAD

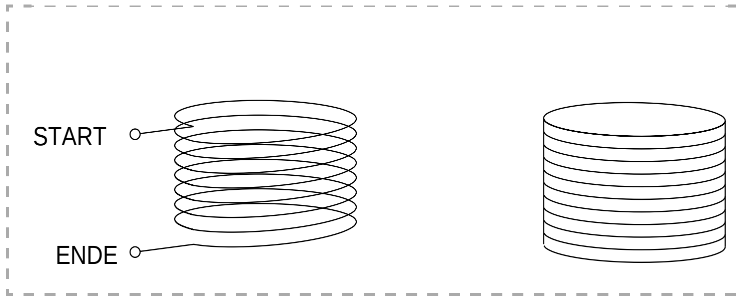


Figure 13-24: Cutting Odd Threads

To cut a right handed OD thread:

1. Start at the top of the boss for an OD thread.
   a. Turn comp on, moving up to the minor diameter.
   b. Helical cut down using G2 and a Z- (lead) for each thread.
   c. Move away from the minor diameter turning comp off.

The tools are set at the top of the part. The thread is located at X0 Y0. The amount of thread to cut is .500

(OD Thread example 1 1/16 - 28 3A)

Major dia. - 1.0625/1.056

Minor dia. - 1.0187

N46 (TOOL #5 3/4 DIA SINGLE POINT CARB THREAD MILL (CRC)
N47 M6 T5
N48 G90 G0 S10000 M3 X-1. Y0
N49 H5 Z.0357 M7
N50 X-.5094 G41 G1 F25.
N51 G91
N52 I.5094 G2 Z-.0357 L15
N53 G90
N54 X-1. G40 G0
N55 M5 M9

Line N48 moves the tool to a safe position to bring the tool down away from the boss.

Line N49 brings the tip of the threading tool to one thread above the boss.

Line N50 turns comp on while moving up to the minor diameter.

Line N52 is the helical cut down, in incremental mode. The L15 in line 52 repeats the line fifteen times. This is why it is in incremental mode. Each repeat represents the next thread.

Line N54 moves the tool away from the diameter turning comp off.

13.10.2 CUTTING AN ID THREAD

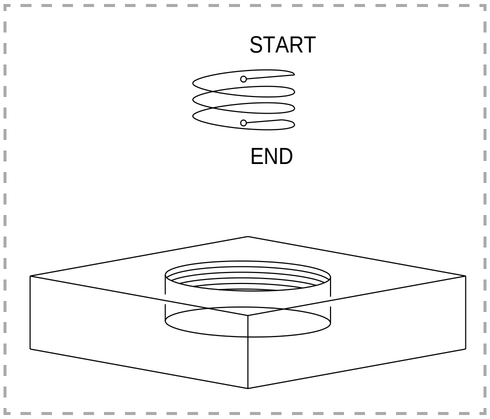


Figure 13-25: Cutting ID Threads

To cut a right handed ID thread:

1. Start at the bottom of the bore.
   a. Turn comp on moving up to the major diameter.
   b. Helical cut up using G3 and a Z+(lead) for each thread.
   c. Move away from the diameter turning comp off.

The tools are set at the top of the part. The thread is located at X0 Y0. The amount of thread to cut is .500

(ID Thread example 1 1/16 - 28 2B)

Major dia. - 1.0625

Minor dia. - 1.0240/1.0301

```
N46 ( TOOL #5  3/4 DIA SINGLE POINT CARB THREAD MILL (CRC)
N47 M6 T5
N48 G90 G0 S10000 M3 X0 Y0
N49 H5 Z.1 M7
N50 Z-.5 G1 F50.
N51 X.5313 G41 G1 F25.
N52 G91
N53 I-.5313 G3 Z.0357 L15
N54 G90
N55 X0 G40 G0
```

Line 48 moves the tool to the center of the hole.

Line 50 brings the tip of the threading tool to the depth of the first thread. Line 51 turns comp on while moving up to the major diameter.

Line 53 is the helical cut up, in incremental mode. The L15 in line 53 repeats the line fifteen times. This is why it is in incremental mode. Each repeat represents the next thread.

Line 55 moves the tool away from the diameter turning comp off.

14.0 COMMUNICATIONS

14.1 COMMUNICATIONS

14.1.1 CHARACTER CODE

The CNC 88 can send or receive either EIA or ASCII character codes. The EIA code output is intended for output to a tape punch only. The communications described in this section are intended for the ASCII mode.

14.1.2 INITIAL CONNECTION

When the CNC 88 is in the command mode (ENTER NEXT COMMAND is on the screen), it constantly monitors the RS-232 port for incoming data. When data is detected, the CNC 88 switches to the terminal mode and subsequent input/output is through the RS-232 port. Also, the user can direct the CNC 88 to send its output to the RS-232 port by entering the proper CD command.

The initial communication to the machine requires the following communications parameters:

7 Data Bits

Even Parity

1 Stop Bit

Compatible Baud Rate (i.e. 2400 at both the computer and the machine).

14.1.3 MACHINE GROUNDING

The machine grounding may affect the communications capabilities. The machine MUST be properly grounded for best communication results. The FADAL VMC shall be grounded using any method listed in article 250 of the National Electrical Code. The minimum grounding method recommended by FADAL Engineering is detailed in the Installation chapter of the VMC Maintenance Manual. Inadequate grounding will result in a wide range of hard-to-diagnose problems in communications, positioning, spindle motion, etc.

14.1.4 TERMINATION

If the MANUAL key is pressed while the control is in the terminal mode, communication will be terminated. This is also accomplished at the computer terminal by typing the word BYE.

14.1.5 HANDSHAKING

Handshaking uses one of two available software protocols:

14.1.6 XON/XOFF PROTOCOL

DC1-READER ON (ASCII 17) DC2-PUNCH ON (ASCII 18)
DC3-READER OFF (ASCII 19) DC4-PUNCH OFF (ASCII 20)

While punching (or outputting) a program, the CNC receives either a DC3 or DC4, and the transmission pauses. When a DC1 or DC2 is received, transmission is resumed.

14.1.7 XMODEM PROTOCOL

132-byte packet with 128 bytes of data and 4 control bytes, including a 1-byte checksum.

14.1.8 HOW TO SEND A FILE USING XMODEM FOR DNCX

The following listing provides an outline for implementing XMODEM protocol. Data is sent 128 bytes at a time. Program lines are concatenated with other lines until there are 128 bytes to send (including carriage returns). Four control bytes are added to the 128 bytes to complete a packet. The first three bytes are 1) CHR$(1); 2) Block Number (from 0-255, starting at 1); 3) 255 - Block Number. The 128 data bytes are next, followed by a checksum byte.

The procedure for sending data is based on ACKs and NAKs. ACK means acknowledged, or success. NAK means not acknowledged, or failure. An EOT is sent if an unrecoverable error or end-of-file has occurred. The logic is better described in sample code than in words.

EXAMPLE:

```
SUB DNC ()
          ' SETUP VALUES
          ' TRANSMIT BLOCK - [3-128-1], 1ST IS CONTROL BYTE
          SENDBLOCK$ = CHR$(1) + STRING$(131, 32)
          ' BLOCKNUMBER WILL BE INCREMENTED TO 1 IN FIXBLOCK
          BLOCKNUMBER% = 0
          ' TIMES BLOCK WAS BEEN TRANSMITTED, 3 FOR ERROR
          XMITEDCOUNTER% = 0
          RECDNAK% = 0
          RECDACK% = 0
          SENDFLAG% = 0
          EOT$ = CHR$(4)
          NAK$ = CHR$(21)
          ACK$ = CHR$(6)
          STARTCHAR% = 4
          CALL CLEARPORT
NEXTBLOCK:
          CALL NEXTDNC(DATA$)
          IF DATA$ = "" THEN GOTO SPACEFILL
BACKTOSLOW:
          DATA$ = DATA$ + CHR$(13)
          DATALENGTH% = LEN(DATA$)
          SELECT CASE STARTCHAR% + DATALENGTH%
                    CASE IS > 132
                              CANFIT% = 132 - STARTCHAR%
                              LEFTOVER% = DATALENGTH% - CANFIT%
                              MID$(SENDBLOCK$, STARTCHAR%, CANFIT%) = LEFT$(DATA$, CANFIT%)
                              GOSUB BEGINSENDING
                              MID$(SENDBLOCK$, 4, LEFTOVER%) = RIGHT$(DATA$, LEFTOVER%)
                              STARTCHAR% = 4 + LEFTOVER%
                    CASE 132
                              MID$(SENDBLOCK$, STARTCHAR%, DATALENGTH%) = DATA$
                              GOSUB BEGINSENDING
                              STARTCHAR% = 4
                    CASE ELSE
                              MID$(SENDBLOCK$, STARTCHAR%, DATALENGTH%) = DATA$
                              STARTCHAR% = STARTCHAR% + DATALENGTH%
          END SELECT
          GOTO NEXTBLOCK
SPACEFILL:
          ' SPACE FILL REST OF BLOCK, TERMINATING CR
          MID$(SENDBLOCK$, STARTCHAR%, 131 - STARTCHAR% + 1) =
          STRING$(131 - STARTCHAR%, CHR$(32)) + CHR$(13)
          GOSUB BEGINSENDING
FINISHUP:
          CALL DWELLSENDING
          CALL CLEARPORT
QUITING:
          CLOSE
EXIT SUB

BEGINSENDING:
          CALL FIXBLOCK
BEGINSENDINGSKIP:
          CALL CHECKFORABORT
          ' WAS LAST BLOCK ACKED OR NAKED?
          IF RECDACK% OR RECDNAK% THEN RECDACK% = 0: GOTO ZXREADYTOSEND
ZXREADYTORECD:
          CALL WAITTILGOT(STOP$)
          SELECT CASE STOP$
```

EXAMPLE:

```
CASE IS = EOT$
                        GOTO QUITING
                CASE IS = NAK$
                        RECDACK% = 0
                        RECDNAK% = 1
                        IF SENDFLAG% THEN
                                ' BLOCK WAS SENT BUT NOT RECEIVED, RE-TRANSMIT UP TO 3 TIMES
                                SELECT CASE XMITEDCOUNTER%
                                        CASE IS = 0, 1
                                                XMITEDCOUNTER% = XMITEDCOUNTER% + 1
                                        CASE ELSE
                                                CALL MHSENDDATA(PORT%, EOT$, ECODE%)
                                                IF ECODE% THEN ERROR ERRTRANSMIT%
                                                GOTO FINISHUP
                                END SELECT
                        END IF
                        GOTO ZXREADYTOSEND
                CASE IS = ACK$
                        IF SENDFLAG% THEN
                                ' BLOCK WAS SENT AND RECEIVED
                                SENDFLAG% = 0
                                XMITEDCOUNTER% = 0
                                RECDNAK% = 0
                                RECDACK% = 1
                                RETURN
                        END IF
        END SELECT
ZXREADYTOSEND:
        CALL DWELLSENDING
        CALL MHSENDDATA(PORT%, SENDBLOCK$, ECODE%)
        IF ECODE% THEN ERROR ERRTRANSMIT%
        SENDFLAG% = 1
        ' WAIT FOR ACK
        GOTO ZXREADYTORECD
END SUB

SUB FIXBLOCK
        ' SET BYTES 2 AND 3 FOR HEADER, 132 HOLDS CHECKSUM
        BLOCKNUMBER% = BLOCKNUMBER% + 1
        IF BLOCKNUMBER% > 255 THEN BLOCKNUMBER% = 0
        MID$(SENDBLOCK$, 2) = CHR$(BLOCKNUMBER%)
        MID$(SENDBLOCK$, 3) = CHR$(255 - BLOCKNUMBER%)
        CSUM% = 0
        FOR I = 1 TO LEN(SENDBLOCK$) - 1
                CSUM% = CSUM% + ASC(MID$(SENDBLOCK$, I))
        NEXT I
        MID$(SENDBLOCK$, 132) = CHR$(CSUM% MOD 256)
END SUB

NEXTDNC:
        ' RETURNS NEXT ASCII LINE TO SEND
        DATA$ = "" IF END OF FILE
DWELLSENDING:
        ' WAIT UNTIL TRANSMISSION BUFFER IS EMPTY
```

14.2 COMMANDS

14.2.1 CNC 88 COMMANDS

All of the CNC 88 commands may be issued from the computer terminal. The MANUAL key is simulated by a control/E (05 hex or ENQ character). The line delete function is accomplished by the @ key and the backspace function is accomplished by the underline key. The TA and PU commands have special requirements and features discussed below.

14.2.2 TAPE INPUT

The following is a description of the two possibilities to initiate data input.

a. From the keyboard, enter the commands: CD, (Baud Rate #) and TA,1
b. While in the command mode if a DC2 is received from the RS-232 port, a TA,1 is assumed by the CNC.

The following events are required to successfully complete the transfer of an NC program.

1. The CNC 88 starts by issuing a DC1 control character. Terminals such as the teletype with the AUTO mode respond to DC1 by turning on the tape reader. If the sending device has a tape reader that does not respond to DC1, the operator must start the reader after the TA,1 command has been issued. The TA,1 command will send a DC3 to activate the tape reader.
2. The next data sent must be a “%” character followed by a CR control character, to signal the start of NC program data.
3. The terminal must now send the body of the NC program. Lines of the program must be separated by a CR control character. An LF character is optional. After all lines are sent, one or more TO commands followed by one or more FO commands may be sent to build the tool and fixture data tables.
4. The terminal must conclude the NC program data by sending a “%” character followed by a CR control character.
5. An optional checksum may follow. The checksum is sent as a string of characters consisting of one to four numeric digits(0-9) and terminated by a carriage return. The checksum may be generated by a high level language like BASIC. An example follows:

```
10 REM update checksum after sending character C$

20 CK + CK = ASC (C$)

30 If CK9999 Then CK=CK-9999
```

NOTE

a. CK is set to zero before the “%” Character is sent in step 20 above.
b. CK includes all characters sent up to, but excluding, the first character of the checksum.
c. The only control character included in the checksum is the CR (i.e. the LF character is not counted).
d. If a null line (only a CR) is sent, the CR must not be included in the checksum.

6. At the completion of a successful TAPE input, when no CHECKSUM is received, the CNC displays the message “TAPE INPUT TERMINATED”. When a CHECKSUM is correct, the message “TAPE IS GOOD” is displayed.

14.2.3 PU COMMAND

The following is a description of two ways to initiate data output:

a. From the keyboard, type CD, (baud rate #) and press ENTER then type PU and press ENTER.
b. While in the command mode if a DC1 is received from the RS-232 port, a PU is assumed by the CNC.

The PU command has two modes of transmission. The first assumes that the remote device is a “dumb” terminal such as a punch or a teletype. This mode is initiated by the PU command (with no parameters). The CNC 88 transmits the NC program as follows:

1. A DC2 control character is sent. Some terminals respond to this by turning on the tape punch.
2. Approximately 300 NULL characters (leader) are sent next (when punch tape format is selected). Leader is transmitted only at baud rates 600, 300, 150, and 110.
3. The data portion is then opened with a “%” character followed by LF and CR characters.
4. The body of the NC program is transmitted next with LF and CR characters terminating each line.
5. After the NC program, TO and FO commands are output (if selected) to download the tool and fixture data tables.
6. The data is finally closed with a “%” character followed by LF and CR characters.
7. The last line sent is the checksum as described in the TA command.

The second mode assumes that the remote device is programmable and is prepared to receive the NC data in the order described below. This mode is invoked by the PU,,,1 command. The order is as follows:

1. The CNC 88 sends one null character and waits for a DC1 or DC2 from the remote device.
2. The remote device then sends a DC1 or DC2 when it is ready to receive data.
3. The CNC 88 responds with ten lines of data.
4. The remote device must process the data received in step 3.
5. Steps 2 through 4 must be repeated until the end of file is detected. The end of file is indicated by a line consisting of a single “%” character. All lines are terminated by LF only.
6. After the end of file, the CNC 88 sends the checksum as described above.

14.2.4 DNC MODE

DNC mode is generally used when a program is too big for the memory of the CNC control. CAD/CAM software usually posts a program too big to fit into the CNC memory, either the program is sectioned and stored, or it is sent DNC.

DNC mode does not store a program in the control. The control receives program code via the RS-232 port in blocks. Enough program code is sent to fill a buffer for the machine tool to perform. Once the machine has performed the code, it looks to the buffer for more information. When the buffer starts to empty, the control talks to the units sending the code to refill or keep filling the buffer so that the machine can have a continuous run. Since the DNC program does not remain in the control, certain differences to the code are as follows:

a. NO O word program identification numbers are allowed.
b. NO comments are allowed.
c. NO user defined subroutines (L01-L89) are allowed.
d. Fixed subroutines are allowed during DNC.
e. Sequence numbers are optional.
f. The program should be ended with either an M0 or with NO code at all.
g. Macro statements (see section 18.0 MACROS, MAN-0131) ARE allowed.

Using FADAL communications software such as Assist, ProSend, or CNC Memory Manager in the DNC mode, a server program can be accessed by encoding a drive, path and filename within the program. During DNC the program lines indicating the path and filenames may NOT have line sequence numbers.

EXAMPLE:

```
N1 G0 G90 S3000 M3 E1 X0 Y0
N2 H1 Z.1 M7
C:\ASSIST\CNCDATA\MOLD.CNC
N3 G0 G90 H0 Z0
N4 M0
```

> **NOTE**
> A % sign is not required in sub program

14.2.5 DNC PROTOCOLS

The control may use either one of two types of handshaking protocols.

1. **DNC**, the standard protocol is XON/XOFF.

   a. This protocol uses DC1 through DC4 to control data flow.
   b. When the machine cannot continue to receive data, or the buffer is full, the XOFF stop signal is sent. When transmission can resume, the XON start signal is sent.
   c. This protocol is used for normal DNC operations.

2. **DNCX**, Xmodem protocol is used at a baud rate setting of 9600 baud or **above**.

   a. X modem protocol sends data in blocks of 128 bytes.
   b. After sending each block a CHECKSUM is performed. These blocks are used to report errors that may occur during transmission.
   c. X modem protocol requires the use of the DNCX command during DNC operations.
   d. DNCX must be used with computer software or a device that can send a file at X modem protocol.
   e. DNCX command is available with the 1400-3 processor only.

14.2.6 RS-232-C INTERFACE CONNECTION

Data and command input to the CNC is normally via the keyboard on the VMC pendant. However, the CNC can be instructed to receive its input from the RS-232-C interface. Likewise, data normally sent to the CRT can also be directed to the interface. This interface is set up in the data terminal equipment (DTE) mode per the Electronic Industries Association (EIA) Standard numbered RS-232-C published by:

Electronic Industries Association
Engineering Department
2001 Eye St. N.W.
Washington, D.C. 20006

Table 1 shows the circuits that are implemented in the CNC's RS-232-C interface. In this table, DCE, stands for Data Communication Equipment. The DCE is normally a telephone modem. If the user wishes to connect a computer or a data terminal directly to the CNC, it must be determined whether the device is set up as DTE or DCE.

If the device is DCE, a commercially available 9 wire cable will be satisfactory. The CNC requires the socket housing (female) connector at its end of the cable. If the device is DTE, the user must make or buy a modem bypass cable (also called a null modem).

Two versions of a null modem are shown in tables 2 and 3. Table 2 is for simple 3 wire cabling. Table 3 is for equipment that uses the DTR circuit to indicate conditions, such as buffer full. This condition requires a brief pause in transmission from the CNC (see handshaking). If the CNC finds its DSR line low for more than 1 second it ceases transmission until it is reconnected.

The CNC RS-232-C interface has a programmable baud rate. It can be changed by use of the CD command (see section 8.0 COMMANDS, MAN-0131). When the CNC is powered on, the initial baud rate is set by the SETP command. The CNC is set up for Asynchronous Communication only.

The CNC control expects and transmits seven (7) data bits and one (1) stop bit. For ASCII data, even parity (E) is required, and for EIA-RS-244-B data, odd parity (O) is required.

FADAL's Parts Department has RS232C cables and NULL modem cables available, properly configured for communication with the VMC.

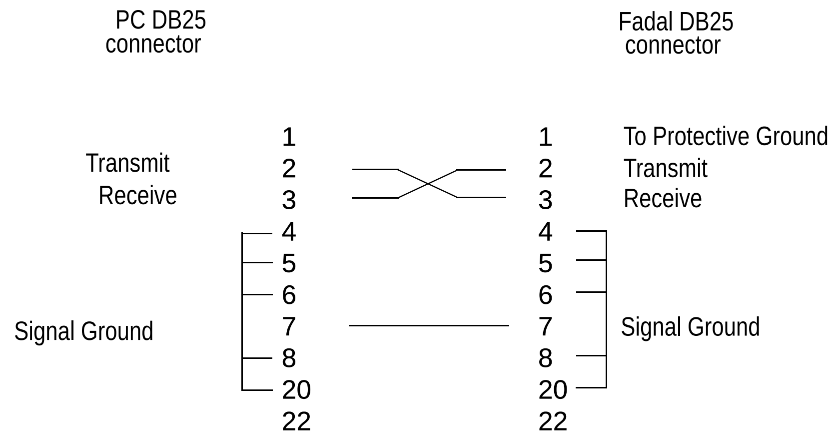
14.2.7 CABLE CONFIGURATION

Table 14-1: Cable Configuration

PIN NUMBER	DESCRIPTION	SIGNAL SOURCE	
25 PIN DIN CONNECTOR		DTE (VMC)	DCE (MODEM)
1	Protective Ground (PG)		
2	Transmitted Data (TD)	X	
3	Received Data (RD)		X
4	Request To Send (RTS)	X	
5	Clear To Send (CTS)		X
6	Date Set Ready (DSR)		X
7	Signal Ground (SG)		
8	Received Line Signal Detector (RLSD)	X	
20	Data Terminal Ready (DTR)	X	

14.2.8 IBM-COMPATIBLE PC-SIMPLE DB25 NULL MODEM

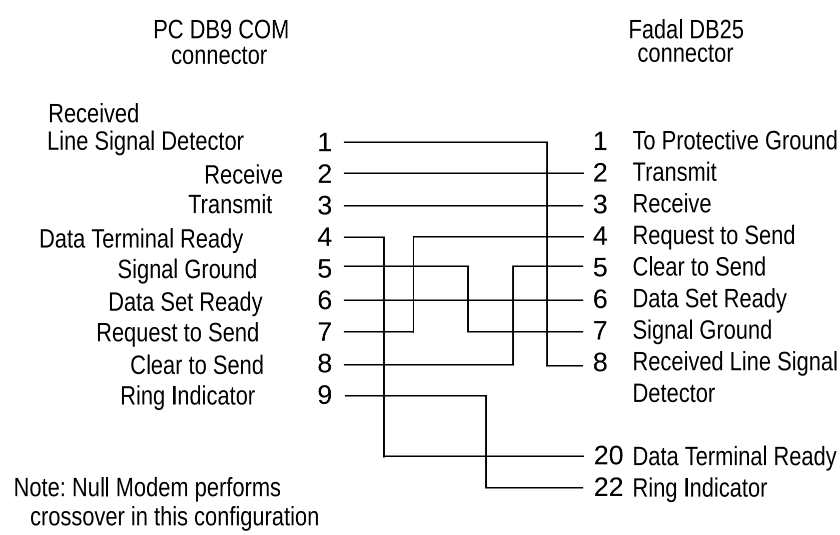
RS232 Pinouts for the IBM-compatible PC
Simple DB25 Null Modem



> **NOTE**
> Connect pin one to the shield **only** at the machine end.

14.2.9 IBM-COMPATIBLE PC-DB9 TO DB25 FOR DTE EQUIPMENT-NULL MODEM NOT INCLUDED

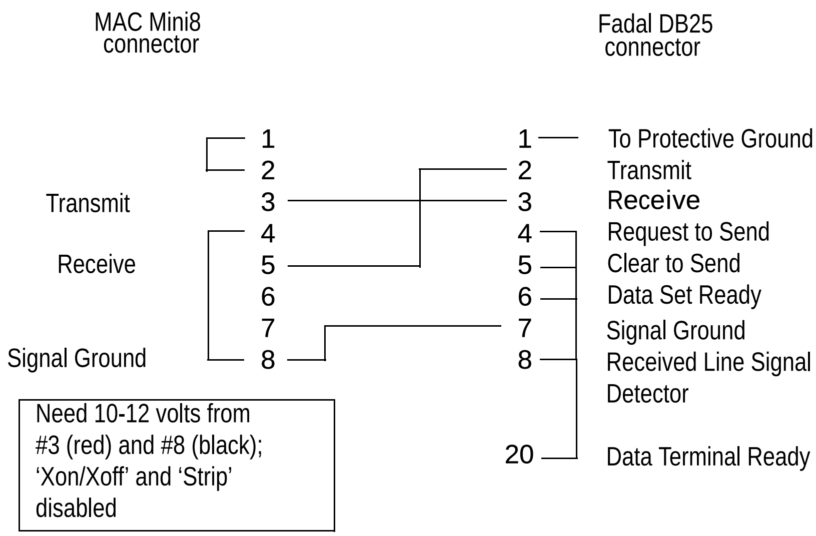
RS232 Pinouts for the IBM-compatible PC
DB9 (@PC) to DB25 (@VMC) for DTE Equipment



> **NOTE**
> Connect pin one to the shield **only** at the machine end.

14.2.10 MACINTOSH II MINI-8 CONNECTOR

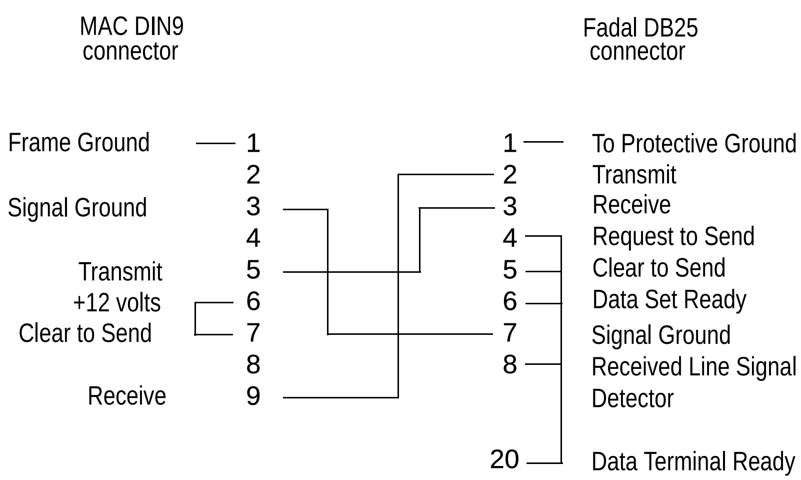
RS232 Pinouts for the Macintosh II



> **NOTE**
> Connect pin one to the shield **only** at the machine end.

14.2.11 MACINTOSH PLUS DIN-9 CONNECTOR

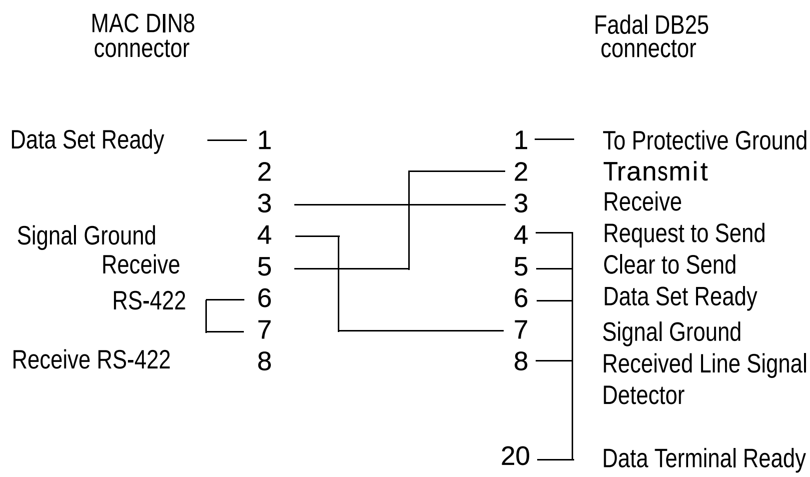
RS232 Pinouts for the Macintosh Plus



> NOTE
> Connect pin one to the shield only at the machine end.

14.2.12 MACINTOSH PLUS DIN-8 CONNECTOR

RS232 Pinouts for the Macintosh Plus



> NOTE
> Connect pin one to the shield **only** at the machine end.

14.3 VMC COMMUNICATIONS PROCEDURES

14.3.1 VMC CONTROL PARAMETERS

The first step to any communications is to prepare the VMC control. At the ENTER NEXT COMMAND prompt, enter the command CD,# and press ENTER. This sets the machine baud rate and readies the RS-232 port. Use the appropriate command for the desired baud rate.

CD,1 = 110 baud
CD,2 = 150 baud
CD,3 = 300 baud
CD,4 = 600 baud
CD,5 = 1200 baud
CD,6 = 2400 baud
CD,7 = 4800 baud
CD,8 = 9600 baud

CD,9 = 19200 baud
CD,10 = 38400 baud
CD,11 = 57600 baud
CD,12 = 115200 baud

14.3.2 RECEIVING DATA / PROGRAMS AT THE VMC

EXAMPLE: TO RECEIVE DATA:

CD,6 ENTER This sets the baud rate at 2400 baud

The line feed and echo parameters may also be used with the CD command. (For detailed information on these parameters see section 8.0 COMMANDS, MAN-0131.)

When a file is to be received at the VMC the TA,1 command is used. This command is entered after the CD command.

EXAMPLE: The program sent to the control should start and conclude with the “%” sign.

CD,6 ENTER
TA,1 ENTER The VMC is now ready to receive a program

After entering the TA command the control is ready to receive a file. The program can now be sent to the control from the source device.

14.3.3 SENDING DATA / PROGRAMS FROM THE VMC

When a program is to be transmitted from the VMC to a receiving device, the PU command is used. Prior to using the PU command the receiving device MUST be ready to receive. Check the device requirements for receiving a file. When the device is ready to receive, enter the CD and PU commands at the VMC.

EXAMPLE: TO SEND DATA:

CD,6 ENTER
PU,2 ENTER This transmits the program data only to the receiving device

The use of the PU command parameters may be required for different devices.

Data Options
0 = Currently active program and offsets
1 = Offsets only

2 = Currently active program only
3 = All programs in memory

Code Option
0 = ASCII (Standard)
1 = EIA

TTY Option
0 = Computer (Standard)
1 = Leader, Program, Trailer (This option is used for tape punches)

After entering the PU command the program is transmitted to the receiving device.

14.4 DEVICE COMMUNICATIONS PROCEDURES

14.4.1 COMPUTER

The procedures described in this section are for generic computer communications.

The computer MUST be equipped with a serial port (RS-232 port). If a mouse or pointing device is installed, insure that there is a serial port available for communications. Identify whether the port is COM1 or COM2. This information is required for the software being used.

The RS-232 cable MUST be configured for communications. The standard 25-pin configuration in Table 2 above should be used. This pin configuration may be achieved by using the FADAL Null Modem cable. Connect the cable(s) from the VMC to the computer serial port.

The communications parameters must now be set at the computer software. The software documentation will describe the procedure for setting these parameters. To communicate with the FADAL VMC use the following parameter settings:

1. Set the baud rate compatible with the machine (Normally 2400).
2. Even Parity
3. 7 Data Bits
4. 1 Stop Bit
5. XON/XOFF Handshaking In Effect (On)

Some software may have additional parameters. Use the following settings:

6. Strip Line Feeds (No)
7. Code Format (ASCII or EIA)
8. Strip Carriage Returns (No)
9. Echo Keyboard To Screen (Yes)
10. Delay After End Of Line (0.0 seconds)

The VMC and computer are now ready for communications.

Receiving at the VMC:

1. Enter the appropriate CD command at the VMC control.
2. Enter the TA command desired to receive a program at the VMC.
3. Perform the software procedures to transmit the file to the VMC.
4. After receiving the file, press the MANUAL button to return to the command mode.

Transmitting from the VMC:

1. Enter the appropriate CD command at the VMC control.
2. Prepare the computer to receive a file. See the software documentation for these procedures.
3. Enter the desired PU command at the VMC control.
4. After receiving the file, press the Manual button to return to the command mode.

14.5 TAPE PUNCH / TELETYPE

The procedures outlined in this section are designed for the DSI tape punch model LRP300. Procedures for other tape punches or teletypes may vary with model. Contact your device manufacturer for exact procedures.

This tape reader uses the RS-232 cable for data transfer. The cable MUST be straight wired. This means that the wire connected to each pin must connect to the corresponding pin at the opposite end of the cable. The Null Modem is NOT used for the tape reader or teletype. The RS-232 cable should be attached to the DSI tape punch at the bottom 25 pin port.

The DSI tape punch switch settings for communication are defined as follows:

OC/L RS-232	*RS-232*
HIGH/LOW	*HIGH*
FULL/HALF	*FULL*

LINE/LOCAL	*LOCAL*
OFF/REM CTL	*OFF*
OFF/PRINT	*PRINT*
ON/PUNCH	*PUNCH*
RUN	*Press to start*

Prepare the machine for communications by entering the CD command. Use the command CD,3 to set the baud rate at 300 baud. This is the baud rate that the tape punch uses. The machine and the tape punch are now ready for communications.

Receiving at the VMC:

1. Enter the CD,3 command at the VMC control.
2. Enter the TA command desired to receive a program at the VMC.
3. Load the paper tape into the tape reader or teletype.
4. Press the RUN button to begin the transmission.
5. After receiving the file, press the Manual button to return to the command mode.

Transmitting from the VMC:

1. Enter the CD,3 command at the VMC control.
2. Press the RUN button on the tape reader to prepare the reader to receive the file.
3. Enter the desired PU command at the VMC control.
4. After receiving the file, press the MANUAL button to return to the command mode.

14.5.1COMMUNICATIONS TROUBLESHOOTING

For communication problems several factors must be examined. Review the environment, then check the file, the VMC, the cables, the computer, and finally the computer's communications software. Any one of these subsystems can propagate a communication problem.

If, after following these procedures, communications cannot be successfully carried out, please call FADAL's its representative's) Programming Support department for assistance.

14.5.2 ENVIRONMENT

In areas where lightning strikes are common, communications can be interrupted when or after lightning has struck a power line. The lightning may affect the memory of the CNC control, which would then have to be zeroed from the DI mode. It is suggested to call a service person or see the maintenance manual for this zeroing procedure.

> NOTE
> An RS-232 surge suppressor adds protection in this kind of environment.

14.5.3 FILE

1. Has the file ever successfully been transmitted to the VMC before? If the file has been transmitted before, has it recently been edited and has something been introduced into the file? Use an editor or word processor to show hidden characters. If hidden characters are present in the file, remove them.
2. If the file has not been successfully transmitted, use an editor or word processor to show hidden characters. If hidden characters are present in the file, remove them.
   a. The only accepted characters allowed before the initial percent sign are DNC, DNCX, or TA,1. In addition, no O words are allowed in DNC mode.

EXAMPLE: *DNC*
%

   b. A two-second dwell is required between the DNC command and the first percent symbol (%). If the software being used to transmit the program cannot support this dwell time, remove the DNC, DNCX, or TA,1 command from the file and type the command at the VMC before starting file transmission.

3. When a file is posted for a paper tape, or when a paper tape is read and placed on a disk, it may have leader characters before the initial percent sign — they need to be removed.
4. If some of the file can be transmitted, but some of it cannot, check for syntax errors.
   a. Examine the area in the file for syntax errors and repair them. For example, double motion words or missing words are syntax errors.

EXAMPLE: *X1.23 X4.5 Double motion words*
-Y2.3 Incorrect placement of - (minus)
.534 No word at all

> NOTE
> Use the TA,1,1 command for program transmission so that the control will halt transmission and display the line where the syntax error occurs.

5. If a word processor or editor is used to write the program, it must be saved in a text only format. Some processors will add formatting characters to the file which will disallow communications.

14.5.4 VMC

1. Check to see if the grounding wires are properly attached.

   a. The only proper and acceptable primary ground is a single continuous copper wire attached from the ground bus in the junction box of the VMC to the main power box of the building. A green sticker in the junction box further expounds the grounding requirement. Any other methods of grounding, such as grounding to the conduit, or to a ground rod, are not acceptable and will lead to communication problems. See the Maintenance Manual for grounding procedures.
   b. Check to see if the screws attaching the ground wire to the VMC are tight at both ends.
   c. The ground wire of the RS-232 DB25 plug on the inside of the CNC control cabinet must be attached from the RS-232 port to the inside of the CNC control cabinet. This wire is attached to pin #1 of the DB25 plug, and is used to shield the cable at the VMC end ONLY (see Table 2 for proper pin configuration).

> NOTE
> If the paint has not been removed from this area, scrape the paint away and re-attach the wire.

2. Check the cable from the RS-232 port on the inside of the CNC control cabinet to the 1030 board.

   a. Sometimes this cable will work itself out of the plug. Press the plug into the board to confirm a good connection.
   b. Examine the cable for cuts or kinks and replace it if there is evidence of damage.

3. For VMCs with a phone modem, make sure that the DB25 plug on the inside of the CNC control cabinet is NOT plugged into the back of the modem — if it is, it will interrupt normal communications through the RS-232 port.

4. Use the test plug to test the 1030 board. The test plug is stored in the bottom of the CNC control cabinet when the VMC is shipped. It is a DB25 plug with no wires coming from it. On the inside of the plug, pins 2 and 3 are crossed to complete the communications path. Plug the test plug into the RS-232 port on the outside of the back of the CNC control cabinet.

Use the diagnostics mode to complete the test.

   a. Move to the cold start position.
   b. From the command mode type DI then press ENTER.
   c. Type G0 3000 and press ENTER to enter the test menu.
   d. Press 4 to select the RS-232 (1030) test.
   e. Select a baud rate (usually the baud rate used in normal communications).
   f. Observe the screen.

- Numbers next to the word TESTING should be changing constantly. This is good, and the 1030 board has passed the test.
- The numbers at the end of the other sentences should all be zero. Any numbers appearing at the end of these sentences indicates that the 1030 board is faulty (assuming the cable was checked in step (2) above).

> NOTE
> It has been our experience that the 1030 is rarely faulty.

- If the diagnostics test passes, then it is safe to conclude that the VMC is not at fault for a communication problem.

g. Press the MANUAL button to end the communication test.
h. Power off the VMC, wait 10 seconds, then power on again.

5. Has the proper baud rate been selected at the VMC?

   a. The CD,# command is used to select a baud rate. The MU command has a list of each baud rate in the menu, or see section 8.0 COMMANDS, MAN-0131 for the same list.
   b. The SETP command, for the VMC parameters, can be used to select a baud rate as default. The CD,# command will temporarily override the selection in the SETP parameter page.

14.5.5 CABLES

1. Verify that all connections between the communications cable and both the VMC and the computer are firmly seated, including surge suppressors, gender changers, and couplers. Pick up the cables and physically confirm that each connection is properly together.

   - Loose cable connections are one of the most common causes of communication problems.

2. If a switch box is used, determine if the switch is in the proper position.

   a. Examine the cable connections to see if they are in the correct ports.
   b. Turn the switch handle back and forth a few times. Sometimes the contacts are corroded and turning the handle will temporarily correct the problem.

- If turning the handle corrects the problem, it is suggested to clean the contacts or replace the switch box with a new one.

3. If the cable is coiled because the cable is too long, it is suggested to get a shorter cable. The coiling may cause intermittent problems. Also, the shorter the cable, the less chance a parity error will occur.
4. Check to see if the cable is draped over fluorescent lights or wrapped around or connected on the high-voltage line for the power to the VMC or other machines. This can cause RF noise and inductive voltages on the cable, and communications will be interrupted from time to time.
   a. Welding machines and EDM machines close to the VMC or the cable will also cause communication problems. These machines also cause RF noise which interrupts communications.

5. Open the DB25 plugs to see if the solder on the pins has been applied properly. Cold solder joints for these wires will need to be soldered again.
6. Check to see if the wires are connected to the proper pins. See earlier in this section for the proper pinout.
   - Confirm that, with shielded cable, pin one is connected to the shield at the VMC side only. Pin one cannot be connected at the computer side.

7. If straight cable is used a null modem is required. A null modem cable can be purchased from the FADAL Parts Department (part # WIR-0150), or from any computer store. A null modem is a short portion of cable or ribbon cable that has pins 2 and 3 crossed. The crossover and the proper jumpers on each end can be found earlier in this section. If a null modem is used that was supplied from FADAL, the switch must be in the outward position away from the cable.
8. Use the mirror plug test as described in step 4 of the VMC section above to test each section of cable from the VMC to the computer, replacing any section that fails.
9. What is the length of the cable?
   a. The longer the cable the slower the baud rate must be.
   b. The faster baud rates may work for the longer cables, but the chance of losing information increases the faster the baud rate.
   c. Try using a slower baud rate, and if this works better, this may be the best solution. However, this may also indicate an IO port at the computer with low voltage. See the Computer and Computer IO Port section on the next page.

14.5.6 COMPUTER AND COMPUTER IO PORT

1. Is the cable plugged into the proper port in the back of the computer?

   a. Some ports are marked COM or SER — these are the proper ports.
   b. If the ports are unmarked, a COM or serial port will be the male gender port (the port with the exposed pins — the female gender port will have sockets).

> NOTE
> A gender changer plug may be needed if the gender on the cable will not plug into the proper COM or serial port.

2. Is the port active?

   a. Is the cable on the inside of the computer cover attached to the port?
   b. Check the voltage across pins 2 and 7 while transmitting a file (use a file large enough to allow time to check the voltage while the transmission is in progress). With the positive lead on pin 2 and the negative on pin 7, the voltage should range between 10 and 12 volts. Any voltage below 10 volts could result in interrupted communications.

> NOTE
> A FLUKE meter (or equivalent) used to measure voltage must be set to AC to obtain a reading.

3. When the computer was set up, did any interrupts interfere with the communications port?

> NOTE
> A qualified computer setup person will be able to confirm that the port is free of other interrupts.

4. If an IBM-compatible computer has a serial mouse, is the mouse plugged into the proper port?

   a. Usually the serial mouse is used in COM1 or serial port one.
   - Move the mouse to COM1 and the VMC cable into COM2 and try to communicate again.
   - If the mouse is not to be used during the DNC process, remove the mouse software commands from the CONFIG.SYS and/or AUTOEXEC.BAT file(s), and then reboot the computer. (CAUTION: Only a person familiar with altering these files should perform this operation, as these files are necessary for startup of an IBM-compatible computer. Refer to the DOS manual for any questions regarding these files and how to properly modify them.)

5. Some screen saver software can interrupt communication, change the baud rate, or transmit an odd hidden character, when the screen saver starts to display. Remove or disable the screen saver software and try to communicate again.

6. The quality of the IO board must be considered. Multipurpose IO boards are usually not recommended. High quality boards dedicated to only IO functions are recommended. DigiBoard and Quad Tech are examples of companies which produce high quality IO boards. These boards are specific to the operating system used by the computer — one board is used for DOS, another for Windows, and another for micro channel (IBM PS/2). These boards usually test 12 volts across pins 2 and 7 at the computer, for the best quality transmission.

> NOTE
> A FLUKE meter (or equivalent) used to measure voltage must be set to AC to obtain a reading.

7. If a network board is installed in the computer, the computer technician who installed the board must check for conflicting interrupts and IO addresses.

8. The FADAL Assist software has in its utility menu an RS-232 tester. Use the mirror plug in the port and follow the instructions on the screen for test number one.

14.5.7 SOFTWARE

1. Check the communication parameters in the software.

a. TA, DNC, and PU (using Xon/Xoff)
- Baud rate - This is variable and it must match the baud rate set at the VMC with the SETP command or with the CD,# command.
- Parity - E (Even)
- Data Bits - 7 (Seven)
- Stop Bits - 1 (One)
- EOB (End Of Block) - CR,LF (ASCII 13,10)
- Starting and ending character - % (ASCII 37)
- Xon/Xoff (Software handshaking) disabled for TA
- Enabled for DNC and PU
- Hardware handshaking - disabled

b. DNCX (using XMODEM)
- Baud rate - This is variable and it must match the baud rate set at the VMC with the SETP command or with the CD,# command.
- Parity - N (None)
- Data Bits - 8 (Eight)
- Stop Bits - 1 (One)

- EOB (End Of Block) - CR,LF (ASCII 13,10)
- Starting and ending character - % (ASCII 37)
- Packet Data Bytes - 128
- Hardware handshaking - disabled.

2. Use DOS to send a file, bypassing the communications software in the computer. If this works, the software is in doubt.

   a. From the DOS prompt type the following two DOS commands:

Type MODE COM2:2400,E,7,1   MODE port:parity,data bits,stop bits

MODE	=	*DOS command (followed by a space)*
port	=	*COM2*
:	=	*: (the : is required DOS syntax)*
baud	=	*2400*
,	=	*, (the , is required DOS syntax)*
parity	=	*EVEN*
,	=	*, (the , is required DOS syntax)*
data bits	=	*7*
,	=	*, (the , is required DOS syntax)*
stop bits	=	*1*

Type TYPE C:\CNCDATA\PN1234.NC>COM2TYPE pathnamefilename>port

TYPE	=	*DOS command (followed by a space)*
pathname	=	*C:\CNCDATA\ substitute the drive-and-directory(ies) where the file being sent to the VMC is stored*
filename	=	*N1234.NC substitute the name of the file sent to the VMC*
>	=	*> (the > is required DOS syntax to redirect the file to COM2*
port	=	*COM2*

- Note: Before typing these two lines, the VMC should be in the TA,1 mode ready to receive the program at 2400 baud.
- Refer to the DOS manual for answers to additional questions about the DOS commands used here.

3. Running DOS-based software from Windows may not work for communications.

4. Introducing another computer into a situation where communications are not happening may be useful only if the computer has been proven to work in other situations. Otherwise the second computer may also have a problem.

Table 14-2: Fadal Software Error Codes

04	OUT OF DATA	52	BAD FILE NUMBER
05	ILLEGAL FUNCTION CALL	53	FILE NOT FOUND
06	OVERFLOW	54	BAD FILE MODE
07	OUT OF MEMORY	55	FILE ALREADY OPEN
11	DIVISION BY ZERO	57	DEVICE I/O ERROR
12	ILLEGAL DIRECT	58	FILE ALREADY EXISTS
13	TYPE MISMATCH	61	DISK FULL
14	OUT OF STRING SPACE	62	INPUT PAST END
15	STRING TOO LONG	63	BAD RECORD NUMBER
16	STRING FORMULA TOO COMPLEX	64	BAD FILE NAME
17	CAN'T CONTINUE	67	TOO MANY FILES
18	UNDEFINED USER FUNCTION	68	DEVICE UNAVAILABLE
19	NO RESUME	69	COMMUNICATION BUFFER OVERFLOW
21	UNPRINTABLE ERROR	70	DISK WRITE PROTECTED
22	MISSING OPERAND	71	DISK NOT READY
23	LINE BUFFER OVERFLOW	72	DISK MEDIA ERROR
24	DEVICE TIMEOUT	74	RENAME ACROSS DISKS
25	DEVICE FAULT	75	PATH/FILE ACCESS ERROR
27	OUT OF PAPER	76	PATH NOT FOUND

Table 14-3: CNC Main Errors

150	“THE CNC NEEDS TO BE UPGRADED TO USE THIS PROGRAM”
151	”THIS VERSION OF CNCMAIN IS INCOMPATIBLE WITH YOUR CNC“
152	”CANNOT ESTABLISH LINK WITH CNC“
153	”INTERNAL ERROR: “
154	”CNC WILL NOT RESPOND“
155	”CANNOT RESPOND TO CNC“
156	”NO RESPONSE FROM CNC“
157	”INTERNAL ERROR: R“
158	”CNC DOES NOT RESPOND“
159	”OPERATION ABORTED“
160	”CNC REQUIRES A COLD START“
161	”NO SUCH STARTING BLOCK“
162	”SERIAL PORT NOT RESPONDING“
167	”STARTING BLOCK NOT FOUND“
170	”TOO MANY RETRIES ON A SINGLE PACKET“
180	”CANNOT FIND FILE“
181	”NOT AVAILABLE IN DESKTOP VERSION“

Table 14-4: Syntax Errors

190	“ALPHA CHARACTER MUST SEPARATED BY NUMBERS”
191	”DATA INPUT ERROR”
192	”ILLEGAL CHARACTER”
193	”ILLEGAL ENDING CHARACTER”
194	”ILLEGAL STARTING CHARACTER”
195	”CANNOT FIND NESTED FILE”
196	”FILES NESTED MORE THAN 3 DEEP”
251	”THIS PROGRAM REQUIRES A MOUSE”
253	”THIS PROGRAM REQUIRES A VGA COLOR MONITOR”
254	”.SYS FILE IS MISSING”
255	”INCORRECT VERSION OF .SYS FILE”

15.0 TOUCH PROBES

15.1 TOUCH PROBES

15.1.1 TOUCH PROBE - LENGTH OFFSET

The tool setting probe is used with the UTILITY command to establish the length offset. It can also be used for tool breakage detection and setting tool diameter offsets. The UTILITY command is used to set diameter and locate the length offset. Enter the command UT and the following menu appears:

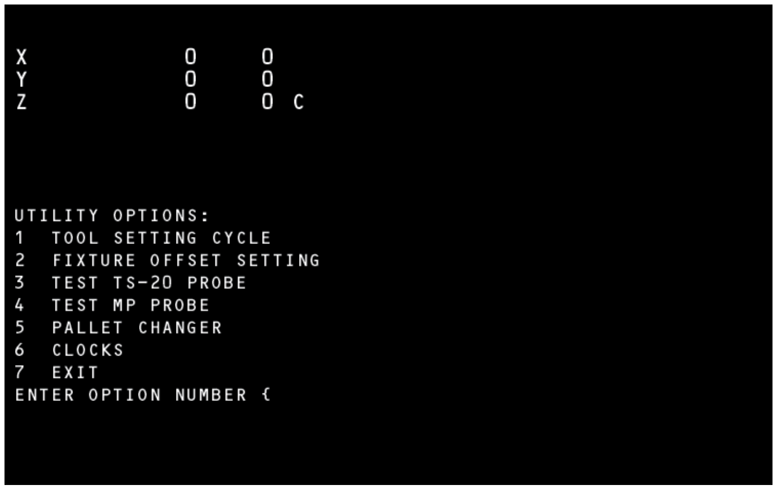


Figure 15-1: Utility Options Menu

15.1.2 TOOL SETTING CYCLE

The utility command provides two basic functions to aid in the setup procedure. The first function steps the operator through the process of retrieving tools from the turret, entering the diameters and locating the length offsets.

After selecting this cycle the CNC displays the following menu:

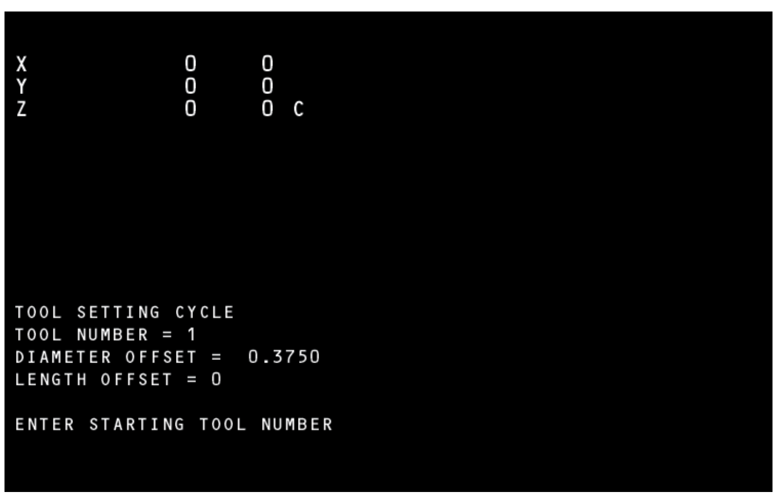


Figure 15-2: Tool Setting Cycle Menu

The information regarding the current tool in the spindle is displayed and the CNC prompts the user for the starting tool number. After entering the first tool number, the CNC responds with the message:

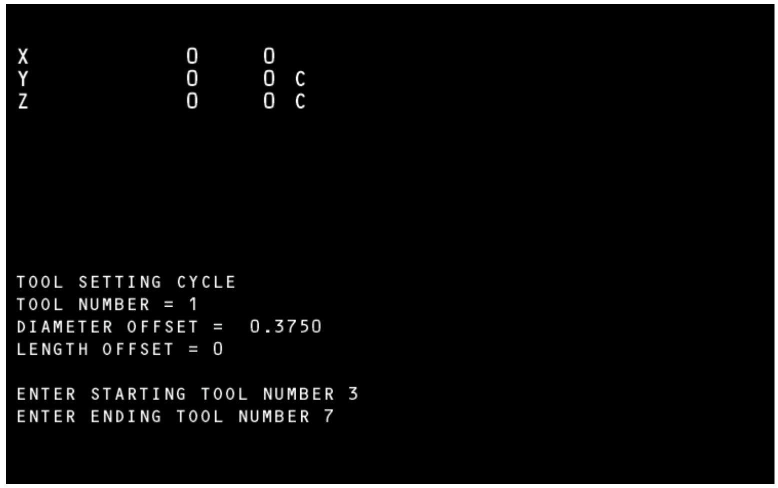


Figure 15-3: Starting Tool Number Prompt

Respond by entering the last tool number to be set.

The following menu appears after the starting and ending tool numbers are defined:

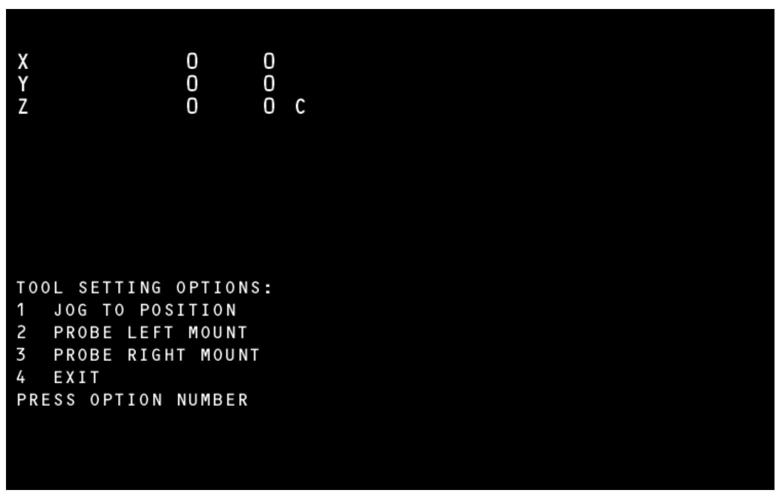


Figure 15-4: Tool Setting Options Menu

15.1.3 OPTION 1=JOG TO POSITION

This option is used for locating the tool length offset by using the JOG function of the CNC. The CNC prompts the operator to press the JOG key and manually move to the gauge point instead of the automatic process using the probe.

15.1.4 OPTION 2 & 3 - PROBE MOUNT

The CNC allows for two probe mounting positions. The left or right orientation is defined as standing in front of the machine, looking Y+ direction.

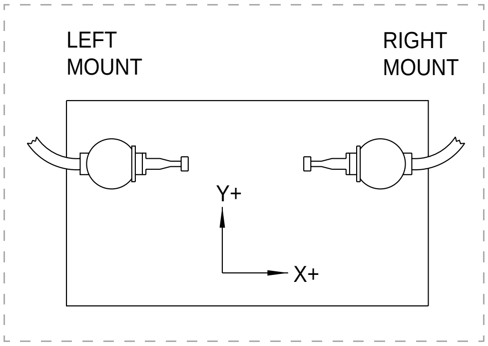


Figure 15-5: Probe Mounting Positions

Selecting left or right mount determines the X axis shift direction when locating the tool length of an end mill. The left mount probe shifts the X axis in the positive direction before moving Z down. The right mount shifts the X axis in the negative direction before moving Z down.

15.1.5 PROBE FIXTURE OFFSET NUMBER (0, 1-48)

Once the probe is installed, a fixture offset can be dedicated to tool setting. The X and Y fixture offset value is the location of the center of the tool setter stylus relative to the HOME position.

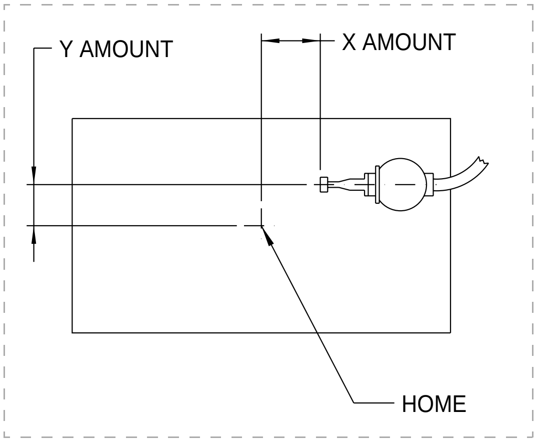


Figure 15-6: Probe Fixture Offset Number

Respond with 0 to ignore this feature or enter the fixture offset number.

15.2 PROBE OFFSETS

15.2.1 Z FIXTURE OFFSETS

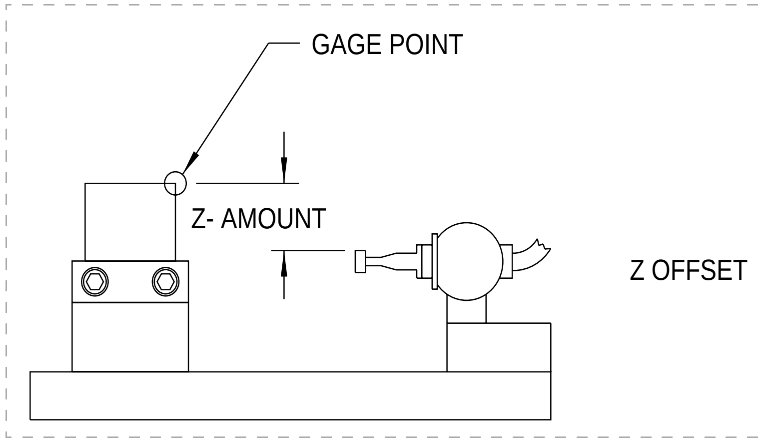


Figure 15-7: Z Fixture Offsets

After selecting a fixture offset the CNC displays the current X, Y, and Z values and prompts the user with the following message:

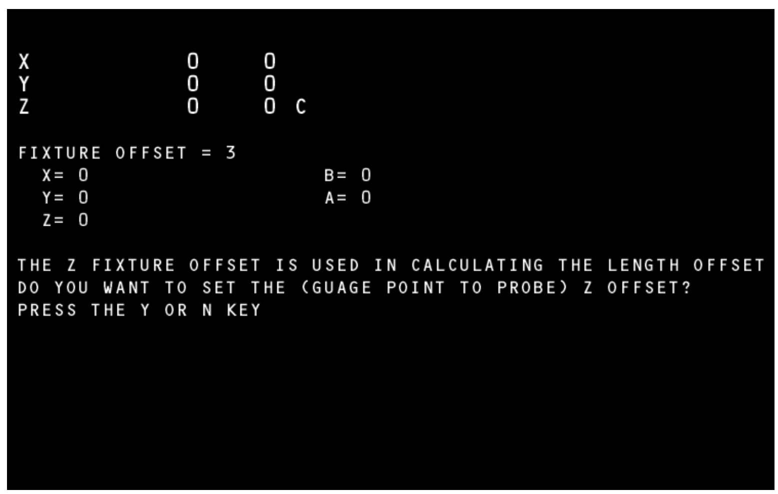


Figure 15-8: Z Fixture Offset Message

Respond by pressing the Y key and begin the process of establishing the Z fixture offset. Then press the N key and the CNC continues the setup procedure using the current Z value.

When a fixture offset is selected, the Z fixture offset value is subtracted from the current touch position when determining the length offset. A positive value increases and a negative value shortens the length offset.

After selecting a fixture offset, the CNC moves X and Y axes to the fixture offset, loads the first tool, and begins the tool setting process by displaying the following menu:

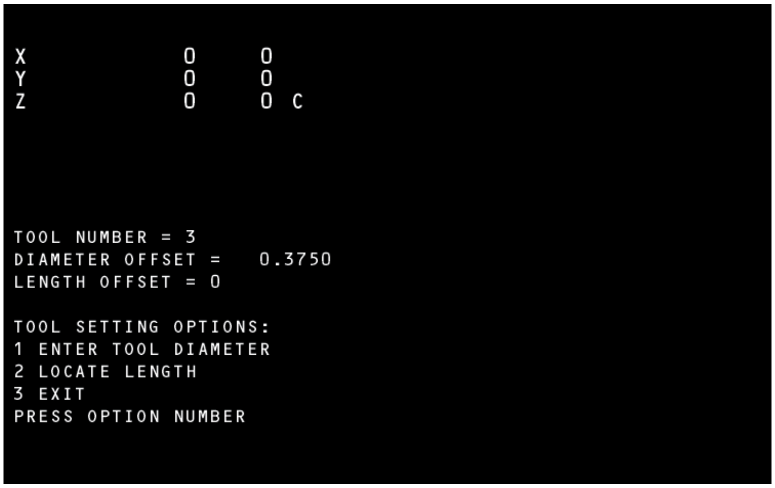


Figure 15-9: Tool Setting Options Menu

If the program requires the diameter or the tool is an end mill, the operator selects function 1 and enters the diameter. Select function 2 to begin locating the length offset.

15.2.2 LOCATING LENGTH USING THE JOG FUNCTION

The CNC displays the message:

PRESS JOG AND MOVE TO HEIGHT BLOCK OR PRESS MANUAL TO EXIT

The operator then presses the JOG key and moves the tool to the desired offset position. When the tool is at the desired location press the MANUAL key.

15.2.3 LOCATING LENGTH USING THE PROBE

Selecting function 2 causes the CNC to move the tool down in one of two ways:

1. If a diameter is entered, the X axis shifts the radius amount, spins the tool backwards, then locates the length.
2. Specifying a zero tool diameter causes the CNC to move the Z down without shifting the X axis or spinning the tool.

The CNC performs the following actions for locating the length of an end mill:

a. Spindle ON reverse at 100 RPM
b. X axis is shifted the radius of the tool
c. Z axis down until initial touch
d. Z axis is raised .010"
e. Spindle speed is increased to 500 RPM
f. Z axis down until final touch
g. Spindle off, length offset is stored

h. Z axis returned to COLD START
i. X axis shift position returned

After locating the length by either the JOG or PROBE method, the CNC automatically performs a tool change to get the next tool and displays the following menu:

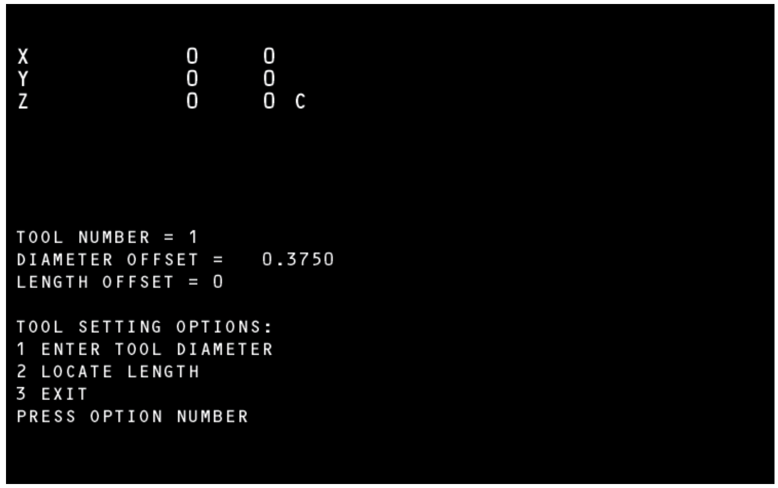


Figure 15-10: Tool Setting Options Menu

The process is repeated until the last tool has been set.

15.2.4 TOOL BREAKAGE DETECTION

Tool breakage detection is achieved by a two step process as follows:

1. The CNC is programmed to touch the probe. The tool touch is programmed as a normal program operation except using the G31 P1 code. The G31 is used to stop probe motion. When coded with axis motion, the motion terminates with the probe touch.

2. The L9101 R1+6. code is used to perform the touch check. After performing the touch check, the CNC aborts the program if the tool did not touch, otherwise the program continues.

EXAMPLE: Breakage detection for a .25" diameter drill.

N1 G0 G90 E24 X0 Y-.5
N2 H2 Z-.1 M65
N3 G1 G31 Y0 F25. P1
N4 L9101 R1+6.

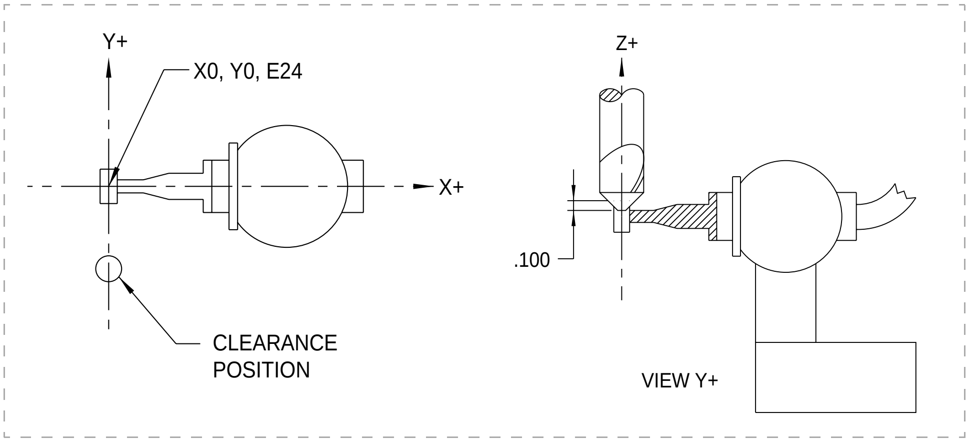


Figure 15-11: **Clearance Position & Touch Probe Selection**

N1 moves X and Y to the clearance position (Left Figure Above).

N2 moves the tool .100 below the top of the probe (Right Figure Above) and selects the Touch probe.

N3 moves the tool to the center of the probe. Because the block contains the G31 code, the CNC stops moving as soon as the tool touches the probe. The P word used in combination with the G31 causes the CNC to store the touch location.

N4 verifies the previous point (P1) has been interrupted by the probe. A broken tool causes the move to go to completion (without a touch) and abort the operation. (See PROBE L9101 FUNCTIONS in this section.)

15.2.5 TOUCH PROBE - TOOL DIAMETER OFFSET

The tool diameter offset is achieved by a two step process as follows:

1. The CNC is programmed to touch the probe at two points.
2. The L9101 R1+8. code is used to perform the calculation.

EXAMPLE: After the length offset has been located, the following program is used to establish the diameter of a 1/2" end mill.

N1 G0 G90 S500 M4 E24 X0 Y-.5 (.200+.25+.05
N2 H1 Z-.1 M65
N3 G1 G31 Y0 F5. P1
N4 G0 Z.1
N5 Y.5
N6 Z-.1
N7 G1 G31 Y0 P2
N8 L9101 R1+8. R2+.4 D1

N1: The E24 shifts the XY zero to the center and the Z zero to the top of the stylus. The X0 moves to the center of the stylus. The Y-.5 moves to a clearance position, calculated as follows:

1/2 the width of the stylus: .200

1/2 the approximate tool diameter: .250

Clearance: .050

N2: moves the tip of the tool .100" below the top of the stylus while spinning the tool backwards at 500 RPM.

N3: moves to touch point 1.

N4: moves Z .100 above the stylus.

N5: moves to a clearance position in preparation for the next touch.

N6: moves Z below the top of the stylus.

N7: moves to touch point 2.

N8: performs the diameter calculation.

The stylus width is specified by R2. The D word specifies the diameter is to be stored as offset 1 in the tool table.

15.2.6 GENERAL RULES TO FOLLOW: MP SERIES PROBE

1. Start the program by selecting the probe. M64 selects the MP Series probe, M65 selects the TS Series probe.
2. A move with the G31 must be a linear (G1) move.
3. No other codes are allowed with the G31 except G1, P# (Point Number), and feed rates.
4. The Probe functions may only use three points for each calculation, P1, P2, and P3.
5. The probing is to be in the absolute mode (G90).
6. CRC, Mirror Image, Rotation and Drill Cycles are not allowed during the execution of the G31 code.

15.2.7 LOCATING THE POINTS

There are two procedures available to locate and store the points:

1. Using the G31 P# codes.
2. Using function 1 of the L9101 fixed subroutine.

15.2.8 G31 PROBE TOUCH FUNCTION

The G31 is only used in conjunction with a probe. This code causes the machine to stop motion when the probe is touched and then execution continues at the next line in the program. The G31 can be used with table or spindle probes (See also G31.1).

The motion can be defined in absolute or incremental terms. The positions can be stored with a P word, a macro V variable, and output through the RS232 port. All G31 moves must be G1 linear moves. No G0, G2, or G3 moves are allowed.

- Rotation can be in effect when the G31 is used.
- CRC should not be in effect when G31 is used.
- Mirrored axes should be canceled before using the G31 code.
- Fixed cycles need to be canceled before using the probe.

> NOTE
> Program a move that would normally be excessive. For example, if a one inch move is required to get the probe up to a wall, use a two inch move in the program. The probe will stop the motion, and whatever motion is remaining, for that line, will be discarded and the control will continue execution of the program at the next line.

Expect some over travel if the feed rate used with the move is high, and also if the stylus in the probe is long. A method for accuracy would be to use the G31 and the G31.1 codes together. Use the G31 at a high feed rate to get up to the wall. With the high feed rate, the stylus is angled and over the edge because of the time required to read the probe and stop the motion. Then reverse the motion to move away from the wall with the G31.1 code in the line. Do this at a slow feed rate. At F1. the motion is slow enough that it will usually stop within one tenth. The G31.1 will stop motion when the probe is not touching. This means that the stylus will be perpendicular to the table and directly at the edge of the wall when the probe is not touching.

If the stylus is not running true, or a chip is in the spindle, the probe will not give a true position reading. For consistency use an M19 to orient and lock the spindle at the same position each time the probe inserted in the spindle. If an operator is to place the probe in the spindle by hand, orient the spindle prior to inserting it in the spindle. Sometimes the stylus will work itself loose, confirm that it is tightly screwed in before using the probe.

15.2.9 STORING PROBED POSITIONS

Saving positions through the RS-232 port:

1. Any software designed to save data from the port will be sufficient to retain the data.
2. When a touch is made, the motion will stop and the current position wIll be outputted through the port.

G1 G31 X1. F50. This line sends just the X axis location to the port
G1 G31 X2. Y5. F50. This line sends the X and Y locations to the port
G1 G31 X3. Y-4. Z-2. F50. This line sends the X, Y, and Z locations to the port

3. Macro SPRINT statements can be used just before the probe line to identify the information being saved

#SPRINT “PROBE TOUCH #1:” G1 X1. Y1. G31

EXAMPLE:

SAVING POSITIONS TO P WORDS:

1. P1, P2, and P3 are used to save the touch positions when the fixed probe subroutines are going to be used in the program.

G1 X3. Y-6. G31 P1   The first touch position is saved to P1
X0 Y0 G5
G1 X0 Y6. G31 P2   The second touch position is saved to P2
X0 Y0 G5
G1 X-3. Y-6. G31 P3   The third touch position is saved to P3
L9101 R1+2.   Use probe fixed subroutine function #2 to find center

2. P1, P2, and P3 can be used with the macro PX1-3, PY1-3, PZ1-3, PA1-3, and PB1-3 variables. When a probe touch (G31) or probe no-touch (G31.1) is used on a line

with a P1, P2, or P3 each axis position is stored regardless of the axis that moved to get the touch point.

G90 G0 X3. Y-6.
Z1. H21
G1 F30. G31 Z-2.
F1. Z0 G31.1 P1   P1 has stored the XYZAB position at this line
#V1=PZ1
#PRINT “THE TOUCH POINT IS: X”,PX1,", Y",PY1,", AND Z",PZ1

EXAMPLE:

SAVING THE POSITION AS A V VARIABLE:

1) The current position can be saved to a V variable by using a macro AX, AY, AZ, AA, or AB command.
G90 G1 X4. Y4. G31 F50.
X0 Y0 G31.1 F1.
#V1=AX This saves the X position to V1#V2=AY This saves the Y position to V2

15.2.10 G31.1 PROBE NO TOUCH FUNCTION

This code causes the machine to stop motion when the probe is not touching and then execution continues at the next line in the program. The G31.1 can be used with table or spindle probes. This code functions exactly like the G31 code. See also G31.

15.2.11 USING G31

During the probing function, the CNC is programmed as normal. Programming a G31 with an axis move causes the CNC to monitor the probe interface and stop all motion in the event of a touch.

The CNC stores three touch points in memory. Coding a P1, P2, or P3 in the same block with the G31 designates the touch point to stored. This code also sends the touch or no touch data to the RS-232 serial port.

EXAMPLE:

G1 G31 F25. X10. P1
Moves the X axis until the probe touches or the move is completed

EXAMPLE:

IF THE PROBE TOUCHES:

The axis motion stops, the X, Y, Z, A, B locations are stored as point 1 and the program continues.

EXAMPLE:

IF THE PROBE DOESN’T TOUCH:

The CNC stores a value of .100E9 (to signify no touch) as the X location of point #1 and continues the program.

The following example shows how to locate three points inside a 3" diameter hole. The approximate center is at X0 and Y0. The Z0 is at the top of the diameter.

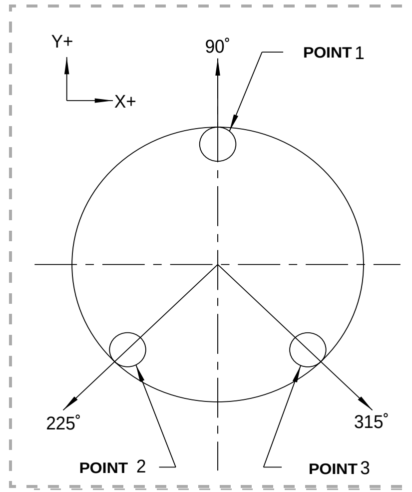


Figure 15-12: Locating Points

N1 M6 T1 (LOAD THE PROBE
N2 G0 G90 X0 Y0 (POSITION TO THE APPROX. CENTER
N3 Z-.25 H1 M64 (MOVE TIP .25 BELOW TOP OF PART
N4 G1 G31 P1 F25. Y1.8 (FIRST POINT
N5 F150. Y0 (MOVE OFF PART AND RETURN
N6 G31 P2 F25. X-1.273 Y-1.273 (APPROACH AT 225 DEGREES
N7 F150. X0 Y0 (MOVE OFF PART AND RETURN
N8 G31 P3 F25. X+1.273 Y-1.273 (APPROACH AT 315 DEGREES
N9 F150. X0 Y0 (MOVE OFF PART AND RETURN

> NOTE
> After performing a G31 touch, before another G31 is programmed, the probe must be moved off the part as in N5 and N7 above.

15.2.12 L9101 PROBE FUNCTIONS

The L9101 fixed subroutine has 10 probe functions available:

1. LOCATE TOUCH POINT
2. CENTER LOCATION AND RADIUS
3. PART ORIENTATION
4. MID-POINT AND ANGLE
5. Z DATUM LOCATION
6. TOOL BREAKAGE DETECTION

7. TOUCH/POSITION CHECK
8. COMPUTE DIAMETER
9. SET PROBE CALIBRATION
10. SET TOUCH POINT

The code L9101 is used to call a probe function, the R word R1 selects the specific function. For example: L9101 R1+2. Selects function #2 - CENTER LOCATION AND RADIUS.

Upon completion of the L9101, the R words R1-R3 contain the results. The R words can then be used as indirect references throughout the remainder of the program.

The touch points are retained in memory until power is removed, thus making it possible to do a mid-program start after the points have been located. As in circular motion, the G17, G18, and G19 modes determine the output of the L9101 subroutine. G17= XY, G18= ZX, and G19= YZ.

For example, use function 2 to compute the center location of 3 points. The logical X is returned in R1. When G18 is in effect, R1 contains the physical Z center location.

The probing process is designed to be used as a three step process:

1. Locate the points.
2. Use fixed subroutine L9101 to perform the calculations.
3. Continue the program using the R words returned by step 2.

15.2.13 LOCATE TOUCH POINT

FUNCTION #1: Locates Touch Point

Coding: L9101 R1+1. X, Y or Z moves, Approach/Return Feed
P1, P2 or P3 define point number
Result: Stores Location Specified By P word
R1= Logical X Touch, R2= Logical Y Touch

Moves to locate a touch point, apply the probe correction and return to the starting position. This function uses a two touch process. The first touch locates the initial point, the second touch is performed at a slow feed rate for best accuracy.

EXAMPLE:

EXAMPLE CODING:

L9101 R1+1. X1. Y1. F50. P1
Generates the following incremental motion:
X1. Y1. F50. G31 G1 (INITIAL TOUCH
X.0035 Y.0035 F10. (OVER TRAVEL PROBE SWITCH
X-.0707 Y-.0707 F10. G31.1 (MOVES UNTIL NO TOUCH
X-.0035 Y-.0035 F10. (MOVES OFF .005 MORE
X.0707 Y.0707 F.5 G31 P1 (FINAL TOUCH

The following program uses function 1 to locate the 3 points of a circle, instead of G31 as in the previous example.

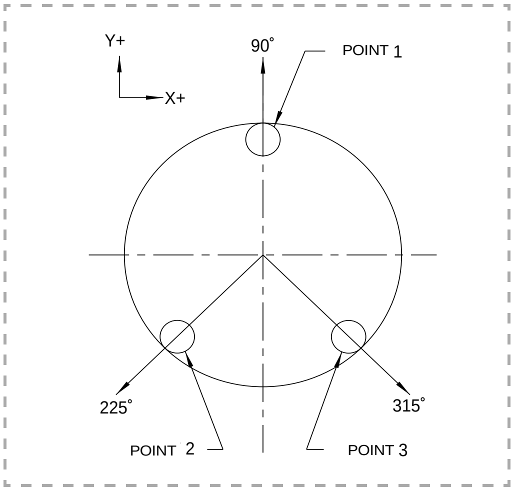


Figure 15-13: **Point Location with Function 1**

N1 M6 T1 (LOAD THE PROBE
N2 G0 G90 X0 Y0 (POSITION TO APPROX. CENTER
X-1. Y-1. F50. G1 (RETURN TO START POSITION
N3 Z-.25 H1 M64 (MOVE .25 BELOW TOP OF PART
N4 L9101 R1+1. P1 F25. Y1.8 (FIRST POINT
N5 L9101 R1+1. P2 F25. X-1.273 Y-1.273 (APPROACH AT 225 DEGREES
N6 L9101 R1+1. P3 F25. X+1.273 Y-1.273 (APPROACH AT 315 DEGREES

15.2.14 COMPUTE CENTER AND RADIUS

FUNCTION #2: Compute Center Location and Radius

Coding: L9101 R1+2.
Points Used: P1, P2, P3
Results: R1= Logical X, R2= Logical Y, R3= Radius

This function computes the center location and radius relative to the three touch points. The X center position is returned in R1, the Y center position returned in R2. and the radius returned in R3. This function may be used with the ID or OD of a circle. The following is a sample program to locate a center and perform a drilling operation at the center of the circle, using the ID of the circle:

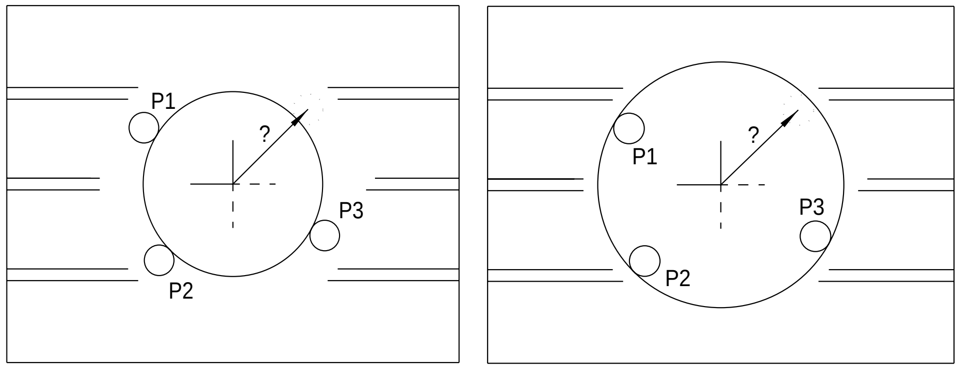


Figure 15-14: Circle Center Location

N1 M6 T1 (LOAD PROBE
N2 G0 G90 X6. Y0. (POSITION TO CENTER
N3 Z-.25 H1 M64 (POSITION Z
N4 L9101 R1+1. X5. Y1. F25. P1 (LOCATE POINT 1
N5 L9101 R1+1. X5. Y-1. F25. P2 (LOCATE POINT 2
N6 L9101 R1+1. X7. Y-1. F25. P3 (LOCATE POINT 3
N7 L9101 R1+2. (COMPUTE CENTER
N8 M6 T2 (LOAD DRILL
N9 G0 X+R1 Y+R2 (POSITION TO CENTER
N10 H2 Z.1 (BEGIN DRILLING OPERATION
N11 G1 F10 Z-2.0

15.2.15 PART ORIENTATION

FUNCTION #3: Part Orientation

Coding: L9101, R1+3., R2= Expected Angle from P2 to P3
Points Used: P1, P2, P3
Results: R1= Logical X, R2= Logical Y, R3= Angular Error

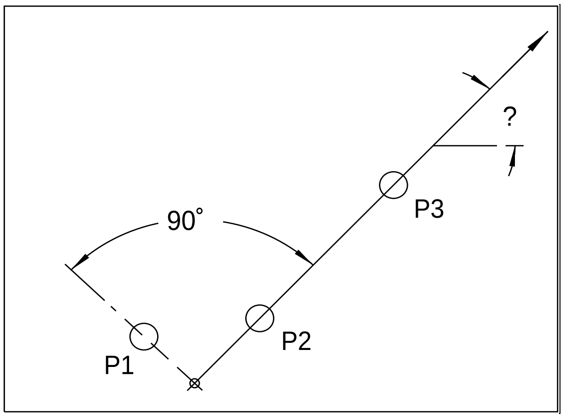


Figure 15-15: Intersection Point & Angular Correction

This function computes the location of the intersection point and angular correction needed for program rotation. The computed intersection point assumes P1 has a 90 degree relationship to the line created from P2 to P3.

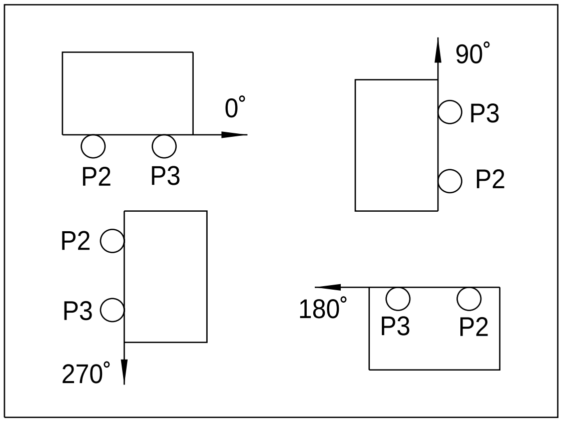


Figure 15-16: Expected Angle Changes

The expected angle is related to the touch surface. The examples above illustrate how the expected angle changes, depending upon how P2 and P3 touch the part.

The angle is coded in decimal degrees, 0 degrees starts at the X+ direction and increases in the counterclockwise direction.

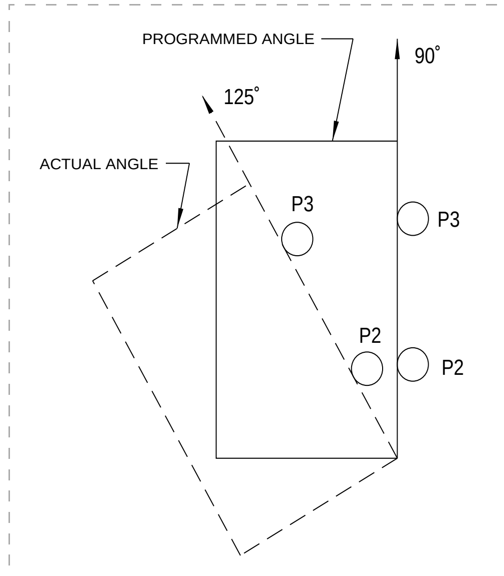


Figure 15-17: Programmed Angle P2 to P3

The angular error returned in R3 is the result of subtracting the expected angle from the probed angle (P2 to P3). This angle (R3) becomes the angular correction needed for program coordinate rotation (G68). The sample above shows the programmed angle from P2 to P3 to be 90 degrees. The angle necessary for program rotation would then be 35 degrees if the actual angle is 125 degrees.

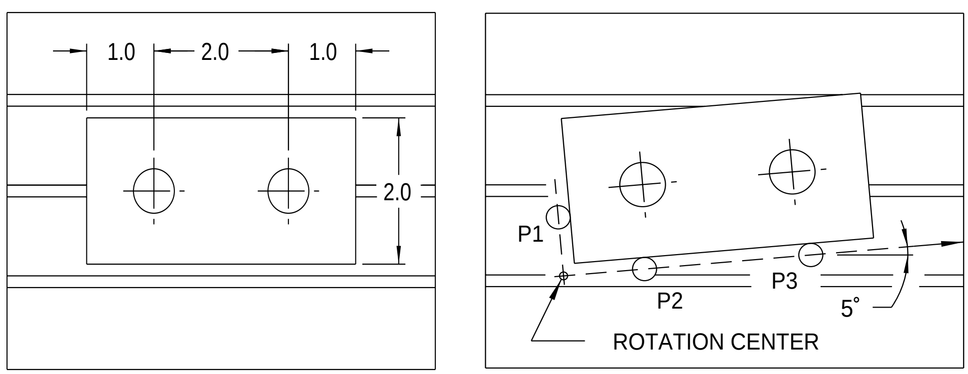


Figure 15-18: Sample Program to Drill Two Holes in a Part

The example above is a sample program to drill two holes in a part. The program XY zero position is approximately the lower left corner of part.

The probe stylus is a .25" diameter.

N1 M6 T1(LOAD PROBE

```
N2 G0 X-.5 Y1.  (CLEARANCE POSITION FOR POINT 1
N3 Z-1.0 H1 M64
N4 L9101 R1+1. X.5 F25. P1  (MOVE TO TOUCH POINT 1
N5 Z0 G0
N6 X1.0 Y-.5  (CLEARANCE POSITION FOR POINT 2
N7 Z-1.0
N8 L9101 R1+1. Y.5 F25. P2  (MOVE TO TOUCH POINT 2
N9 G0 Z-1.0
N10 X5.0 Y-.5  (CLEARANCE POSITION FOR POINT 3
N11 Z-1.0
N12 L9101 R1+1. Y.5 F25. P3  (MOVE TO TOUCH POINT 3
N13 L9101 R1+3. R2+0  (CALCULATE PART ORIENTATION
N14 G90 G0 H0 Z0
N15 M6 T2  (LOAD DRILL
N16 G90 G0 X+R1 Y+R2 S10000 M3  (MOVE TO INTERSECTION POINT
N17 G92 X0 Y0  (SET ABSOLUTE LOCATION
N18 G68 R0+R3 X0 Y0  (SET ROTATION
N19 Z.1 H1 M8
N20 G81 G99 Z-1.0 R0+.05 F50. X1.125 Y1.125  (DRILL CYCLE
N21 X3.125  (DRILL SECOND HOLE
N22 G69 G80
```

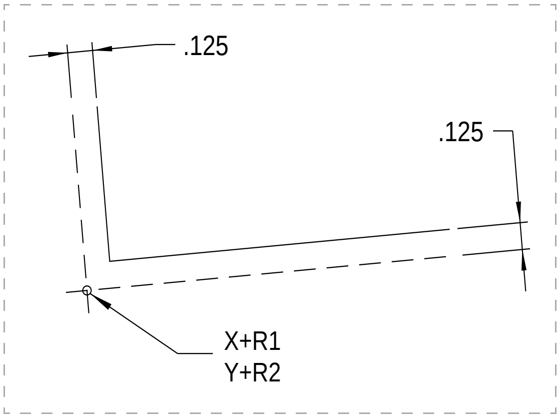


Figure 15-19: CNC Movement to Corner

After probing the part and establishing the rotation, programming X.125 Y.125 causes the CNC to move the center of the tool to the corner of the part.

15.2.16 MID-POINT AND ANGLE

Function #4: Mid-Point and Angle

Coding: L9101, R1+4., R2= Expected Angle
Points Used: P1, P2
Results: R1= Logical X, R2= Logical Y, R3= Angular Error

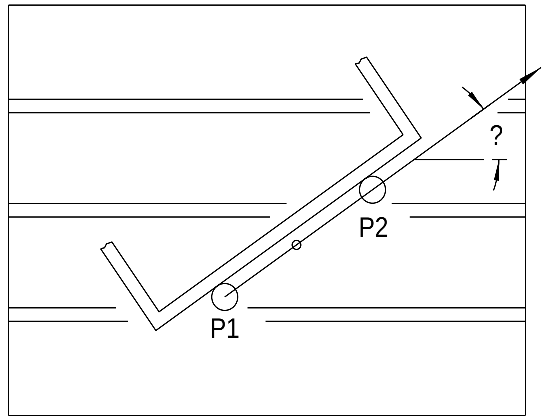


Figure 15-20: Mid-Point and Angle

This function computes the location of the point between P1 and P2 and the angular correction needed for program rotation. The angular error is determined the same as function 3 - PART ORIENTATION previously described.

The following are two examples using this function:

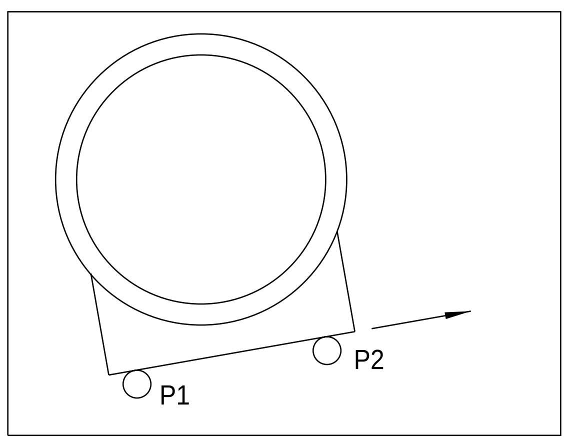


Figure 15-21: Part with Unknown Rotation

The example above shows a part having an unknown rotation. After using this function, the R3 contains the angular correction needed for rotation.

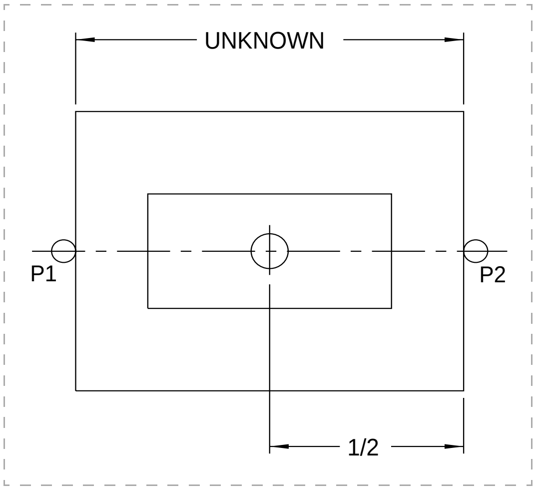


Figure 15-22: **Part with Unknown Width**

The example above shows a part having an unknown width. After using this function, the R1 contains the location of the midpoint.

15.2.17 Z DATUM LOCATION

FUNCTION #5: Z Datum Location

Coding: L9101, R1+5., Optional Z Modifier
Points Used: P1
Results: R1= Distance From Length Offset Position To Z Datum

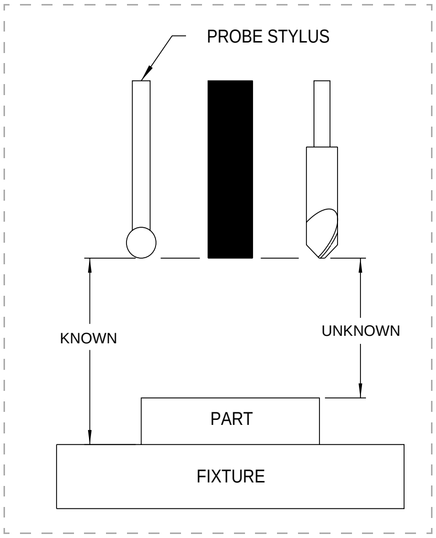


Figure 15-23: **Z Datum Location**

This function is used to calculate the distance between an unknown surface and the gauge point (Tool Length Offset Position).

The procedure is as follows:

1. Setup: A length offset is set for the probe as a normal tool.
2. Programming:
    a. The probe is moved to the gauge point using the H word.
    b. The Z touch for P1 is found using G31 or function 1 of L9101.
    c. Function 5 is used to calculate the distance.
    d. The R1 is referenced as the distance to the surface.

The following example shows how to locate the top surface and remove .010":

N1 M6 T1   (LOAD PROBE
N2 H1 M64   (MOVE TO OFFSET, SELECT MP8 PROBE
N3 G1 G31 F25. Z-30. P1   (LOCATE P1
N4 L9101 R1+5. Z-.010   (CALCULATE, ADD -.01
N5 M6 T2   (LOAD END MILL
N6 M3 S10000   (SPINDLE ON
N7 Z+R1 H2   (POSITION Z
N8 G1 F100. X10.   (MACHINE .010 OFF TOP

Block 7 above positions the Z axis .01 below the touch position. Another method available is to use the G92 preset code as follows:

N6 M3 S10000 (SPINDLE ON
N7 H2 (POSITION Z
N8 G92 Z-R1 (PRESET ABS. Z LOCATION
N9 Z0
N10 G1 F100. X10. (MACHINE .010 OFF TOP

Using the G92 code allows the absolute Z location to be relative to the touch position for the remainder of the program.

When machining multiple parts, the Z datum can be stored as a fixture offset using G10 L02 Z-R1 P#.

15.2.18 TOOL BREAKAGE DETECTION

FUNCTION #6: Tool Breakage Detection

Coding: L9101, R1+6.
Points Used: P1
Results: A No Touch Causes The CNC Program To Stop

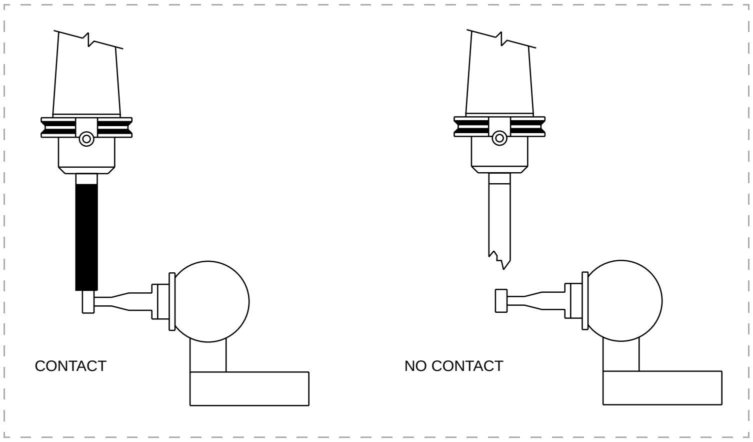


Figure 15-24: Tool Breakage Detection

This function is used to check if P1 had a successful Y or Z touch position stored. A block containing a G31, P1 with a move that doesn't touch the tool setting probe causes the CNC to store a value to signify a touch was not made.

See 15.2.3 USING THE TOUCH PROBE - 15.2.4 TOOL BREAKAGE DETECTION" previously described.

15.2.19 TOUCH CHECK, POSITION CHECK

FUNCTION #7: Touch Check, Position Check

Coding:
Touch Check: L9101, R1+7., R2= False #, R3= True #
Position Check: L9101, R1+7., R2= False #, R3= True #
R4= Approach Direction, X, Y, Z, A or B # Points Used: P1
Results: R2= True # or False #

The following diagram describes this function:

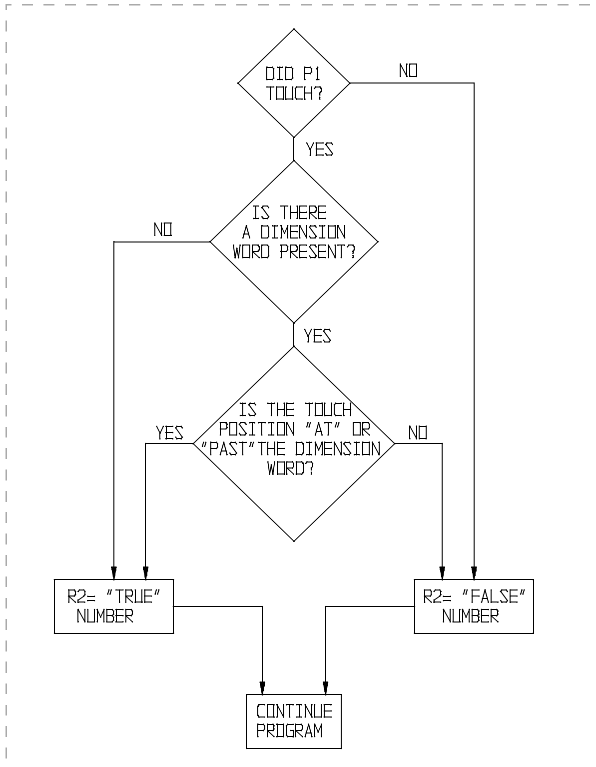


Figure 15-25: Touch Check, Position Check

An X, Y, Z, A, or B dimension word included with the L9101 block is used for the position check. Motion will not occur; the CNC uses the axis word only for the position check. The R4 indicates approach direction; R4+1.= Positive, R4-1.= Negative.

EXAMPLE: L9101 R1+7. R2+10. R3+20. R4-1. Y-2.5

1. Approaching negative with a touch at Y-3.0 returns R2 with a value of 20.
2. Approaching negative with a touch at Y+1.0 returns R2 with a value of 10.

15.2.20 TRUE OR FALSE COMPARISON

FUNCTION #7: Continued

Coding:
Touch Check: L9101, R1+7., R2= False #, R3= True #
Position Check: L9101, R1+7., R2= False #, R3= True #
R4= Approach Direction, X, Y, Z, A or B # Points Used: P1
Results: R2=True # or False #

Using the True/False response to redirect the program allows for numerous capabilities. The following is an example how the program operation changes according to the touch position:

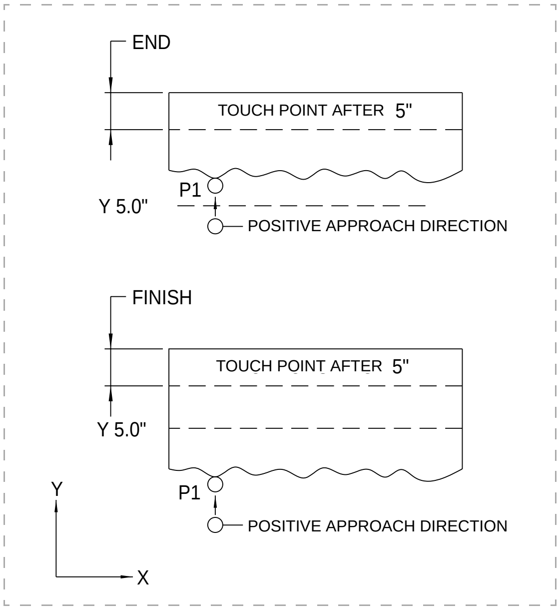


Figure 15-26: True/False Comparison

N1 M6 T1   (LOAD PROBE
N2 G0 G90 X0 Y0   (POSITION XY
N3 Z-1.0 H2 M64   (POSITION Z, SELECT MP8
N4 G1 G31 Y10. F25. P1   (MOVE TO TOUCH
N5 L9101 Y5.0 R1+7. R2+8. R3+11. R4+1.   (COMPARE POSITION
N6 M6 T2   (LOAD TOOL
N7 M99 P+R2   (“AT” or “PAST”= N11,BEFORE =
N8 G10 L12 P2 R0+.55   (SET DIAMETER FOR ROUGHING
N9 F25.0   (SET ROUGHING FEED RATE
N10 M98 P1   (CALL SUB. TO MACHINE PART
N11 G10 L12 P2 R0+.5   (SET DIAMETER FOR FINISHING
N12 F50.0   (SET FINISH FEED RATE
N13 M98 P1   (CALL SUB. TO MACHINE PART

This example finds a touch point and begins the finishing operation at N11 when the Y touch position is “AT” or “PAST” the Y+5", otherwise the roughing operation is performed.

15.2.21 CALCULATE DIAMETER

FUNCTION #8: Calculate Diameter

Coding: L9101, R1+8., R2= Stylus Width
D#= Offset Number (Optional)
Points Used: “Y” of P1 and “Y” of P2
Results: R1= Tool Diameter/Radius
Logical X of P1= Tool Diameter/Radius

This function performs the diameter/radius calculation. The value returned depends on the selection made using the SETP command. When the machine is in the DIAMETER mode the result is diameter, otherwise radius is returned. Specifying a D word in the block with L9101 causes the CNC to store the value in the tool offset table.

The X of P1 contains the result of the calculation. This enables function 7 to perform a Position Check. See 15.2.5 THE TOUCH PROBE - TOOL DIAMETER OFFSET previously described.

15.2.22 SET CALIBRATION

FUNCTION #9: Set Probe Calibration

Coding: L9101, R1+9., R2= Radial Over travel
X, Y or Z= Shift Amount
Points Used: None
Results: Used With Function #1

This function establishes the probe compensation. The L9101 Function 1 uses these values when present for error correction. In most cases this function is not needed because most tolerances are greater than the probe errors.

15.2.23 XY SHIFT ERROR

The shift error is the difference between the center of the spindle and the center of the probe. The probe has an adjustment to align the centers. The CNC XY shift capability allows another method to compensate for the error.

15.2.24 RADIAL OVER TRAVEL

Before the probe indicates a touch, the stylus must touch the surface and open the contacts inside the probe. This causes a slight over travel error. This probe error varies with the length of the stylus. For example, an over travel error of approximately .0012" occurs with a 2" stylus length. The error increases to .0043" with a 7" stylus length. Applying over travel correction improves the 2" stylus error to +-.0004" and the 7" stylus error to +-.0012.

When locating the center of a circle, the over travel of the probe doesn't change the computed center location. The radius of the circle appears larger when inside a hole and smaller when outside a boss.

15.2.25 MP8 PROBE CALIBRATION

1. Mount an inspection ring on the table.
2. Set the XY home position at the center of the ring.
3. Install the probe and run the following program:

N1 M64 G1 G90 Z-??(MOVE Z INSIDE RING
N2 L9101 R1+1. Y5.0 F25. P1
N3 L9101 R1+1. X-4.33 Y-2.5 F25. P2(TOUCH 120 DEG. APART
N4 L9101 R1+1. X+4.33 Y-2.5 F25. P3
N5 L9101 R1+2.(COMPUTE CENTER

4. Enter the command; SUM

FUNCTION #9: Continued

Coding: L9101, R1+9., R2= Radial Over Travel
X, Y or Z= Shift Amount
Points Used: None
Results: Used With Function #1

After entering the SUM command the CNC displays the point data, R words, points used and the probe correction setting as follows:

P1=0,0,0,0,0,0,0,0,0
P2=0,0,0,0,0,0,0,0,0
P3=0,0,0,0,0,0,0,0,0
R WORDS R0-R9
0,0,0,0,0,0,0,0,0,0
POINTS USED X Y

The point values are displayed in the axis order of P1=X, Y, Z, U, V, W, A, B, C.

The R values are displayed in ascending numerical as R0, R1, R2, R3, R4, R5, R6, R7, R8, R9.

15.2.26 ENTERING XY SHIFT VALUES

The negative values of R1 and R2 are used for the shift values. R1 becomes the X shift and R2 becomes the Y shift.

EXAMPLE: The probe indicates the center location to be at X.001 and Y.002 after performing the calibration. The compensation is entered as L9101 R1+9.R2+ X-.001 Y-.002

15.2.27 ENTERING RADIAL OVER TRAVEL

After the ring gauge test, R2 contains the effective radius. The amount entered for compensation is computed by subtracting the effective radius from the apparent radius.

EXAMPLE:

Ring Gauge Radius= 1.500" Stylus Radius= .125"
Apparent Radius= 1.375" Effective Radius= 1.378
1.375 - 1.378 = -.003 (Correction Needed)
The XY shift and radial correction is entered as:
L9101 R1+9. R2-.003 X-.001 Y-.002

The correction is entered in the program before L9101 function 1 is used.

15.2.28 SET COUNTER

FUNCTION #10: Set Counter

Coding: L9101, R1+10., P1= Increments the counter
by 1 X = Sets the counter starting number
Points Used: None
Results: Used With Function #7 to create IF - Then statement

This function is used to create an IF - Then statement for program redirection. The examples below use this function to perform G52 program shifts for multiple part programming.

```
%
N1O101 (SUB FOR PART MACHINING
N2G1G90F250.
N4X1.
N5Y1.
N6X0
N7Y0
N8M99
N1O9110  (IF THEN PROGRAM REDIRECTION EXAMPLE
N2L100  (SUB TO SHIFT THEN MOVE
N3G52X+R8Y+R7  (LOCAL COORDINATE SYSTEM
N4M98P101  (INPUT SUB PROGRAM NUMBER
N5M17
N6M30
N7R9+0R8+0R7+0  (ASSIGN R VARIABLES
N8L101
N9G91G10L109P1  (INCREMENT X COUNT BY +1
N10G90
N11G91G10L108P2  (INCREMENT X STEP BY +2
N12G90
N13L9101R1+10.X+R9P1  (SETS THE X VALUE TO COUNT
```

N14L9101R1+7.X3.R2+24.R3+16.R4+1. (IF COUNT IS X OR HIGHER
N15M99P+R2 (GOTO R3 ELSE GOTO R2
N16G91G10L107P-2 (INCREMENT Y STEP BY -2
N17G90
N18L9101R1+10.Y+R7P1 (SETS THE Y VALUE TO COUNT
N19L9101R1+7.Y-5.R2+21.R3+25.R4-1. (IF COUNT IS Y OR HIGHER
N20M99P+R2 (GOTO R2 ELSE GOTO R3
N21R8+0 (ASSIGN VARIABLES
N22G52X0Y+R7 (MOVE TO X0 Y(R7 VALUE)
N23R9+0 (ASSIGN R VARIABLE
N24M99P8 (GOTO LINE #8
N25M5M9
N26M2

15.3 SET TOUCH POINT

FUNCTION #10: Set Touch Point

Coding: L9101, R1+10.,X, Y, Z, P
X, Y, AND Z = Location of point
Points Used: None
Results: The P word identified by the X, Y, and Z location is used with other L9101 functions.

This function may be used to identify the centers of two bores and calculate the center and angle between them. The example below locates the center of two bores and calculates the center and angle between them.

N1 M6 T1 (LOAD PROBE
N2 G0 G90 X6. Y0. (POSITION TO CENTER OF THE FIRST BORE
N3 Z-.25 H1 M64 (POSITION Z
N4 L9101 R1+1. X6. Y1. F25. P1 (LOCATE POINT 1
N5 L9101 R1+1. X4. Y-1. F25. P2 (LOCATE POINT 2
N6 L9101 R1+1. X8. Y-1. F25. P3 (LOCATE POINT 3
N7 L9101 R1+2. (COMPUTE CENTER OF THE FIRST BORE
N8 R9+R1 R8+R2 (RENAME THE LOCATION
N9 G0 G90 X10. Y0 (POSITION TO CENTER OF THE SECOND BORE
N10 L9101 R1+1. X10. Y1. F25. P1 (LOCATE POINT 1
N11 L9101 R1+1. X8. Y-1. F25. P2 (LOCATE POINT 2
N12 L9101 R1+1. X12. Y-1.F25. P3 (LOCATE POINT 3
N13 L9101 R1+2. (COMPUTE CENTER OF THE SECOND BORE
N14 R7+R1 R6+R2 (RENAME THE LOCATION
N15 L9101 R1+10. X+R7 Y+R6 P2 (SET THE CENTER OF THE SECOND BORE TO POINT 2

N16 L9101 R1+10. X+R9 Y+R8 P1 (SET THE CENTER OF THE FIRST BORE TO POINT 1
N17 L9101 R1+4.R2+0 (COMPUTE CENTER AND ANGLE OF THE TWO BORES
N18 G90 G10 L2 P1 X+R1 Y+R2 (SET THE CENTER OF THE BORES AS FIXTURE OFFSET 1
N19 G0 G90 E1 X0 Y0 (MOVE TO FIXTURE 1 ZERO
N20 G68 X0 Y0 R0+R3 (SET ROTATION WITH THE ANGLE OF THE BORES

15.4 PROBE TUTORIAL

15.4.1 EXAMPLES IN FORMAT 2

Always test the probe to determine if it is functioning properly by typing the M64 M66 code in MDI. Then go into jog, touch the stylus, and look for the touch/no touch message.

The stylus should run true when the probe is rotated by hand. Place an indicator in a magnetic base and put the indicator tip on the end of the stylus. Rotate the probe head by hand and observe the run out. Use the adjustment screws to get the stylus to run true.

1. This example will demonstrate how the G31 code stops motion and allows the control to move to the next line in the program. The move on the line with the G31 or the G31.1 is usually a move that is beyond the desired touch point.

EXAMPLE:

G91 G1 F50.
M64 M66
X-3. G31
M0 Look at the X axis position at this point
X1. G0
M99 P1

Start by jogging the probe to approximately one inch to the right side of a solid object. The end of the stylus should be below the top of the solid object.

2. This example will demonstrate the effect of feed rate on over travel after the probe indicates to the control that a touch has been made.

EXAMPLE:

G91 G1 F50.
M64 M66
X-3. G31
M0
X1. G0

M99 P1

Start by jogging the probe to approximately one inch to the right side of a solid object. The position display will present the current X axis location. Press the START button to run this routine each time the position is displayed. Notice that the location changes over several times it is run. Next, change the feed rate on line one. Make it faster, then try it at slower feed rates. The position should repeat more consistently at the slower feed rates. The faster the feed rate, the further the machine will over travel at the touch point.

3. This example will demonstrate a two-touch method of finding a point on an edge.

EXAMPLE:

G91 G1 F50.
M64 M66
X-3. G31
X.05
F1.
X-3. G31
M0
X1.G0
M99 P1

Start by jogging the probe to approximately one inch to the right side of a solid object. The end of the stylus should be below the top of the solid object.

The first touch is at a high feed rate and is only used to get the stylus in the general area of the edge. Then the stylus moves away and the second approach is used to get a “good” point. From the second example it was demonstrated that the slower feed rate results in a more consistent touch position.

4. This example will demonstrate the use of the G31.1 code.

The G31 code will stop motion when the probe switch is opened. The G31.1 code will stop motion when the probe switch is closed. The G31.1 code is used just after a G31 code is used. The probe switch is opened when the stylus touches an edge and the motion continues to cause the stylus to open the switch. At this point, the stylus should be at an angle. It should be mentioned that the longer the stylus, the more over travel will be required to open the switch. Motion in the opposite direction can now be stopped with a G31.1 code. When the stylus is vertical, the probe switch is now closed and the motion will stop because of the G31.1 code.

EXAMPLE:

G91 G1 F50.
M64 M66
X-3. G31
F10.
X1. G31.1

M0
X1. G0
M99 P1

Try to vary the second feed rate. Again observe the X axis position. The slower feed rate will result in a more consistent final position.

5. This is a variation on the fourth example.

Start by jogging the probe to approximately one inch to the right side of a solid object. The end of the stylus should be below the top of the solid object.

EXAMPLE:

G91 G1 F50.
M64 M66
X-3. G31
X1. G31.1
F.5
X-3. G31
M0
X1. G0
M99 P1

Notice the amount of time required to pick up a point with each of these methods. Compare the time from examples 4 and 5. Also compare the positions picked up using each method. Consistency and time should be issues to be aware of.

15.5 USING THE PROBE WITH MACRO STATEMENTS

15.5.1 EXAMPLES IN FORMAT 2

Always test the probe to determine if it is functioning properly by typing the M64 M66 code in MDI. Then go into jog, touch the stylus, and look for the touch/no touch message.

The stylus should run true when the probe is rotated by hand. Place an indicator in a magnetic base and put the indicator tip on the end of the stylus. Rotate the probe head by hand and observe the run out. Use the adjustment screws to get the stylus to run true.

Macro statements can be used to make determinations from the positions picked up with the probe.

1. Use the AX, AY, AZ, AA, and AB macro statement to collect and use the current axis position.

Start by jogging the probe to approximately one inch to the right side of a solid object. The end of a stylus should be below the top of the solid object.

EXAMPLE:

```
G91 G1 F50.
M64 M66
X-3.G31
X1. G31.1
F.5
X-3.G31
#V1=AX
M0
X1. G0
M99 P1
```

The macro statement states, “Make the V1 variable equal to the current X axis location (AX). To see this value in memory, use the DV command from the command mode. The current location will be stored in the V1 variable location on the screen. These examples can all be used for the other axes, and they could all be used in absolute as well.

2. This example will demonstrate how to use a macro statement to determine the center point of the part.

Start by placing a 1-2-3 block in a vice. hold the block with the three inch sides in the jaws. Jog the probe to approximately one inch to the right side of the block. The end of the stylus should be approximately .25 below the top of the block.

EXAMPLE:

```
G91 G1 F50.
M64 M66
X-3. G31
X1. G31.1
F.5
X-3. G31
#V1=AX
Z1. G0
X-4.
Z-1.
F50.
X3. G31
X-3. G31.1
F.5
X3. G31
```

#V2=AX
#V3=(V1+V2)/2
Z1. G0
X4.
Z-1.
M0
M99 P1

The V1 variable represents the right side touch; the V2 variable represents the left side touch point. The V3 variable represents the mid-point of the block. This mid-point will be relative to the SETX position. View the V3 value in the variable table by using the DV command in the command mode.

3. This example will demonstrate the ability to alter or establish a fixture offset using the probe and a macro statement.

Start by placing a 1-2-3 block in a vice. Hold the block with the three inch sides in the jaws. Jog the probe to approximately one inch to the right side of the block. The end of the stylus should be approximately .25 below the top of the block.

EXAMPLE:

G91 G1 F50.
M64 M66
X-3. G31
X1. G31.1
F.5
X-3. G31
#V1=AX
Z1. G0
X-4.
Z-1.
F50.
X3. G31
#V2=AX
#V3=(V1+V2)/2
#FX1=V3
Z1. G0
X4.
Z-1.
M0
M99 P1

The V1 variable represents the right side touch; the V2 variable represents the left side touch point. The V3 variable represents the mid-point of the block. This mid-point will be relative to the SETX position. The FX1 macro statement is used to enter the value of

V3, which is the mid-point position, into the X value of fixture offset 1. Use the DF command and the DV command to compare the values in the fixture table and the variable table.

15.6 FIXED SUBROUTINES

15.6.1 EXAMPLES IN FORMAT 2

Always test the probe to determine if it is functioning properly by typing the M64 M66 code in MDI. Then go into jog, touch the stylus, and look for the touch/no touch message. The stylus should run true when the probe is rotated by hand. Place an indicator in a magnetic base and put the indicator tip on the end of the stylus. Rotate the probe head by hand and observe the run out. Use the adjustment screws to get the stylus to run true.The control has fixed subroutines specially designed to work with the probe. This section will discuss the use of these routines.

1. Each subroutine requires that the positions used in the calculations are stored in P variables. There are two methods to pick up and store P values:

EXAMPLE:

METHOD 1

The P1, P2, or P3 variables will store the values of the moving axes in the line with the G31 or G31.1 codes.

G91 G1 F50.
M64 M66
X-3. G31 P1
M0
X1. G0
M99 P1
The P1 on the line with the G31 will store the X value of the touch point

G91 G1 F50.
M64 M66
X-3. G31.1 P1
M0
X1. G0
M99 P1
The P1 on the line with the G31.1 will store the X value of the touch point

G91 G1 F50.
M64 M66

Z-5. G31 P1
M0
X1.G0
M99 P1
The P1 on the line with the G31 will store the Z value of the touch point

G91 G1 F50.
M64 M66
X-3. Y-3. G31 P1
M0
X1. G0
M99 P1
The P1 on the line with the G31 will store the X and Y axis values

METHOD 2

Use the L9101 subroutine function 1 to move and store the point. Note that the program must be written in absolute terms. When using the example use the SETX command to set the X axis home approximately one inch to the right of the object to touch.

EXAMPLE:

G90 G1
M64 M66
L9101 R0+1. X-3 F25. P1
M0
X1. G0
M99 P1

The function for the L9101 subroutine is selected with the R0+1. selects function 1 of the L9101 subroutine. The function requires four items:

1. The R0 variable to select the function
2. The move to the point
3. The approach feed rate
4. The desired P variable

In the example above only the X axis will be stored because it is the only axis move in the L9101 line.

Compare this method of picking up and storing a touch point to the methods discussed previously in this section. Each method will store the points needed; selecting one method over the other is a matter of programmer's preference. Note that one disadvantage of using the L9101 fixed subroutine is that it must be written in absolute terms.

EXAMPLE:

```
G90 G1
M64 M66
L9101 R0+1. X-3. Y-3. F25. P1
M0
X1. G0
M99 P1
```

In the example above, the X and Y axis positions will be stored because they are the axes in motion in the L9101 line.

16.0 COMMAND MENUS

16.1 QUICK KEYS MENU

There are three separate prompting menus within the machine control. They are the Quick Keys Menu, the Edit Menu, and the Function Menu. These menus prompt the operator for input to perform machine commands. The Quick Keys menu is used for quick setup functions. Program editing is accomplished with the Edit Menu. The Function Menu is used to perform machine tasks. These menus are activated or deactivated within the machine parameters. Use the SETP command and the CMD MENU parameter to turn the menus on or off. When the menus are off, the machine is operated by commands (See section 8.0 COMMANDS, MAN-0131). The Quick Keys Menu is selected by pressing the space bar from ENTER NEXT COMMAND.

When the power on cold start has been completed the following screen is displayed when the menus are turned on within machine parameters:

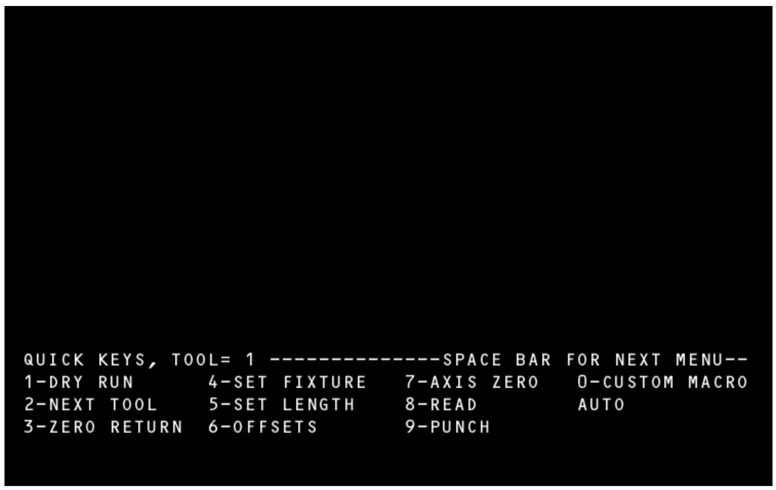


Figure 16-1: Quick Keys Menu

16.1.1 1-DRY RUN

Quick key #1 places the control in auto inside the run time menu. The operator now may choose a dry run option. After a dry run mode is selected, press the MANUAL key and then the AUTO key to start the dry run.

16.1.2 2-NEXT TOOL

Quick key #2 calls the next tool with an M6 T# command. Press the START button to load the next tool.

16.1.3 3-ZERO RETURN

Quick key #3 returns all axes to the HOME position. Press 3-ZERO RETURN then the START button.

16.1.4 4-SET FIXTURE

Quick key #4, places the control in the FIXTURE OFFSET UTILITIES MENU.

16.1.5 5-SET LENGTH

Quick key #5 sets the length of the current tool in the spindle at its current Z location.

a. Jog the current tool down to the desired part Z0
b. Press 5-SET LENGTH from the quick key menu.

16.1.6 6-OFFSETS

This Quick key #6 places the control into the OFFSET tables. By pressing the space bar, the operator may toggle between TOOL LENGTH OFFSET TABLE, FIXTURE OFFSET TABLE, TOOL TIME TABLE, and MACRO VARIABLE TABLE.

16.1.7 7-AXIS ZERO

Quick key # 7 is used to set an axis HOME position.

1. Jog the desired axis to the desired position to be called zero.
2. Press 7-AXIS ZERO from the Quick Keys Menu.
3. Choose the desired axis from the SETUP / AXIS ZERO menu.

This has now set that axis to zero. For example, if you choose X the current position is now X0. This position does not have to be the same place as cold start X0. And it is not a fixture offset location.

16.1.8 8-READ

Quick Key #8 has two options:

1- BEGIN

By pressing quick key # 8,then # 1, the control will begin receiving from the RS-232 port.

2- OPTIONS

Options to the quick key READ allows the operator to customize the input parameters.

MEMORY/INPUT/OPTIONS:
0- NORMAL ERROR CHECKING2- IGNORE ERRORS
1- IGNORE PARITY ERRORS3- OTHER FORMAT

16.1.9 9-PUNCH

Quick Key #9 will output programs and offsets through the RS-232 port.

There are 4 options as follows:

0-PROGRAM AND TOOLING   2-PROGRAM ONLY
1-TOOLING ONLY   3- ALL PROGRAMS

0- PROGRAM AND TOOLING ONLY will output the program any values in the tool offset table and any values in the fixture table. The operator is then prompted for the data format.

0-ASCII: 7 DATA BITS, EVEN PARITY, 1 STOP BITS

1-EIA : 8 DATA BITS, ODD PARITY, 1 STOP BIT

Normal output is 0-ASCII

0-COMPUTER, NO LEADER OR NULLS

1-TTY, LEADER AND NULLS

2-PROGRAM ONLY will output program data only. No tooling fixture offset data will be sent. The operator is then prompted for the data format.

0-ASCII: 7 DATA BITS, EVEN PARITY, 1 STOP BITS

1-EIA : 8 DATA BITS, ODD PARITY, 1 STOP BIT

Normal output is 0-ASCII

0-COMPUTER, NO LEADER OR NULLS

1-TTY, LEADER AND NULLS

3-ALL PROGRAMS will output all programs in the program library. They will be sent as a single file, no offset data will be included. The operator is then prompted for the data format.

0-ASCII: 7 DATA BITS, EVEN PARITY, 1 STOP BITS

1-EIA : 8 DATA BITS, ODD PARITY, 1 STOP BIT

Normal output is 0-ASCII

0-COMPUTER, NO LEADER OR NULLS

1-TTY, LEADER AND NULLS

16.1.10 0-CUSTOM MACRO

Quick Key #0 will run program number 9999 without having to switch the currently active program in memory. This feature allows the user to conveniently run this program.

Any program maybe defined as O9999 or a factory installed sample macro program may be loaded into memory. The factory example program can be loaded by typing TA,6 from the command mode. The example macro program is an example of a menu. This program can be modified and stored on disk for future use.

When the 0 key is pressed and program number 9999 is not found, the control will display the program library menu. The user may then choose to start a new program with a program number 9999, for a user-defined custom macro program.

See the PR command, section 8.0 COMMANDS, MAN-0131

EXAMPLE:

1. Program O1 is a production part. This is the currently active program in memory.
2. Program O9999 is a warm up program written by the programmer which is performed every morning.
   a. Press the space bar until the Quick Key menu is displayed.
   b. Press the 0 button to run the custom macro to warm machine up.
   c. Press AUTO to begin. (Program O9999 will now be run in auto.)
   d. After the warm up program is complete, press MANUAL to exit AUTO.
   e. Press AUTO to run production part. (Program O1 will now be run in auto.)

16.2 EDIT MENU

The Edit Menu is selected by pressing the space bar twice from ENTER NEXT COMMAND, and allows editing or viewing the currently active program. This menu functions the same as the PA command editor (see PA Command, section 8.0 COMMANDS, MAN-0131). When the Edit Menu is entered the following screen is displayed.

Figure 16-2: Edit Menu

16.2.1 EDIT MENU OPTIONS

The menu options displayed function in the following manner.

U-UP Press U to move the cursor up one line
D-DOWN Press D to move the cursor down one line
*F-FUNCTIONS Press F to enter the graphics menu (*See for CNC88HS graphics use, section 20.0 GRAPHICS, MAN-0131.)
C-CHANGE Press C to change the line that the cursor is on
I-INSERT Press I to insert a line after the line that the cursor is on
S-SEARCH Press S to search for data within the program. The cursor will move to the first line that the data is in
R-REPLACE Press R to search and replace data within the program
N-NUMBER Press N to renumber the program
O-COPY Press O to copy program blocks
ENTER-PAGE DOWN Press the ENTER key to move the cursor one page forward in the program
BACKSPACE-PAGE UP Press the BACKSPACE key to move the cursor one page backward in the program
DEL-DELETE Press the DEL key to delete the program line that the cursor is on
P-PROGRAM Press P to switch to another program
The following key commands NOT displayed on the menu are also available:
T-TOP Press T to move the cursor to the first line of the program
B-BOTTOM Press B to move the cursor to the last line of the program

16.3 FUNCTION MENU

The Function Menu is selected by pressing the space bar three times from ENTER NEXT COMMAND. The first twelve (12) lines display the currently active program. The bottom four (4) lines display the Function Menu options. These menu options are selected by number input. The operator may switch between the Function Menu, the Quick Keys Menu, and the Edit Menu by pressing the Space bar.

Selection of menu options may display additional menu options. The top line of the subsequent menus displays the previous menus selected. Press the MANUAL key at any menu to abort the process and return to the MAIN menu. The following pages explain the complete menu structure and the options available.

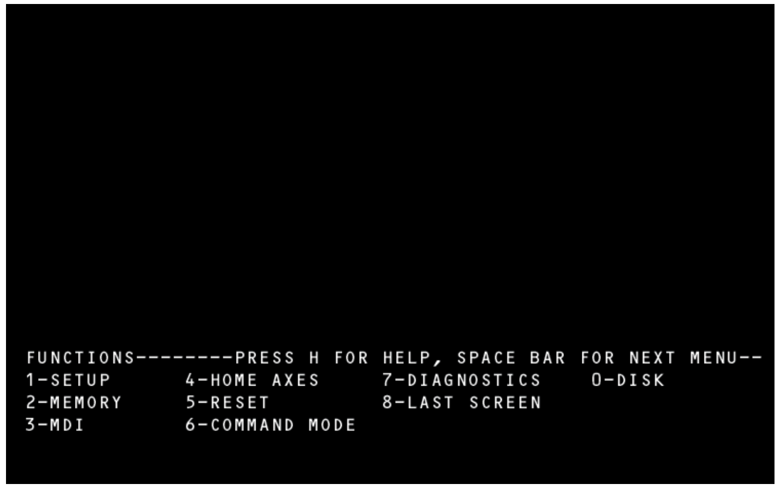


Figure 16-3: Function Menu

16.3.1 1- SETUP FUNCTION

To perform machine Setup functions select option 1 from the main menu. This function allows for the following options.

Select the option below for the indicated desired results:

1. **Tool** - This selection is used to set tooling information. It may be used to set tool length or diameter offsets. These offsets may be input or changed. The following options are available when Tool is selected:

    1. **Single** - This option is used to set a single tool. The options requiring input will affect the tool in the spindle. The following options are available when Single is selected:

        1. **Get Tool** - This option is used to change to a desired tool. Enter the tool number to change to. The machine will perform a tool change to the tool number.

        2. **Enter Diameter** - This option is used to input a tool diameter. The input value will be an absolute change to the tool table. Enter the diameter value. The entered value is placed into the tool table.

        3. **Store Length** - This option is used to input the tool length into the tool table. The machine will input the current Z axis location into the tool table. The control performs the SL command.

        4. **Modify Length** - This option is used to change a tool length offset. The offset will be incrementally changed. Enter the amount to change the tool offset. The machine will perform the SL command to change the offset.

5. **Jog Key** - This option allows the operator the ability to Jog the tool to the gauge point before setting the length. The machine enters the Jog mode. Press the manual key to return to the previous menu.

2. **Multiple** - This option should be selected when setting tool data for more than one tool. When this option is selected, the machine performs the UT command (See the Setting Tool Offsets, Using the UT command).

3. **Offset Table** - This option is used to view the tool table. The control performs the DT command.

4. **Turret Location** - This option is used to set the current tool turret location as number 1.

1. **Reset Current Tool Location As 1** - This selection performs the SETTO command.

2. **Abort** - This options aborts the procedure and returns to the main menu.

2. **Fixture** - This selection is used to set fixture offset information. The following options are available:

1. **Offset Table** - This option is used to view the fixture offset table. This selection performs the DF command.

2. **Read From Jog** - This option is used to set fixture offsets using the UT command. This selection initiates the UT command for Fixture Offset setting.

3. **Zero Axes** - This option is used to set the machine axes home positions. The current axis location is set as the axis home position.

1. Press Keys: X, Y, Z, A, B, (H FOR ALL) OR JOG - The JOG key may be pressed to enter the Jog mode. Press MANUAL to return to this option. The selection of an axis will set the current location as the axis home position.

16.3.2 2 - MEMORY FUNCTION

This selection from the main menu may be used to perform machine memory tasks. The following options may be selected to perform the desired operation:

1. **Run Program** - This option begins execution of the current program.

1. **Automatic** - This option begins program execution.

1. **Begin** - This selection begins program execution from the beginning.

2. **Select Options** - This option allows the operator to select the AU command options individually. Program execution will be performed according to the AU parameters selected.

2. **DNC** - This option begins DNC execution of the program.

1. **Begin** - This selection begins DNC program execution from the beginning.
2. **Select Options** - This option allows the operator to select the DNC command options individually. Program execution will be performed according to the DNC parameters selected.

3. **Summary** - This option is used to perform the SUM command. This command is used to debug the program in memory.

1. **Begin** - This option performs the SU command with all parameters = zero.
2. **Select Options** - This option allows the operator to select the SUM command options individually. Program execution will be performed according to the SUM parameters selected.

2. **Program Library** - This command performs program library maintenance. The PR command is performed when this option is selected.

3. **Input/Output** - This option is used for program transferring and verification.

1. **Input** - This option is used to transmit a file to the machine.

1. **Begin** - This option performs a TA,1 command.
2. **Options** - This option allows the operator to select the TA command options individually. The TA command is performed with the selected parameters.

2. **Output** - This option transmits a file from the machine to an external device.

1. **Begin** - This option performs a PU command.
2. **Options** - This option allows the operator to select the PU command options individually. The PU command is performed with the selected parameters.

3. **Verify** - This option is used to verify the tape that was punched. The command VT,1 is performed when this option is selected.

4. **Baud Rate** - This option is used to set the machine baud rate.

1. **Select Baud Rate** - Enter the selection number to select the machine baud rate.

4. **Learn Mode** - This option is used to enter the learn mode. Selection of this option performs the LE command.

5. **Clear** - This option is used to clear data from the machine memory.

   1. **Current Program** - This option is used to clear the current program from memory. The NE command is performed when this option is selected.

   2. **Offsets and All Memory** - This option is used to completely clear tool and fixture offsets and machine memory. The RI command is performed when this option is selected.

   3. **Display Free Memory** - This option is used to display the percentage of free memory in the machine. The ME command is performed when this option is selected.

6. **Settings** - This option is used to view or change parameter and backlash settings.

   1. **Backlash** - This option is used to view or change the machine backlash settings.

      1. **Display** - This option displays the current machine backlash settings. The BL command is performed when this option is selected.

      2. **Change** - This option is used to change the machine backlash settings. The axes and desired settings are input at the required prompts.

   2. **Parameters** - This option is used to view or change the machine parameters. The SETP command is performed when this option is selected.

16.3.3 3 - MDI FUNCTION

This function performs the MD command, putting the control in Manual Data Input mode. Pressing the MANUAL key will return to the main menu.

16.3.4 4 - HOME AXES FUNCTION

This function is used to send the machine to the home position.

1. **Return To Home** - This option is used to return the machine to the axes home position. The HO command is performed when this option is selected.
2. **Return For Power Off** - This option is used to return the machine to the cold start position for power off. The SETCS and HO commands are performed when this option is selected.

16.3.5 5 - RESET FUNCTION

This function is used to reset the machine's modal codes. The HO command is performed when this option is selected.

16.3.6 6- COMMAND MODE FUNCTION

This function is used to enter the command mode. The machine operates in the same format as when the Menu Mode is turned off. Press the MANUAL key to return to the menu mode.

16.3.7 7- DIAGNOSTICS FUNCTION

This function is used to perform probe and diagnostic tests.

A. Touch Probe - This option is used to test the TS20 or 27 touch probe. The touch probe test begins when this option is selected.

B. MP Probe - This option is used to test the MP probe. The MP test begins when this option is selected.

C. Controller - This option is used for maintenance diagnostics. This option should be used by trained maintenance personnel ONLY.

1. Continue - This option performs the DI command. This is for maintenance personnel ONLY.
2. Abort - This option returns to the main menu. This option is for non maintenance personnel.

16.3.8 8- LAST SCREEN FUNCTION

This function is used to display the previous screen. Warning or error messages that were cleared can be displayed with the use of this function. Only the last message is displayed.

16.3.9 9- COLD START FUNCTION

This function is used to perform the machine cold start procedure. This function should be the first function used after power on. After this function is used, the option is no longer displayed on the main menu. If the option is displayed it should be selected to perform the cold start.

A. Continue - This option should ONLY be used when the axes are aligned with the cold start indicators. If they are not, use option 2.

B. Align Axes - This option is selected with the JOG key. When the axes cold start indicators are not aligned, press the JOG key to align them. Press the MANUAL key to return to the cold start menu. After this option has been completed select option 1 to complete the cold start procedure.

17.0 PALLET CHANGER

17.1 MECHANICAL OVERVIEW

This section will describe the operation of the optional Pallet Changer. The pallet changer may be operated within the program or by use of the UT command. The pallets ride on rails with circular bearings. The pallet is locked in place on the table with a hydraulic clamp.

This clamp is released with program coding of a pallet change or through the utility menu function. The door operates with a hydraulic valve to open and close. The pallets are moved by a mechanical arm. The two pallets are identified as pallet A and pallet B. The forward pallet is pallet A. The rear pallet is pallet B.

There are three pallet positions, loaded, stored, and working. The loaded position is when the pallet is locked onto the table with the hydraulic brake. The stored position is when the pallet is ready to be loaded onto the table. This position locks into place with a mechanical release. The working position is when the pallet is released from the stored position. The pallet must be moved to this position manually. Push the release lever down to pull the table to the working position. This is the outermost position that the pallet can be moved to. This position is the most accessible to the operator for part changes.

There are three M Functions that may be used to operate the pallet changer. These functions can be used to exchange pallets or utilize only one pallet.

> NOTE
> The machine parameter “DO YOU HAVE A PALLET CHANGER?” MUST be answered YES for the pallet to operate.

17.2 G17.1 - G17.2 A/B WORD SWAP

The G17.1 word swap activates B-axis command substitution for the A-axis command. This allows the use of the A-axis rotary moves in one program for use with the rotary heads on both pallets. If the program calls for an A-axis move and the rotary device is connected to the B-axis controller, the G17.1 code will swap the A word for a B word. Existing programs written for the dual 4th axis setups that contain the A and B words are allowed. The G17.1 will automatically swap the B words to A words. The G17.2 word cancels A/B axis command swap mode.

17.3 M31 EXCHANGE PALLETS

M31 performs a pallet exchange. The pallet changer will store the current pallet on the machining table and load the other pallet onto the machining table. No other machine movements will be made. This is the only code allowed on the program line.

> NOTE
> All fixture and machine offsets MUST be canceled prior to attempting a pallet change.

17.4 M32 STORE PALLET B AND LOAD PALLET A

When Pallet B is on the table, M32.1 will STORE Pallet B (outside machining area) and pallet arm will move to Pallet A. The machine will be placed in the WAITING state and the pallet door will remain open until the START button is pressed. When the START button is pressed Pallet A will be returned to the table inside the machining area (LOADED).

When Pallet A is on the table, M32.1 will STORE Pallet A. The machine will be placed in the WAITING state and the pallet door will remain open until the START button is pressed.When the START button is pressed Pallet A will be returned to the table inside the machining area (LOADED).

> NOTE
> All fixture and machine offsets MUST be canceled prior to attempting a pallet change.

17.5 M32.1 STORE PALLET B & LOAD PALLET A & VERIFY PALLET A HAS BEEN LOADED

When Pallet B is on the table, M32.1 will STORE Pallet B and LOAD Pallet A. If Pallet A is at LOAD position (on machining table), M32.1 will verify Pallet A is at LOAD (on machining table). No movement will occur.

17.6 M33 STORE PALLET A AND LOAD PALLET B

When Pallet A is on the table, M33 will STORE Pallet A (outside the machining area) and the pallet arm will move to Pallet B. The machine will be placed in the WAITING state and the pallet door will remain open until the START button is pressed. When the START button is pressed, Pallet B will be returned to the table inside the machining area (LOADED).

When Pallet B is on the table, M33 will STORE Pallet B.The machine will be placed in the WAITING state and the pallet door will remain open until the START button is pressed. When the START button is pressed, Pallet B will be returned to the table inside the machining area (LOADED).

> NOTE
> All fixture and machine offsets MUST be canceled prior to attempting a pallet change.

17.7 M33.1 STORE PALLET A & LOAD PALLET B & VERIFY PALLET B HAS BEEN LOADED

When Pallet A is on the table, M33.1 will STORE Pallet A and LOAD Pallet B.

If Pallet B is at LOAD position (on machining table), M33.1 will verify Pallet B is at LOAD (on machining table). No movement will occur.

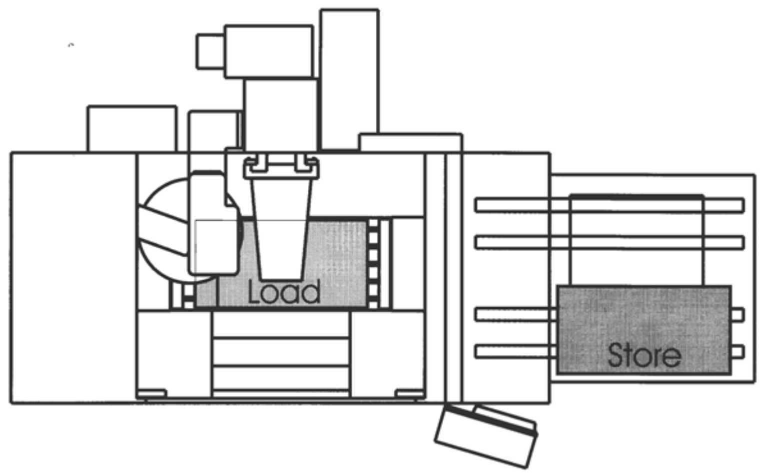


Figure 17-1: Pallet B at LOAD Position

NOTE
The table move to the pallet position may be stopped with the SLIDE HOLD button. The pallet movement to and from the stored position may also be stopped with the SLIDE HOLD button.

NOTE
The pallet MUST be in the stored position to change pallets. When in the working position, the message "RETURN PALLET TO THE STORED POSITION" will appear if a change is attempted.

NOTE
The air and hydraulics are turned off between M32 and M33 pallet changes, and after a STORE PALLET in the pallet utilities.

NOTE
When a pallet is placed in the service position during a pallet change, the machine tool enters the WAITING state.

17.8 PALLET A ROTARY TABLE & PALLET B ROTARY TABLE OVERRIDE POTENTIOMETER

Pallet changer machines that are equipped with dual rotary tables have two potentiometers located on the sheet metal, one to the left and one to the right of the pallet door. They are Pallet A rotary table potentiometer for the rotary table on Pallet A, and Pallet B rotary table potentiometer for the rotary table on Pallet B. This is assuming the rotary table on the Pallet A is A axis, and Pallet B rotary table is B axis.

Potentiometers will become active after Cold Starting the machine and then enabling the M48.2, Pallet A rotary table override pot, or M48.3, Pallet B rotary table override pot. These M codes may be used in a program or in MDI mode. The intention of the potentiometers are to allow the operator to turn the rotary table for removing or installing the workpiece.

The rotary tables will return automatically to the original position when the M49.2, Pallet A Rotary Table Override Potentiometer Disable, or M49.3, Pallet B Rotary Table Override Potentiometer Disable, is programmed. This may also be done in a program or in MDI mode. The movement of the rotary table will be slow when the potentiometer is turned. When the axis is returned with disable codes (M49.2 or M49.3) the move will be automatic and slow in speed. This rate of speed is NOT adjustable. The corresponding pot disable M code will lock out the potentiometer and then bring the rotary tables to the last position. This will ensure that the rotary tables are in the original position before returning to the work area.

17.9 M48.2 PALLET A ROTARY TABLE OVERRIDE POTENTIOMETER ENABLE

M48.2 enables the Pallet A rotary table axis override pot while Pallet A is stored.

17.10 M49.2 PALLET A ROTARY TABLE OVERRIDE POTENTIOMETER DISABLE

M49.2 disables the Pallet A rotary table axis override pot while Pallet A is stored.

17.11 M48.3 PALLET B ROTARY TABLE OVERRIDE POTENTIOMETER ENABLE

M48.3 enables the Pallet B rotary table axis override pot while Pallet B is stored.

17.12 M49.3 PALLET B ROTARY TABLE OVERRIDE POTENTIOMETER DISABLE

M49.3 disables the Pallet B rotary table axis override pot while Pallet B is stored.

EXAMPLE:

N3 E0 X0Y0 A0
N4 M33.1 (STORE PALLET A, LOAD PALLET B IN MACHINING AREA
N5 M48.2 (PALLET A ROTARY OVERRIDE POT ENABLE
...
N506 M49.2 (PALLET A ROTARY OVERRIDE POT DISABLE
N507 E0 X0 Y0 A0
N508 M33.1 (STORE PALLET B ROTARY OVERRIDE POT ENABLE
...
N1017 M49.3 (PALLET B ROTARY OVERRIDE POT DISABLE

17.13 REMOTE MACHINE CONTROL

There are two remote machine control panels mounted by the pallet work area. These controls provide the operator with the START, SLIDE HOLD, and EMERGENCY STOP buttons for machine control. They are located on the side of the machine enclosure at both pallet work areas.

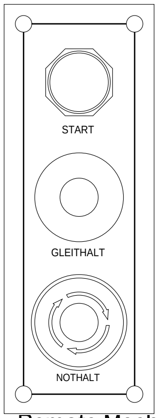


Figure 17-2: Remote Machine Control

17.14 UTILITY MENU

The pallets may also be operated using the UT command. When the UT command is entered, the following menu appears:

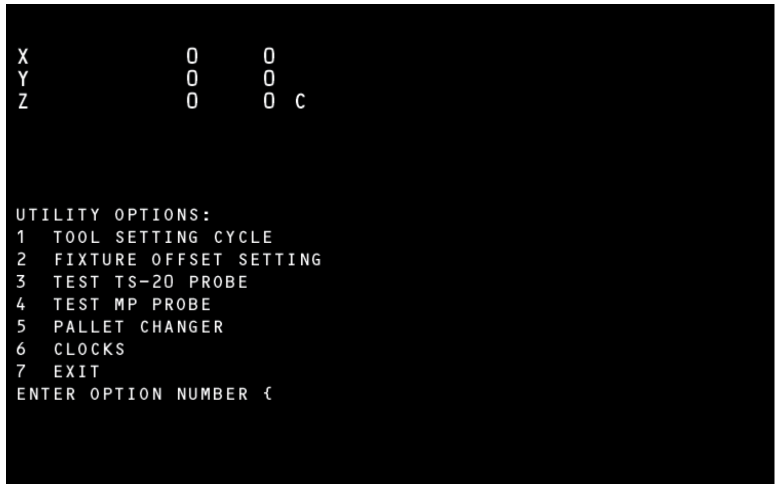


Figure 17-3: Utility Menu

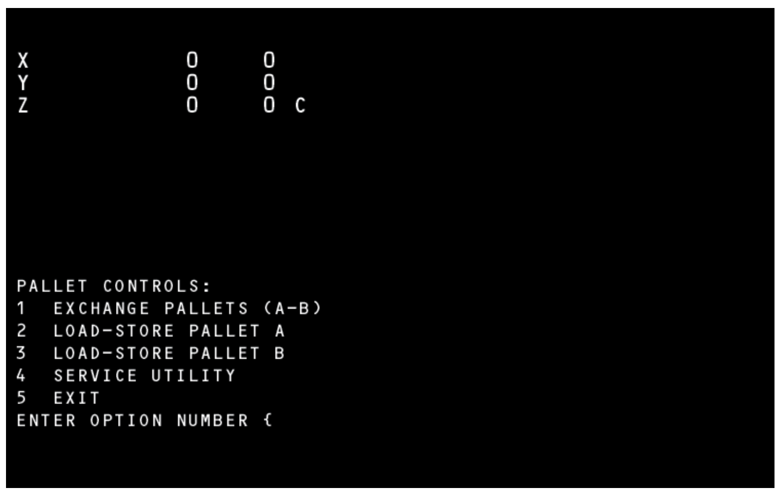


Figure 17-4: Pallet Controls Menu

17.14.1 OPTION 1

When option 1 is selected, the following sequence will occur:

1. The door opens.
2. The machine enters the WAITING state, and the message “PRESS START TO MOVE PALLET” appears. The X, Y, M48, F150, and G1 will also appear if a table move is necessary.
    a. When the START button is pressed, the machine will move to position for the pallet change. When in position the machine will enter the WAITING state.

3. When the START button is pressed, the hydraulic clamp is released and the pallet will move to the stored position. The table then moves into position to receive the other pallet. The pallet is moved into the loaded position on the table. The hydraulic clamp is engaged and the door is closed.
4. The PALLET CONTROLS menu is displayed.

17.14.2 OPTION 2

If Pallet B is on the table, this option performs the same as option 1. When option 2 is selected and Pallet A is on the table, the following will occur:

1. The door opens.
2. The machine enters the WAITING state, and the message “PRESS START TO MOVE PALLET” appears. The X, Y, M48, F150, and G1 will also appear if a table move is necessary.
    a. When the START button is pressed the machine will move to position for the pallet change. When in position the machine will enter the WAITING state.
3. When the START button is pressed the hydraulic clamp is released and the pallet will move to the stored position. The machine enters the WAITING state for the operator to change parts.
4. When the START button is pressed the pallet is returned to the loaded position and the door closes.
5. The PALLET CONTROLS menu is displayed.

17.14.3 OPTION 3

If Pallet A is on the table, this option performs the same as option 1. When option 2 is selected and Pallet B is on the table, the following sequence will occur:

1. The door opens.
2. The machine enters the WAITING state, and the message, “PRESS START TO MOVE PALLET” appears. The X, Y, M48, F150, and G1 will also appear if a table move is necessary.
    a. When the START button is pressed the machine will move to position for the pallet change. When in position the machine will enter the WAITING state.
3. When the START button is pressed the hydraulic clamp is released and the pallet will move to the stored position. The machine enters the WAITING state for the operator to change parts.
4. When the START button is pressed the pallet is returned to the loaded position.
5. The PALLET CONTROLS menu is displayed.

17.14.4 OPTION 4

When this option is selected the following menu appears:

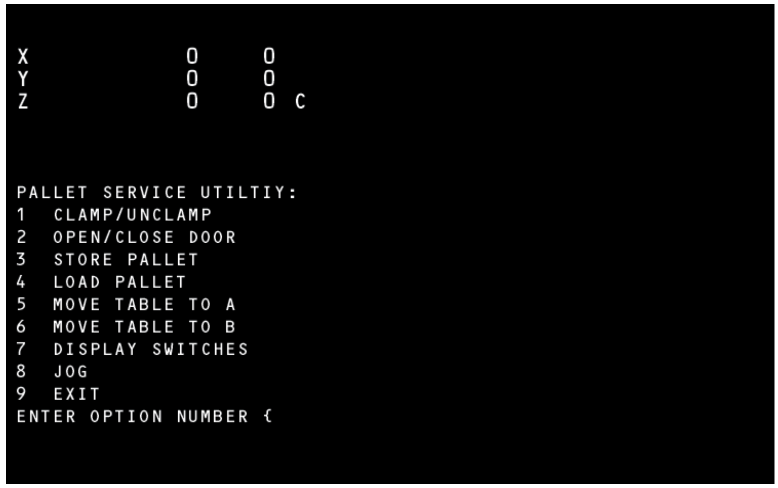


Figure 17-5: Pallet Service Utility Menu

OPTION 1: This option toggles the hydraulic clamp on and off.

OPTION 2: This option toggles the door open and closed.

OPTION 3: The table MUST be in position to receive a pallet the following occurs:

a. The door opens if not already open.
b. The machine enters the WAITING state and displays the message “PRESS START TO MOVE PALLET”.
c. When the START button is pressed, the pallet moves to the stored position. While the pallet moves to this position, the message “STORING PALLET...” appears. If the pallet is already stored, the message “STORING PALLET...PALLET IN STORAGE” appears and the control is returned to the command mode.

OPTION 4: This option is only used when the table is empty. The table MUST be in position to receive the pallet. See options 5 and 6. When the table is in position, the following occurs:

a. The door opens if not already open.
b. The machine enters the WAITING state and displays the message “PRESS START TO MOVE PALLET”
c. When the START button is pressed, the pallet moves to the loaded position on the table. While moving to the loaded position the message “LOADING PALLET...” appears. If the table is not in the change position the message “LOADING PALLET...RAILS NOT ALIGNED” appears and the control returns to the command mode.

OPTION 5: This option moves the table to the change position for Pallet A. The following occurs:

a. The door opens.
b. The machine enters the WAITING state and displays the message "PRESS START TO MOVE PALLET". The X, Y, M48, F150, and G1 will also appear if a table move is necessary.
c. When the START button is pressed, the table moves to the change position for Pallet A.

OPTION 6: This option moves the table to the change position for Pallet B. The following occurs:

a. The door opens.
b. The machine enters the WAITING state and displays the message "PRESS START TO MOVE PALLET". The X, Y, M48, F150, and G1 will also appear if a table move is necessary.
c. When the START button is pressed, the table moves to the change position for Pallet B.

OPTION 7: This option displays the maintenance switches for the pallet system. This option is to be used by trained maintenance personnel.

OPTION 8: This option allows the operator to enter the JOG mode. The message "PRESS JOG TO CONTINUE OR MANUAL TO EXIT" appears. Press the JOG button to enter the Jog mode. When in the Jog mode, press MANUAL to return to the pallet service utility menu. If the MANUAL button is pressed when this message appears, the control returns to the command mode.

OPTION 9: This option is used to return to the PALLET CONTROLS menu.

18.0 MACROS

18.1 MACROS AND PARAMETRIC PROGRAMMING

18.1.1 MACROS

Macros give the programmer the ability to perform arithmetic and comparison functions within a CNC program. Values for variables (V1- V100) can be entered from input statements or passed to the macro from R variables or from the fixture, tool data, and tool time tables. Values from macro statements can be passed to the CNC program by using parametric variables (R0-R9). Macros may be used for probe functions and for part outlines where the part can be defined by an arithmetic formula.

18.1.2 PARAMETRIC PROGRAMMING

Parametric programming gives the operator the ability to use a parametric R variable to represent the value of any program coding word. The variables include: R0 and R1 - R9. Parametric programming is also used to transfer data from macro variables V1-V100 to the CNC program for machine motion, feed and speed.

EXAMPLE:

N13 R1 + 10. R2 + 5.   (R1 AND R2 ARE ASSIGNED VALUES
N14 R3 + 7.5 R4 + 5.5 R5 + 0   (R3, R4 AND R5 ARE ASSIGNED   VALUES
N15 X - R1 Y + R2. G1 F34.   (THIS LINE READS X10. Y5. G1 F34.
N16 X + R3. I - R4 J - R5 G3   (THIS LINE READS X7.5 I5.5 J0 G3

Parametric variables are used when programming words require change during the execution of a program or need to be changed at different runs. One typical use of these variables would be in the use of a subroutine or subprogram to cut a pocket. The Z words in the routine may use a variable to cut many different levels. The L word for the subroutine call may use a variable to change the number of times that the sub is called. The R variable used for the L word must indicate the subroutine number and number of repetitions (see the example below). The X, Y, I, and J of a circular move can all be changed by parametric programming. All program words can use a parametric variable. Parametric programming is also used when programmer is considering a generic program for a family of parts.

> NOTE
> When variables are used as values, a positive or negative symbol must be used between the two variables, R9+R2, R8-R2. This is also true when using radius designation for circular moves or for the minimum clearance plane description with fixed cycles.

18.1.3 R VARIABLE DEFINITION

R variables are defined by coding the R word, + or - symbol and a value.

EXAMPLE:

N12 R8+50.0 This is variable R8 defined as 50.0
N13 F+R8 The R8 transfers a value of 50.0 for the feed rate

> **NOTE**
>
> Variables must be defined in the beginning of the program or just before they are used in the program. Variables are modal and retain their values after the termination of a program, after an HO, and after exiting from MDI. R variables do not have table storage like macro V variables. The programmer should always specify a value for an R variable; otherwise, the last programmed value will be used and will result in unpredictable machining.

18.1.4 EXAMPLE PROGRAM FOR PARAMETRIC PROGRAMMING

The parametric R variables are defined in the main program, on lines 32 and 37, and in the subroutine on line 19 of the program below. The R variables are modal; they remain effective until the same R variable is redefined. Note that on line 21 of the program below, the R1 variable is changed but the R2 variable remains in effect.

```
N01 L100(SUB TO CUT SLOT
N02 (R1=Z, R2=SUB & REPS, R3=RADIUS OF SLOT, R4=DIA OF SLOT
N03 X-1. Z-R1 G1
N04 X-R3 G41
N05 L+R2
N06 X+R3 G40
N07 X1.
N08 L200(SUB FOR SLOT
N09 X+R3 Y-R3 I+R3 G3
N10 X1.
N11 Y+R4 J+R3 G3
N12 X-1.
N13 X-R3 Y-R3 J-R3 G3
N14 L300(CUT SLOT
N15 Z0.01 G1
N16 R1+0.1 R2+201.
N17 G91
N18 L102
N19 R1+0.05
N20 L101
N21 R1+0.01 R2+202.
N22 L101
N23 G0 G49 G90 Z0.1
N24 M17
N25 M30
N26 (
N27 (MAIN PROGRAM
N28 G0 G90 S8000 M3 E1 X1.5 Y-0.876
N29 (R1=Z, R2=SUB & REPS, R3=RADIUS OF SLOT, R4=DIA OF SLOT
N30 H1 Z0.1 M7
```

N31 Z0.01 G1 F30.
N32 R3+0.19 R4+0.38
N33 L301
N34 X4. Y-0.876
N35 L301
N36 X4. Y-3.45
N37 R3+0.255 R4+0.51
N38 L301
N39 X1.5 Y-3.45

18.1.5 MACROS

Each macro line is identified by using the # symbol at the beginning of a line in the program. Macros can reside in a main program, subroutine, or in a subprogram. Macros may be used during AUTO, MDI (Manual Data Input), DNC, and at the command line. The maximum number of characters per macro line is 63.

Parametric R variables are used to transfer macro data to the CNC program.

EXAMPLE:

N1 # V1 = V1 + 2.
N2 # V2 = V2 - 1.
N3 # R1 = V1
N4 # R2 = V2
N5 G0 G90 S8000 M3 E1 X+R1 Y-R2

Machine program code **cannot** be coded in a macro line.

EXAMPLE:

*N1 G0 G90 # V1=INT(V5 * 10000)/10000 This line is incorrect*
*N1 # V1=INT(V5 * 10000)/10000 This line is correct*
N2 G0 G90 This line is correct

EXAMPLE MACRO PROGRAM FOR A RECTANGLE WITH PRINT AND INPUT STATEMENTS:

```
0600 (SUB FOR RECTANGULAR POCKET FINISH
#CLEAR
#PRINT “ENTER THE POCKET LENGTH: ”,
#INPUT V1   Keyboard entry is transferred to V1
#PRINT “ENTER THE POCKET WIDTH: ”,
#INPUT V2   Keyboard entry is transferred to V2
#R3=V1/2   R3 equals V1 divided by 2
#R4=V2/2   R4 equals V2 divided by 2
#R2=V1   Transfers the V1 value to R2 parameter
#R5=V2   Transfers the V2 value to R5 parameter
G0G90Z.1
X0Y0
D1
Z-.250G1F20
G91 G41 Y-R4
X+R3
Y+R5
X-R2
Y-R5
X+R3
Y+R4 G40 G90   Cancel CRC
G0G90Z.1  Safety Z move
M99   End of sub program

O700 (SUB FOR RECTANGULAR POCKET FINISH WITH CORNER RADIUS

#CLEAR
#PRINT “ENTER THE POCKET LENGTH: ”,
#INPUT V1   Keyboard entry is transferred to V1 (X VALUE)
#PRINT “ENTER THE POCKET WIDTH: ”,
#INPUT V2   Keyboard entry is transferred to V2 (Y VALUE)
#PRINT “ENTER THE CORNER RADIUS:”
#INPUT V3   Keyboard entry is transferred to V3
#V4=V3+V3   Diameter
#R2=V1-V4   Transfers the V1-V4 value to R2 parameter (X
#R3=V1/2   R3 equals V1 divided by 2 (X VALUE)
```

#R4=V2/2   R4 equals V2 divided by 2 (Y VALUE)
#R5=V2-V4   Transfers the V2-V4 value to R5 parameter (Y
#R6=V3   Radius
#R7=(V2/2) V3R7 equals V2 divided by 2, minus the radius value (Y
#R8=(V1/2) V3R3 equals V1 divided by 2, minus the radius value (X
G0G90Z.1   H1Safety Z move
X0Y0   Locate to the center of pocket
D1   Diameter of tool must be in tool table
Z-.250G1F20.   Feed down into pocket
G91 G41 Y-R4CRC climb cut
X+R8   Parametric transfer to axis movement
X+R6 Y+R6 R0+R6 G3
Y+R5
X-R6 Y+R6 R0+R6 G3
X-R2   Macro calculations determine the X Y moves here
X-R6 Y-R6 R0+R6 G3
Y-R5
X+R6 Y-R6 R0+R6 G3
X+R8
Y+R4 G40 G90   Cancel CRC
G0G90Z.1   Safety Z move
M99   End of subprogram

18.2 MATHEMATICAL FUNCTIONS

18.2.1 MATHEMATICAL FUNCTION MACROS

Macros operate with the use of mathematical functions, commands and variables. The variables are used in conjunction with the functions to perform calculations. The characters below describe the mathematical function capabilities. Older controls are required to type the Alpha Keys for the characters not on the keyboard.

Table 18-1: Mathmatical Function Macros

CHARACTER	DESCRIPTION	ALPHA KEYS
	Blank	
+	Plus, Positive, Addition	
-	Minus, Negative, Subtraction	
*	Multiply, Multiplication	
/	Divide, Division	
ABS	Absolute value	
ATN	Arc tangent	
COS	Cosine function	
INT	Integer value	
RND	Rounding function	
SGN	Return the sign	
SIN	Sine function	
SQR	Square root	
(	Left Parenthesis	
)	Right Parenthesis	
=	Equal	EQ
<	Less Than	LT
>	Greater Than	GT
<=	Less Than or Equal to	LE
>=	Greater Than or Equal to	GE
<>	Not Equal to	NE
‘	Remark in macro line	

18.2.2 ORDER OF CALCULATION

Arithmetic operations are performed in standard mathematical order:

Parentheses ( )

Functions (SIN,COS ...)

- (negation)

* (multiplication)and / (division)

+ (addition)and - (subtraction).

AND, OR, NOT

Parentheses can be used to establish order of arithmetic operations, overriding standard mathematical order.

18.2.3 DECIMALS

Numbers without decimals will be considered integers. This is true only for macro lines.

18.2.4 EXPONENTIAL FORM

Exponential form is not allowed. For 1.23^5 use 123000.

18.2.5 COMMENTS

Comments are very important part of Macro programming. They explain the program to others, and remind the programmer why they wrote it the way they did. Comments on a macro line begin with an apostrophe (‘), and can extend to a total line length of 63 characters.

N3 # V100=1.2345 ‘V100 IS THE LENGTH OF THE PART 2

18.2.6 CALCULATIONS

The mathematical operations of addition, subtraction, multiplication and division are represented by + , - , *(asterisk) , and / (slash), respectively.

EXAMPLE: *N11 # V10=3/V12*

18.3 VARIABLES

18.3.1 ARRAYS

Variable arrays can now be used; that is, the subscript of a variable can now be a variable or arithmetic expression. The subscript variables are D, FX, FY, FZ, H, PX, PY, PZ, PA, PB, R, and V. The subscript for a variable must not exceed the allowed range for the variable. For instance, the subscript for a V variable must be 1 through 99 and the subscript for a FX variable must be 1 through 48.

EXAMPLE: *If the variables V1=5 and V2=1 then the following are equivalent:*

#V5=1
#V(5)=1
#V(V1)=1
#V(V1+V2-1)=1
#V(V1/V2)=1
#V(V(SQR(V2))=1

18.3.2 AX, AY, AZ, AA, AB, AC: AXIS POSITION VARIABLES

Axis position variables are used to read the current location of the desired axis.

EXAMPLE: *N79 # IF AZ >12.050 THEN GOTO :GETNEXT*

In the above example, if the Z axis is at 12.050 inches by the time this line is executed then the control will jump to the line in the program with the label :GETNEXT.

EXAMPLE:

N99 E2 X0 Y0
N100 Z-30 G31 F50.
N101 G91 Z.05
N102 G90
N103 Z-30. G31 F1. The probe will stop the Z axis when it touches
N104 #IF AZ > 12.050 THEN GOTO :GETNEXT

18.3.3 CP: CLOCK FROM POWER ON

This variable will return the accumulated time stored since power on.

18.3.4 CR: CLOCK FOR ALL RUN TIME

This variable will return the accumulated time for all of the time while the control was in auto mode.

18.3.5 CC: CLOCK FOR CURRENT PART

This variable will return the accumulated time of the program currently being executed. The SETP page must have the TIMERS ON for the accumulated time to register.

18.3.6 D1 - D99: TOOL DIAMETER/RADIUS VARIABLES

These variables are used to read the current value of any diameter/radius from the tool table. It can also be used to establish or write a value in the table. The current value for any diameter can be changed by placing the variable on the left side of the equal sign.

EXAMPLE: Reading Diameters

N20 # IF D5 >.505 THEN GOTO :END
N...
N349 # :END
N350 M0
N350 #PRINT " TOOL DIAMETER #5 SHOULD BE NO LARGER THAN .505",

EXAMPLE: Writing Diameters

N89 # D2 = D2 + .01
This line will read the current value of diameter offset #2, then write a new diameter for #2, .01 larger, into the tool table.

18.3.7 FX1-FX48, FY1-FY48, FZ1-FZ48, FA1-FA48, FB1-FB48: FIXTURE OFFSET VARIABLES

These variables are used to read the current value of each fixture offset. It can also be used to establish or write a value in the table. The current value of any fixture offset can be changed by placing the variable on the left side of the equal sign.

FY3   Read/write the Y of fixture three
FZ34   Read/Write the Z of fixture thirty four

EXAMPLE: Reading

N15 G91 G10 L02 X2. P1
N16 # IF FX1 >30. THEN GOTO :SUB3   Read the X value of fixture one
N...
N24 # :SUB3.
N25 G91 G10 L02 X-2. P1
N26 E0 X0 Y-1.
N27 L315

EXAMPLE: WRITING

N63 # FX1 = FX1 - 2.

18.3.8 H1 - H99: TOOL LENGTH OFFSET VARIABLES

These variables are used to read the current value of any tool length offset from the tool table. It can also be used to establish or write a value in the table. The current value for any tool length offset can be changed by placing the variable on the left side of the equal sign.

EXAMPLE: READING

N34 # IF H16 <-10.67 THEN GOTO :NOTOOL
N35 # IF H16 = 0 THEN GOTO :STOPIT
N...
N144 # :NOTOOL
N145 #R6=V6+1
N146 M6 T+R6

EXAMPLE: WRITING

N654 # H33 = H33 - 16.

18.3.9 I, G, O, HO, TL,VF: MACRO VARIABLES

The TL and VF macros are single variables returning the index of the tool offset table, and index of the fixture offset table. The I-, O-, HO- and G macros are arrays, returning inputs, manipulating spare outputs, relative user selected home positions, and the current value of the G codes. All inputs and outputs are available from the 1040-2A card.

Fadal reserves the right to use and change any input and output pins at any time.

EXAMPLE:

N10#PRINT I(3)
N20#O(2) = 1

EXAMPLE:

LAYOUT OF I MACRO:

The inputs 1 – 6 are user inputs available from the J2 connector on the 1040-2A card. Pin 1 on the J2 connector is given by a little arrow visible on top of the connector.

I(1)	Input 1	J2 pin 17	I(2)	Input 2	J2 pin 18
I(3)	Input 3	J2 pin 19	I(4)	Input 4	J2 pin 20
I(5)	Input 5	J2 pin 21	I(6)	Input 6	J2 pin 22
I(7)	Renishaw Probe		I(8)	Scale Error	
I(9)	Ext. Slide Hold		I(10)	Slide Home	
I(11)	Spindle Orient		I(12)	Turret return	
I(13)	Slide external		I(14)	Orient Arm Home	
I(15)	Low range out		I(16)	Indexer stop sw.	
I(17)	Indexer arm sw.		I(18)	ATC fault	
I(19)	Drawbar down		I(20)	High range out	
I(21)	Oiler		I(22)	Arm at Table	J2 pin 9
I(23)	Arm at storage	J2 pin 10	I(24)	Y Aligned	J2 pin 11
I(25)	Pallet at A	J2 pin 12	I(26)	Pallet at B	J2 pin 13
I(27)	Door Open	J2 pin 14	I(28)	Door Closed	J2 pin 15
I(29)	Inhibit	J2 pin 16			

The input values contained in the I-array are either 0 (the input is low/on) or 1 (the input is high/off).

EXAMPLE: LAYOUT OF G MACRO:

G(l) (Group A G-codes) current interpolation type.

Contains G-code plus one as follows:

1 = (G0) point to point.

2 = (G1) linear.

3 = (G2) circular clockwise (CW).

4 = (G3) circular counter clockwise (CCW).

G(2) (Group B G-codes) current plane for circular algorithms.

G(3) (Group C G-codes) current cutter radius compensation mode.

G(4) (Group D G-codes) current Canned Cycle (CC) mode.

G(5) (Group E G-codes) current absolute/incremental mode

G(6) (Group F G-codes) current mode for return in Canned Cycle.

G(7) current location of spindle with respect to starting point (reference plane)

0 = spindle at 0 (initial plane)

1 = spindle at "R1'

EXAMPLE: LAYOUT OF O MACRO

O(1)	Output 1	J2 pin 6
O(2)	Output 2	J2 pin 7
O(3)	Output 3	J2 pin 8

The output is updated when a value is assigned to one of the slots in O-array. If a zero (0) is assigned to O(1), then the output on J2 pin 6 is pulled low. Any other value than zero assigned to the output will cause the output to float.

EXAMPLE: LAYOUT OF HO MACRO:

The HO-array contains values that represent the location of user selected home relative to absolute position for all axes.

HO(1)	X – axis	HO(6)	W - axis
HO(2)	Y – axis	HO(7)	A - axis
HO(3)	Z – axis	HO(8)	B - axis
HO(4)	U – axis	HO(9)	C - axis

HO(5)V - axis

18.3.10 PX1 - PB1, PX2 - PB2, PX3 - PB3: PROBE TOUCH POINT VARIABLES

These variables are used to read the current value of the touch points. PX1 would read the current X axis value of touch point P1. PY1 would return the value for the Y axis location of touch point P1 and PZ1 returns the Z axis value. The values for ONLY three touch points are available. The values for the X, Y, Z, A, and B axis may be read.

EXAMPLE:

N197 #V1=1
N198 #R8=V1
N199 G1 G31 X20. Y25. F25. P+R1
N200 # PRINT “POINT #”,V1," IS X",PX1," Y",PY1

After touching a point with the probe, the position (P1) is now in memory and each axis position can be used for any calculation or can be output through the RS-232 port with the print command.

The printed line will read as follows:

POINT #1 IS X18.5 Y17.65

18.3.11 R0-R9: PARAMETRIC VARIABLES

These variables are used to pass data from the macro to the CNC program lines. Parametric parameters R0-R9 are also used for fixed subs and fixed cycles (see Parametric Programming).

EXAMPLE:

N15 # R2 = V56
N16 Z-R2 G1 F20.
N17 L9201 R0+.1 R1+0 R2+.25 R3+0 R4+1 Z-R9 F+R8 (ENGRAVING

18.3.12 TN: TOOL NUMBER

This variable will return the numerical value of the current tool in the spindle. This number is determined by the last M6 T# executed from the program.

EXAMPLE:

N32 #V1=TN This would result in V1 being equal to the current tool number

18.3.13 TT: TOOL TIME

This variable will return the time entered in the time table for life expectancy of a specified offset.

> NOTE
> The TT and TU macro are active when the SETP parameters for tool time are enabled (On).One of the options (2-5) must be selected.

EXAMPLE:

N12 #V1=TT3 This will result in V1 being equal to the tool time entered in the time table

18.3.14 TU: TOOL USED

TU variables accumulate the total amount of time that a specific tool has been in use. For example, if tool 5 is used for ten minutes, then TU5 = 10. If tool 5 is used for another seven minutes, then TU5=17. TU variables are retained between part runs and when powering down the machine.

After a tool change, declare a tool length offset (H1 - H99). The current tool length offset applied determines which TU variable will be used. For example, if H7 is in effect, then TU7 will be active. The TU variable increases whenever the control is in the interpolation mode only.

> NOTE
> The TT and TU macro are active when the SETP parameters for tool time are enabled (On) by choosing one of the options (see SETP for tool life management options).

EXAMPLE:

N31 #IF TU3=3 THEN GOTO :CHANGETOOL

———————

N300 #:CHANGETOOL

This will cause a jump to the label :CHANGETOOL when the time is equal to 3. Using the option 2 of tool life management in the SETP page allows the user to program the tool time displayed in the tool time table and the TU to read the used time in the tool time table. The combination of the TT and the TU macro allows the user to customize a macro to include a specialize tool life management application.

18.3.15 V1-V100: MACRO VARIABLES

These variables are used in the macro lines.

EXAMPLE:

N88 # V1 = V2 + V3

18.4 FUNCTIONS

18.4.1 ABS

ABS will return the absolute value of a number.

EXAMPLE:

V2 = ABS (V1)
If V1 is equal to -2.3 then V2 would be 2.3, and if V1 is equal to 2.3, then V2 would be 2.3. The absolute value of a number is always the positive value of the number.

18.4.2 ATN

ATN will return the Arc tangent of a number. See the SET command for degrees or radians.

EXAMPLE:

V5 = ATN (V6/V7)
If V6 is the opposite side of a right angle and V7 is the adjacent side of the right angle, then V5 would be equal to the angle between the sides.
To calculate mathematical function Tangent of an angle use the following formula: TANGENT(angle) = SIN(angle) / COS(angle)

18.4.3 COS

COS will return the Cosine of an angle. See the SET command for degrees or radians.

EXAMPLE:

V56 = COS (V34 + V72)
If V34 and V72 represent angles, V56 would then be equal to the cosine of the sum of the two angles.
To calculate mathematical function Inverse Cosine of an angle use the following formula:

ARCCOS(angle) = 1570796-ATN(angle / SQR(1-angle*angle)

18.4.4 INT

INT will return the integer of a number.

EXAMPLE:

V100 = INT(V23)
If V23 = 12.345 then V100 = 12.
If V23 = -12.345 then V100 = -12.
If V23 = 12.513 then V100 = 12.
If V23 = -12.513 then V100 = -12.

The integer value uses only the whole number portion of the number.

18.4.5 RND

RND will return a rounded value of a number. The number of places to round to is set using the #SET RND# command. To set the rounding to four places enter the command below.

EXAMPLE:

N5 #SET RND4 This line sets the number of places to round to
N6 # V1 = RND(V1) This rounds the V1 value to 4 places

> NOTE
> When higher accuracy is desired, the rounding may be calculated. This eliminates the need to use the #SET RND# command. To round off numbers to the fifth place use this equation: V1 = INT(V5 * 10000)/10000. The value of V5 will be rounded to the fifth place and V1 will be used for the rounded number. This example only affects the individual line.

18.4.6 SGN

SGN is used to determine the sign of a value.

EXAMPLE:

V50 = SGN (V77 - V78)
If (V77 - V78) is a positive value, then V50 is +1.
If (V77 - V78) is zero, then V50 is +1.
If (V77 - V78) is a negative value, then V50 is -1.

18.4.7 SIN

SIN will return the sine of an angle. See the macro SET command for selecting degrees or radians.

EXAMPLE:

V56 = SIN (V34 + V72)
If V34 and V72 represent angles, V56 would then be equal to the sine of the sum of the two angles.
To calculate mathematical function Inverse Sine of an angle use the following formula:
ARCSIN(angle) = ATN(angle) / SQR(1-angle*angle)

18.4.8 SIN/COS

To calculate mathematical function Tangent of an angle use the following formula: TANGENT(angle) = SIN(angle) / COS(angle)

EXAMPLE: *V12 = SIN(30) / COS(30)*
V12 would be equal to the tangent function of a 30 degree angle

18.4.9 SQR

SQR will return the square root of a number. If the number is negative, an error is printed and the program will halt.

EXAMPLE: *V46 = SQR (9) V46 would be equal to 3*
*V45 = SQR (V1 - (V63*V29)) V45 would be equal to the square root of (V1 - (V63*V29))*

18.5 MACRO COMMANDS

Macro commands help direct each equation. Processing will stop if the commands are incorrectly used. Computational errors will cause an error message to be printed on the screen. The processing of macros can be executed step by step using the DEBUG command.

18.5.1 CLEAR

CLEAR is used to zero macro variables. The variables may need to be cleared at the beginning of a macro routine or at some place within the program. The CLEAR macro can zero one variable at a time or a range or list of variables. If the CLEAR statement is left blank, ALL variables (V1-V100) are cleared.

> NOTE
> It is recommended that all variables be cleared at the start of each program.

EXAMPLE: *#CLEAR This zeroes all macro variables*

EXAMPLE: *# CLEAR V1 This line will zero only variable V1*
 # CLEAR V1-V20 This line will zero variables V1 through V20
 # CLEAR V3-V7,V15,V30,V45-V60 This line will zero variables V3 through V7, then V15 and V30, then V45 through V60

18.5.2 GOTO

The GOTO statement is used to redirect the program. When used separately, the program is redirected to the line number or label indicated.

EXAMPLE: *N3 #V1= 1 This sets V1 equal to 1*
N4 # IF V1 >2 THEN GOTO :INSIDE

This checks the value of V1. If V1 is less than or equal to 1 continue at N5, if the value of V1 is greater than 1 then continue at label :INSIDE
N5 X1.0 Y-2.3 G1
N6 X3.6 Y-3.0
N15# :INSIDE   This is label :INSIDE
*N16 # R3 = V2 * V17   This line transfers the multiplication of V2 * V17 to R3 parameter*
N25 X+ R3 F20. G1   This transferred R3 to X
N26 #GOTO :INSIDE   Continue at label :INSIDE at N15

18.5.3 LABELS

Labels are used to identify a GOTO location in the program. It is usually best to choose labels that are descriptive of the area or instruction to which they are assigned. The colon (:) assigns the LABEL field.

EXAMPLE:

N5 #V1=1   This sets the value of V1 to be 1
. . .
N9 #GOTO :DOMATH   This sends the operation to line N100 to label DOMATH
N10 #:WORK   This the label WORK the end destination of line N110
N11 #V1=V1+1   This changes variable V1 by adding 1 to its value
N12 #IF V1=5 THEN GOTO :DOMATH
This checks the value of V1 If V1 is equal to a value of 5 it goes to N100, label DOMATH,if not the program continues at line N14
N13 #R9=R9+V1   This line transfers the value of V1 to parameter R9
N14 T+R9 M6   This line transfers the value R9 to the Tool number
N...
N100 #:DOMATH   This is the end destination from the GOTO on line N5.
N...
N110 #GOTO :WORK   This sends the operation to line N10 to label WORK
Labels may not have a comment on the same line. See below:

N2 #:TEST ‘THIS IS A TEST   A comment is not allowed on a label line

18.5.4 IF - THEN

The IF-THEN command is used for comparisons. The IF part of the macro line will examine a variable or equation and if it is true, it will execute the THEN part of the macro line. If the IF part of the macro is not true, it will not execute the THEN part of the macro line and will continue with the next line in the program. When N words are used for the THEN part of the statement, they are NOT renumbered when the NU command is used in the control. If the program is renumbered, then the N word must be altered to match the program.

> **NOTE**
> The THEN part of the statement may also use labels. See Label definitions.

EXAMPLE: Example 1:

N200 # IF V1 > V2 THEN GOTO :PART3
N201 E2 X0 Y0
...
N299 # :PART3
N300 E3 X0 Y0

In line N200, if it is true that V1 is greater than V2, then the program will jump to line number N300. If V1 is not greater than V2 the program will skip to line N201.

The THEN part of the statement may contain any valid macro statement.

EXAMPLE: Example 2:

N100 # IF (V5 + V6) <= 0 THEN V7 = ABS(V5 + V6)

In line N100, if the sum of V5 plus V6 is less than or equal to zero, then let the value of V7 be the absolute value of the sum of V5 plus V6.

18.5.5 INDEX

The INDEX macro is used to send indexer code directly to the FADAL indexer. This feature is only available when used with the FADAL indexer. For more information on the FADAL indexer see the indexer manual.

EXAMPLE: *N5 # INDEX “M1 A90. F300.” This sends CNC code to the FADAL indexer to move the indexer 90 degrees at a feed rate of 300*

18.5.6 INPUT

The INPUT command is used to allow the operator to enter program data during execution of the macro. When the INPUT statement is executed, the program processing stops until the operator presses the ENTER key. The PRINT command may be used to prompt the operator for the desired data to enter. Placing a comma at the end of the PRINT statement will move the cursor to the end of the text on the screen.

EXAMPLE:

N1 #CLEAR
N2 #PRINT “ENTER THE POCKET LENGTH”,
N3 #INPUT V1
N4 # PRINT “ENTER THE POCKET WIDTH”,
N5 #INPUT V2

These statements prompt for the V1 and V2 data required for the program. The program suspends operation until each value is entered.

18.5.7 LABELS

Labels designate a place in the program where program execution may be directed when preselected conditions have or have not been met. Labels are unaffected by program renumbering and take the form #:LABEL. Nothing else may appear on the line. The call to a label will include the colon followed by the name of the label. Each label in a program must have a unique name. If there are any duplicate names, program execution will always go to the first label in the program. Labels may be any alphanumeric the programmer chooses. Labels must be preceded by a colon (:) and there may be no comments or other code in the line with the label. A label indicates a place in the program where program execution may continue after a jump. This may be a routine that is to be repeated several times, or a routine that is to be executed only when certain conditions are met. To make a program easier to read, labels should describe the operation taking place. Such as :LOOP :JUMPBACK.

18.5.8 PRINT

The PRINT statement is used to print text and data to the screen.

> NOTE
> Text that is to be displayed MUST be enclosed in quotation marks.

EXAMPLE:

N4 #V6=25.45
N5 #PRINT “THE VALUE CALCULATED: ”,V6,", IS THE X VALUE."

Using a comma to separate the variable from the text, the screen display is as follows:

THE VALUE CALCULATED: 25.45, IS THE X VALUE.

> NOTE
> Do not use a semicolon to separate the variable from the text.

18.5.9 SET

The SET command is used to change macro parameters. There are five parameters to establish using the SET command. They are DEBUG, RUN, RND#, DEGREES, and RADIANS. Setting DEGREES or RADIANS affects the SIN, COS, and ATN commands. The SET commands are inputted after entering the machine command MA at the command prompt or in the macro program. The DEGREES and RADIANS parameters may also be set within a macro as described below. The SET command, without specifying the setting function, restores all SET parameters to the default.

EXAMPLE:

MA Example:

MA
SET RND5
The “#” sign is not required when using the MA command. This Command is typed at ENTER NEXT COMMAND Prompt.

EXAMPLE: Macro Example:

N24 # SET RADIANS
N25 # V2 = ATN(V1)
N26 # PRINT, V2
N27 # SET DEGREES
N28 # V2 = ATN(V1)
N29 # PRINT, V2

Line N26 prints the angle V2 in radians. Line N28 prints the angle V2 in degrees.

18.5.10 SET DEBUG

The debug command is used temporarily in the macros to proof the macro lines. Debug mode works only when SU from the command mode is used. The advantage to using the debug mode is that it will display the values of the variables. To start the Debug mode, enter #SET DEB in the macro program or SET DEB using the MA command. To end the Debug mode, to enter SET RUN using the MA command or enter #SET RUN in the macro program.

18.5.11 SET DEGREES / RADIANS

SET DEG and SET RAD commands are modal and cancel each other. When using the SIN, COS, or ATN functions the calculations are based upon the DEGREE or RADIANS setting. The default setting is degrees.

*Radians = ((Angle in Degrees * PI) / 180)*
*Degrees = ((Angle in Radians * 180) / PI)*

PI=3.14159265

18.5.12 SET RND#

This function will return a rounded value of a number. The number of places to round to is set using the SET RND# command. To set the rounding to four places enter the command below. The maximum number of places to round to is five.

EXAMPLE: *MA*
SET RND4

> NOTE
> When higher accuracy is desired, the rounding may be calculated. This eliminates the need to use the SET RND# command. To round off numbers to the fifth place use this equation: V1 = INT(V5 * 10000)/10000. The value of V5 will be rounded to the fifth place and V1 will be used for the rounded number. This example only affects the individual line.

18.5.13 SET RUN

This command is used to exit the DEBUG mode. Using the MA command, enter SET RUN to exit the DEBUG mode or enter #SET RUN in the macro program. The program may then be executed.

18.5.14 SINPUT

The SINPUT command is used to wait for and accept data though the RS-232 port during execution of a macro. When the SINPUT statement is executed, the program processing stops until the control receives data through the RS-232. The control will not look ahead of the line with the SPRINT macro command. The LOOK AHEAD process begins when data is received through the RS-232 port. The data sent to the port must terminate with a carriage return.

EXAMPLE:

N1 #CLEAR
N2 #PRINT "WAIT ! NOW ENTERING THE POCKET LENGTH DATA",
N3 #SINPUT V1
N4 #PRINT "WAIT ! NOW ENTERING THE POCKET WIDTH DATA",
N5 #SINPUT V2

18.5.15 SPRINT

The SPRINT command is used to send data out through the RS-232 port. Text and variable data can be sent. The SPRINT statement prints through the RS-232 port the same as the PRINT statement displays to the screen.

> NOTE
> Text must be enclosed in quotation marks.

EXAMPLE:

N43 # SPRINT "PART NUMBER ",V4," DATA"
N44 # SPRINT "X ",AX,"   Y ",AY

In line N43 the words, part number, and data are considered to be text. If the value of V4 is 98645 then N43 would print out:

PART NUMBER 98645 DATA.

18.5.16 START #

The START macro is used to jump to another program. This is used in a situation where the program being called has subroutines. Subroutines are not allowed in a subprogram, this is the reason why this START command is used. This command can be used to link many user main programs. It executes a jump only. No return will occur. The START command would generally be used in place of an M2 or M30 at the end of each program being linked. When the next program is called, the operator must press the AUTO button again to run the next program.

EXAMPLE:

N3245 G0 G90 G49 Z0
N3246 E0 X0 Y0
N3247 #START 7 This calls a jump to program O7

WAIT

This command temporarily pauses processing of the program lines at the line with the WAIT command. Processing will continue when the execution buffer is completely exhausted. This might be used to Print out a message when the program is completely finished. The control will print out through the RS-232 port during the preprocess period unless the WAIT command is used. The WAIT command is also used to have the machine absolutely stationary before printing out the current location of the machine using the axis variables.

EXAMPLE:

N256 G1 G31 Z-30. F1.
N257 # WAIT
N258 # SPRINT "POINT #",V5,": X",AX," Y",AY

In line N257 the control will stop processing the program and wait for the execution buffer to be exhausted before executing line N258 which would print out the current location of the machine.

> NOTE
> The execution buffer is a part of memory used by the control to store processed program data. When the control "looks ahead", this processed data is stored in the execution buffer. Using the SETP command allows the programmer to vary the size of the buffer. On the machine parameter page the term "Binary buffer" is used for the execution buffer.

18.5.17 AND, OR, AND NOT

AND, OR, and NOT are logical operators that allow the programmer to construct compound tests from one or more expressions. In BASIC, a compound Boolean expression is created by connecting two Boolean expressions with a "logical operator". The FADAL macro language allows "logical operators". The number of logical operators in a statement is limited by the maximum number of characters allowed on a single line, 63. This includes all spaces and the pound sign (#).

A good rule for writing compound expressions is to always verify that the statement will evaluate to both a TRUE and a FALSE condition. The items being compared must be segregated with parentheses to avoid confusion in lengthy or complex statements. This will also ensure that the statement's components are evaluated in the order that the programmer intended.

EXAMPLE: Example 1:

AND
IF (V1 GT V2) AND (V1 LT V3) THEN GOTO :LOOP
This first example is true only if expression 1 and expression 2 are both true, then control jumps to the label :LOOP.

EXAMPLE: Example 2:

OR
IF (V1 GT V2) OR (V1 LT V3) THEN GOTO :LOOP
In this example, either condition can be true for control to jump to the label :LOOP.

EXAMPLE: Example 3:

NOT
IF NOT(V1=0) THEN GOTO :LOOP
In this example if V1 is NOT equal to zero then control jumps to the label :LOOP.

EXAMPLE: Example 4:

AND, OR, and NOT
IF (V1 LT V2) OR (V1 LT V3) AND (NOT(V1 GT V4)) THEN GOTO :LOOP

The fourth example shows multiple operators strung together in one statement. Combinations of AND, OR, and NOT can be used to build up very complex expressions. There is no practical limit to their complexity, except the obvious one of writing an expression that may be difficult to read and exceeds 63 characters.

18.6 MACRO TUTORIAL

18.6.1 OVERVIEW

This subsection is designed as a tutorial for FADAL's Macro Language. Several Macro Language commands are referenced here. For a complete list with applicable syntax, review the Macro Commands section.

18.6.2 SUMMARY

Macro programming uses a subset of BASIC to manipulate data for use in G code programs. If a shape can be defined with an equation or all of its elements with variables, then it may be advantageous to use macro programming.

The availability of compare statements and algebraic operators in the form of IF THEN loops and +, -, *, /, SIN, COS and ATN (to list a few), makes it possible to create complex geometry with a few lines of code.

This discussion will include examples of macro programs with their explanations and, where appropriate, the equations from which the geometry is derived. It will start with simple examples and proceed through to the more complex functions.

18.6.3 CONVENTIONS

All macro lines must be preceded by a pound sign # immediately following a line number. The open paren (and asterisk * are macro operators and cannot be used in a macro statement to denote a comment. Instead use an apostrophe ‘ to denote a comment in a macro line. The open and closed parens (|) are grouping operators used to indicate order of operation, use as in an algebraic equation. The asterisk * is a multiplication symbol.

18.6.4 COMMENTS

Comments are very important part of macro programming. They explain the program to others, and remind the programmer why they wrote it the way they did. Comments on a macro line begin with an apostrophe (‘), and can extend to a total line length of 63 characters.

N3 # V100=1.2345 ‘V100 IS THE LENGTH OF THE PART 2

18.6.5 CALCULATIONS

The mathematical operations of addition, subtraction, multiplication and division are represented by +, -, * (asterisk), and / (slash), respectively.

EXAMPLE:

N11 # V10=3/V12

18.7 VARIABLES

18.7.1 V1-V100: MACRO VARIABLES

There are 100 available variable or ‘V’ registers numbered from V1 to V100. These registers may be used to manipulate data but may not be used directly in the G code program. The manipulated values must be passed to the R registers for use in the program. There are ten R registers numbered R0 to R9. Since the control uses R0 through R4 in various fixed cycles, it is usually advantageous to pass values to the R registers starting with R9 and proceeding backwards to R0. Be careful not to ‘step on’ any values in use. Re-assign values if necessary.

These variables are used in macro language statements.

EXAMPLE:

N88 # V1 = V2 + V3

18.7.2 ARRAYS

Variable arrays can now be used; that is, the subscript of a variable can now be a variable or arithmetic expression. The subscript variables are D, FX, FY, FZ, H, PX, PY, PZ, PA, PB, R, and V. The subscript for a variable must not exceed the allowed range for the variable. For instance, the subscript for a V variable must be 1 through 99 and the subscript for a FX variable must be 1 through 48.

EXAMPLE:

If the variables V1=5 and V2=1 then the following are equivalent:

#V5=1
#V(5)=1
#V(V1)=1
#V(V1+V2-1)=1
#V(V1/V2)=1
#V(V(SQR(V2))=1

18.7.3 AX, AY, AZ, AA, AB, AC: AXIS POSITION VARIABLES

Axis position variables are used to read the current location of the desired axis.

N79 # IF AZ >12.050 THEN GOTO :GETNEXT

In the above example, if the Z axis is at 12.050 inches by the time this line is executed, then the control will jump to the line in the program with the label :GETNEXT.

EXAMPLE:

N99 E2 X0 Y0
N100 Z-30 G31 F50.
N101 G91 Z.05
N102 G90
N103 Z-30. G31 F1. The probe will stop the Z axis when it touches
N104 #IF AZ > 12.050 THEN GOTO :GETNEXT

18.7.4 CLEAR

The CLEAR statement is used to zero the values in the variable table. If used alone it will clear all of the variables in the table. If a variable number is included after the command CLEAR the variable or variables specified will be cleared.

EXAMPLE:

#CLEAR Clears all V registers OR
#CLEAR V1 Clears only V1 OR
#CLEAR V10-V30 Clears V10 through V30

In normal programming practice a CLEAR statement at the beginning of the program will ensure that all of the variables will be zeroed. This is done to prevent a variable register that the program will use from having a value that may shorten a loop sequence or exceed the test value.

For example, if a statement reads #IF V1 <|> 10 THEN GOTO :JUMP with the intent of looping 10 times, and the value of V1 is 15 from a previous program, the program will always go to the label :JUMP. This line is an example of a bad programming practice. The only way a loop written with this statement will ever end would be if V1 were equal to 10. If the loop were counting up it would be better written as #IF V1 <= 10. In this form once the count exceeded 10 the program would drop out of the loop. If a specific value is required in a register it must be entered by the programmer with an assign statement.

To assign a value to a V variable use the equal sign. Ex: #V1=10 #V1=V2 #V1=R9 #V1=SIN(V10) any valid Macro function may be used to assign a value to a Macro variable. The R variables may also be assigned values in the same manner as a V variable. In addition, values may be directly assigned outside of a Macro statement using a + or - . The value passed to an R variable in this manner must be either a number or another R variable.

EXAMPLE: *R9+10. R8-18. R7+R6*

18.8 OPERATOR INTERACTION

18.8.1 PRINT

PRINT statements are used to display messages on the screen for the operator. Anything written that will not change must be enclosed in quotes. Any values represented by V or R variables must be outside of the quotes (“) and separated by a comma (,).

#PRINT V10
#PRINT “CHANGE PART”
#PRINT “THIS IS PART NUMBER ”,V10
#PRINT “THIS IS PART NUMBER ”,V10,"THERE ARE “,V11,”PARTS LEFT"

SCREEN DISPLAY:

36920
CHANGE PART
THIS IS PART NUMBER 36920
THIS IS PART NUMBER 36920 THERE ARE 121 PARTS LEFT

18.8.2 INPUT

An INPUT statement requires that a variable register be specified to receive the input value, this may be requested by the PRINT statement. The operator will enter a requested value and that value will be placed in the variable register specified in the INPUT statement then continue program operation. Pressing the return key without entering a value will enter a zero (0) in the variable register specified and continue program execution.

The INPUT statement is used to halt program execution to receive data input and to display a print statement. It is assumed that a print statement will request some action from the operator. A PRINT statement only displays to the screen. PRINT will not halt program execution and therefore may scroll off of the screen before the operator can read it.

EXAMPLE:

#PRINT "ENTER THE DIAMETER OF THE END MILL"
#INPUT V100

18.9 PROGRAM BRANCHING

18.9.1 LABELS

Labels designate a place in the program where program execution may be directed when preselected conditions have or have not been met. Labels are unaffected by program renumbering and take the form #:LABEL. Nothing else may appear on the line. The call to a label will include the colon followed by the name of the label. Each label in a program must have a unique name. If there are any duplicate names, program execution will always go to the first label in the program. Labels may be any alphanumeric the programmer chooses. Labels must be preceded by a colon (:) and there may be no comments or other code in the line with the label. A label indicates a place in the program where program execution may continue after a jump. This may be a routine that is to be repeated several times, or a routine that is to be executed only when certain conditions are met. To make a program easier to read, labels should describe the operation taking place. Such as :LOOP :JUMPBACK.

18.9.2 GOTO

The GOTO statement may also be used in a line by itself to redirect program execution. In this form it is an unconditional jump, meaning there are no conditions to be met for the jump. Program execution is directed to the specified line or label.

EXAMPLE:

#GOTO N45   OR
#GOTO :LABEL

These lines will cause program execution to branch to the line or label specified. GOTO statements may address a line number directly; however, if the program is re-numbered it will be necessary to verify that all of the addresses have not changed. The control WILL NOT update any N words in GOTO statements when a program is re-numbered. A preferred method is to use macro labels.

18.9.3 IF-THEN

The IF-THEN statement provides the FADAL macro programming language the flexibility necessary for compact, powerful programs. It takes the form IF some condition is true THEN do something or go to an address.

The condition to be met may be a simple statement comparing some A to some B:

EXAMPLE:

#IF V1=V2
This compare statement looks for equal values in variables V1 and V2
#IF V1=8
This compare statement checks to see if the content of variable V1 is equal to 8

Some valid equalities: V1=R1 R1=V1 V1=SIN(V2) V5=TN. The condition may also be a Boolean Equation.

EXAMPLE:

*#IF V1*V2=3   Which reads: if V1 times V2 equals V3*
or
*#IF V1*V2< >V3 Which reads: if V1 times V2 is not equal to V3*

18.9.4 SYMBOLIC OPERATORS

The Boolean operators used in the FADAL Macro language are:

=	*or*	*EQ for Equal*
<	*or*	*LT for Less than*
>	*or*	*GT for Greater than*
<=	*or*	*LE for Less than or equal to*
>=	*or*	*GE for Greater than or equal to*
<|>	*or*	*NE for Not equal to*

The THEN portion of the statement may be used to assign or re-assign a value to a variable or register.

EXAMPLE:

THEN V1=V5 or THEN R6=SIN(V34)

The THEN statement may also be used to re-direct program execution.

EXAMPLE:

THEN GOTO N45   Re-directs program execution to line number N45
or
THEN GOTO :LOOP   Re-directs program execution to the label :LOOP
If program execution is re-directed to a line number, it will be necessary to verify that the line number is correct any time the program is re-numbered. When the program is re-numbered the control WILL NOT update the line numbers in an IF THEN loop or a GOTO statement. For this reason it is preferable to use labels to re-direct program execution.

A complete IF THEN statement will take the form:

#IF V1 <= 28 THEN GOTO :LOOP

This line reads, if the contents of variable register V1 are less than or equal to 28, then go to LOOP. This line could be used in a program with V1 as a counter to perform some operation until the value of V1 is greater than 28, once the value of V1 exceeds 28, program operation will continue with the program line following the IF THEN statement.

#IF V1 = 1 THEN V20 = V5

This line reads, if the contents of variable register V1 are equal to 1, then make the contents of variable register V20 equal to the contents of variable V5. This IF THEN statement would be used to set or reset the value of variable register V20 to the value in V5 once V1 is set to 1.

18.10 AND, OR, AND NOT

18.10.1 LOGICAL OPERATORS

AND, OR, and NOT are logical operators that allow the programmer to construct compound tests from one or more expressions. In BASIC, a compound Boolean expression is created by connecting two Boolean expressions with a “logical operator”. The FADAL macro language allows “logical operators”. The number of logical operators in a statement is limited by the maximum number of characters allowed on a single line, 63. This includes all spaces and the pound sign (#).

A good rule for writing compound expressions is to always verify that the statement will evaluate to both a TRUE and a FALSE condition. The items being compared must be segregated with parentheses to avoid confusion in lengthy or complex statements. This will also ensure that the statements components are evaluated in the order that the programmer intended.

EXAMPLE:

Example 1:

AND
IF (V1 GT V2) AND (V1 LT V3) THEN GOTO :LOOP
This first example is true only if expression 1 and expression 2 are both true, then control jumps to the label :LOOP.

EXAMPLE:

Example 2:

OR
IF (V1 GT V2) OR (V1 LT V3) THEN GOTO :LOOP
In this example either condition can be true for control to jump to the label :LOOP.

EXAMPLE:

Example 3:

NOT
IF NOT(V1=0) THEN GOTO :LOOP
In this example if V1 is NOT equal to zero then control jumps to the label :LOOP.

EXAMPLE:

Example 4:

AND, OR, and NOT
IF (V1 LT V2) OR (V1 LT V3) AND (NOT(V1 GT V4))
THEN GOTO :LOOP

The fourth example shows multiple operators strung together in one statement. Combinations of AND, OR, and NOT can be used to build up very complex expressions. There is no practical limit to their complexity, except the obvious one of writing an expression that may be difficult to read and exceeds 63 characters.

18.10.2 COUNTING LOOPS

Counting loops may be initiated to count the number of parts machined, holes drilled, etc. The basic code generally takes the form:

 # V49 = V49 + 1
 # D98 = D98 + 1

The values may be placed in any unused register, for example, any valid V, D, R, H, TT, or FO register. When using a register for counting, care must be taken to insure that the value in the register is not overwritten by another function in use by the control.

Inaccuracies in the count may be caused by the look-ahead feature in the control. This may be overcome with the use of a macro WAIT statement in the line preceding the count statement. The WAIT statement does not cause the machine to stop, it stops look ahead past this point until the program execution actually reaches this point in the program. This will keep the counter from incrementing until the event being counted actually occurs.

Counting loops are best placed at the end of the program.

18.11 FADAL MACRO LANGUAGE EXAMPLES

18.11.1 D-HOLE MACRO

1. Move to the center of the D-Hole.
    a. This move can be made incrementally, but this macro switches to absolute. So remember, if you are programming in incremental, enter the G91 code on the next line of the program after the call to this macro.
2. Position the Z axis to the desired Z depth.
3. Establish a feed rate.
4. Use a D word to establish the tool diameter offset to use.
    a. This macro assumes that the tool diameter is entered into the tool table.
    b. For tool tables set up for radius Y, enter the radius.
5. Type the R words and M98 P800.
    a. R5: Use 0 for CW, and 1 for CCW
    b. R6: Circle radius
    c. R7: Blend radius
    d. R8: Distance to the line
    e. R9: Angle

EXAMPLE:

```
O1 (D-HOLE EXAMPLE: MAIN PROGRAM
(TOOL #1, .5 DRILL
M6 T1
G90 G0 S5000 M3 E1 X2. Y-3.
H1 Z.1 M8
G81 G99 R0+.1 Z-.6 F40. X2. Y-3.
G80
M5 M9
G90 G0 G49 Z0
(TOOL #2, .5 HSS 2FL EM
M6 T2
G90 G0 S7000 M3 E1 X2. Y-3.
H2 Z.1 M8
Z-.3 G1 F30.
F45.
D2
R5+1. R6+2. R7+.5 R8-.7 R9+0 M98 P800
Z.1 G0
M5 M9
G49 Z0
E0 X0 Y0
    M2
```

18.11.2 SUB PROGRAM 800

```
N1O800(D-HOLE MACRO: SUB PROGRAM
N2#'1ST MOVE TO D HOLE CENTER, THEN POS. Z AXIS)
N3#'R5=0 FOR CW, 1 FOR CCW)
N4#'R6=RADIUS OF CIRCLE
N5#'R7=BLEND RADII
N6#'R8=X DIR & DIS TO LINE
N7#'R9=ANGLE
N8#V1=AX 'GET CURRENT X POSITION
N9#V2=AY 'GET CURRENT Y POSITION
N10#V3=R5 'CW OR CCW
N11#V4=R6 'RADIUS OF CIRCLE
N12#V5=R7 'BLEND RADIUS
N13#V6=R8 'X DIR & DIS TO LINE
N14#V8=V4-V5
N15#V10=-(ABS(V6)-V5) 'X CENTER OF BLEND
N16#V11=SQR((V8*V8)-(V10*V10)) 'Y CENTER OF BLEND
N17#V12=V11/V10 'SLOPE OF LINE THROUGH CENTER OF CIRCLE AND CENTER
N18#V13=-SQR((V4*V4)/(1+(V12*V12))) 'X END PNT ON BND RAD
N19#V14=V12*V13 'Y END PNT ON BND RAD
```

```
N20#IF R5=0 THEN GOTO :CW
N21#R3=V6 'FIRST X MOVE TO LINE
N22#R4=0
N23 G90 G8
N24 M98 P900
N25 X+R5 Y+R6 G1
N26#R3=V6
N27#R4=-V11 'Y MOVE TO BND RAD
N28 M98 P900
N29 X+R5 Y+R6
N30#R1=R5
N31#R2=R6
N32#R3=V10
N33#R4=-V11
N34 G8
N35 M98 P900
N36#R1=R5-R1 'I TO B.R. CENTER
N37#R2=R6-R2 'J TO B.R. CENTER
N38#R3=V13 'X BND RAD END POINT
N39#R4=-V14 'Y BND RAD END POINT
N40 M98 P900
N41 X+R5 Y+R6 I+R1 J+R2 G3
N42#R1=V1-R5
N43#R2=V2-R6
N44#R3=V13
N45#R4=V14 'Y CIRCLE END POINT
N46 M98 P900
N47 X+R5 Y+R6 I+R1 J+R2
N48#R1=R5
N49#R2=R6
N50#R3=V10
N51#R4=V11
N52 M98 P900
N53#R1=R5-R1
N54#R2=R6-R2
N55#R3=-V6 'X BND RAD END POINT
N56#R4=V11 'Y BND RAD END POINT
N57 M98 P900
N58 X+R5 Y+R6 I+R1 J+R2
N59#R3=-V6
N60#R4=0
N61 M98 P900
N62 X+R5 Y+R6
```

```
N63#R3=0
N64#R4=0
N65 M98 P900
N66 X+R5 Y+R6 G40
N67 M99
N68#:CW
N69#R3=V6 ‘FIRST X MOVE TO LINE
N70#R4=0
N71 G90 G8
N72 M98 P900
N73 X+R5 Y+R6 G1
N74#R3=V6
N75#R4=V11 ‘Y MOVE TO BND RAD
N76 M98 P900
N77 X+R5 Y+R6
N78#R1=R5
N79#R2=R6
N80#R3=V10
N81#R4=V11
N82 M98 P900
N83#R1=R5-R1 ‘I TO B.R. CENTER
N84#R2=R6-R2 ‘J TO B.R. CENTER
N85#R3=V13 ‘X BND RAD END POINT
N86#R4=V14 ‘Y BND RAD END POINT
N87 M98 P900
N88 X+R5 Y+R6 I+R1 J+R2 G2
N89#R1=V1-R5
N90#R2=V2-R6
N91#R3=V13
N92#R4=-V14 ‘Y CIRCLE END POINT
N93 M98 P900
N94 X+R5 Y+R6 I+R1 J+R2
N95#R1=R5
N96#R2=R6
N97#R3=V10
N98#R4=-V11
N99 M98 P900
N100#R1=R5-R1
N101#R2=R6-R2
N102#R3=V6 ‘X BND RAD END POINT
N103#R4=-V11 ‘Y BND RAD END POINT
N104 M98 P900
N105 X+R5 Y+R6 I+R1 J+R2
```

N106#R3=V6
N107#R4=0
N108 M98 P900
N109 X+R5 Y+R6
N110#R3=0
N111#R4=0
N112 M98 P900
N113 X+R5 Y+R6 G40
N114 M99

18.11.3 SUB PROGRAM 900

N1O900(ROTATE X & Y)
*N2#V30=(R3*COS(R9)-R4*SIN(R9))*
*N3#V31=(R3*SIN(R9)+R4*COS(R9))*
N4#R5=V30+V1
N5#R6=V31+V2
N6 M99

18.11.4 ROW COLUMN PATTERN MACRO

1. Start the fixed cycle
2. Type the R words:

R9 = Number of holes across *R7 = Number of holes down*
R8 = Space between X axis holes *R6 = Space between Y axis holes*

EXAMPLE:

O10(ROW & COLUMN: MAIN PROGRAM#CLEAR
#SET RUN
G90 G0 G40 G80 G17
E0 X0 Y0
G90 G0 S1000 M3 E1 X0 Y0
H1 Z0.1 M8
G81 G99 Z-1.R+0.1 F100. M45
R9+3.R8+1.R7+3.R6+1.
M98 P810
G80
M5 M9
G90 G0 G49 Z0
E0 X0 Y0
M2

18.11.5 SUB PROGRAM 810

```
O810(ROW & COLUMN: SUBPROGRAM
#'R9 IS THE NUMBER OF HOLES ACROSS
#'R8 IS THE SPACE BETWEEN X HOLES
#'R7 IS THE NUMBER OF HOLES DOWN
#'R6 IS THE SPACE BETWEEN Y HOLES
#V1=R9
#V2=R8
#V3=R7
#V4=R6
#V5=V1
#R5=V5
#V50=0'V50 IS THE HOLE COUNTER ACROSS
#V60=0'V60 IS THE HOLE COUNTER DOWN
#V70=0'V70 IS LAST ROW COUNTER
G91
#:RIGHT
#IF V50=V1 THEN GOTO:BACK
#V50=V50+1
X+R8
#GOTO:RIGHT
#:BACK
#V60=V60+1
#V50=0
#R4=(R9*R8)-R8
X-R4 Y-R6
#IF V60=V3 THEN GOTO:LAST
#GOTO:RIGHT
#:LAST
X+R8
#V70=V70+1
#IF V70=V5 THEN GOTO:END
#GOTO:LAST
#:END
M99
```

18.11.6 SPIRAL CUT MACRO

```
O5757
#CLEAR  Clears all macro variables
G0 G90 G80 G40 G49
S1800 M3 M7
G0 G90 X0 Y0
H1 Z.1
G1 Z-.25 F20.
```

```
#V7=0 - Sets V7 to zero
#:LOOP - Label :LOOP
#V7=V7+1 - Add 1 to V7
#IF V7>=360 THEN V7=0  Reset angle if over 360 degrees
#V1=V1+.00077  Set radial increments per degree
#V2=SIN (V7)*V1  Calculate X component
#V3=COS(V7)*V1  Calculate Y component
#R9=V2  Transfer V2 value to R9
#R8=V3  Transfer V3 value to R8
#R7=V1  Transfer V1 value to R7
G2 X+R9 Y+R8 F30. R+R7  Machines part to macro calculations
#IF V1<10. THEN GOTO :LOOP  Stop when 10. inches
N20 G0 Z.1 M5 M9
N21 G28
N22 M2
```

18.11.7 DRILL GRID PATTERN MACRO - WHOLE NUMBER INCREMENTS

The following is a simple program to drill a grid of 100 holes in a 10 by 10 pattern. The program begins with normal G code programming, then clears the values in the variable registers to ensure a proper count for our loops. It uses the variable V1 for the X position counter and V2 for the Y position counter. These values will be passed to the R9 and R8 registers for use in the G code portion of the program. Only one label will be required for the loop. The first IF THEN statement will check to see when all of the holes in the X axis are drilled. When the count reaches 10, program execution will ‘fall out’ of the loop. The next statement zeroes (re-initializes) the X axis counter and the following statement increments the Y axis value. The second IF THEN statement checks to see when all of the rows in the Y axis have been drilled. In this program the values in the variable registers serve as counters as well as the value to increment for each X and Y move.

```
O1234(DRILL GRID 10-1" X STEPS, 10-1" Y STEPS
G90 G80 G40 G0 Z0
S2500 M3 M7
G0 X0 Y0 E1
Z.1 H1
G81 X0 Y0 Z-.75 R0.1 G98 F20.
#CLEAR  Sets all variable registers to zero
#:LOOP   Label for the loop
#V1=V1+1   Increases the value in V1 by 1
#R9=V1   Assigns the value in V1 to R9
#R8=V2   Assigns the value in V2 to R8
X+R9 Y+R8   Moves to locations specified by R8 and R9
#IF V1<=9 THEN GOTO :LOOP  Tests if 10 holes have been drilled
#V1=0   Resets V1 to zero for the next row
```

```
#V2=V2+1  Increases the value in V2 by 1
#IF V2<=9 THEN GOTO :LOOP  Tests if 10 rows have been drilled
G80 M5 M9
G49 Z0
G0 E0 X0 Y0 Z0
M2
```

This program drills 100 holes with 20 lines of code. An equivalent G code program would use approximately 109 lines to accomplish the same thing.

18.11.8 DRILL GRID PATTERN MACRO - DECIMAL INCREMENTS

If it is necessary to program the moves in other than whole number increments, then it will be necessary to change the program format only slightly. The R9 and R8 values will be incremented by the desired values and must be initialized prior to the loop statement. The variables V1 and V2 are retained as counters.

```
O1235(DRILL GRID 10-.375" X STEPS, 10-.5" Y STEPS
G90 G80 G40 G0 Z0
S2500 M3 M7
G0 X0 Y0 E1
Z.1 H1
G81 X0 Y0 Z-.75 R0.1 G98 F20.
R9+0 R8+0   Clears the R9 and R8 registers
#CLEAR   Sets all variable registers to zero
#:LOOP   Label for the loop
#V1=V1+1    Increases the value in V1 by 1
#R9=R9+.375    Increases the value in R9 for the next hole
X+R9 Y+R8    Moves to the locations specified by R8 and R9
#IF V19 THEN GOTO :LOOP    Tests if 10 holes have been drilled
#V1=0    Resets V1 to zero for the next row
#V2=V2+1    Increases the value in V2 by 1
#R9=0    Resets R9 to zero to start at beginning of line
#R8=R8+.5    Increases the value in R8 for the next row
#IF V29 THEN GOTO :LOOP    Tests if 10 rows have been drilled
G80 M5 M9
G49 Z0
G0 E0 X0 Y0 Z0
M2
```

18.12 TUTORIAL PROGRAM SUMMARIES

18.12.1 SYNOPSIS

The following programs have been designed as an integral part of this tutorial. Each program is summarized in this section and then each program is explained line-by-line in the following section. The last section contains listings of each program.

18.12.2 PROGRAM NUMBER 1

This is a simple program to check the length of tool number 1 and enter this length into the tool offset table. Fixture offset number 23 must contain the distance from the Z zero surface to the top of the tool probe. Fixture offset number 24 must contain the X and Y location of the tool probe.

18.12.3 PROGRAM NUMBER 2

This program is similar to program 1 in that it uses the TS-20 or TS-27 tool setting probe to check for tool condition. The function of this program is to check for broken tools.

18.12.4 PROGRAM NUMBER 3

This program gets the tool number for the tool in the spindle then uses that tool to drill a set of holes. It then checks the time that the drill is cutting against the time entered in the tool table. If the time is less than the total time in the table then the routine will continue. Once the time has exceeded the time allowed, the program will increment the tool number and load the next tool. This process will continue until all tools in the tool changer have been used. In order for this program to function properly, the timers must be set to 2)DO NOT CHECK. This function is found on page two of the parameters. In addition, all tools to be used must be loaded and their respective tool lengths entered in the tool table.

18.12.5 PROGRAM NUMBER 4

This program takes the diameter of a large drill then calculates the amount of each peck and the minimum dwell time required to relieve pressure on the drill. This allows the machine to more efficiently drill a large hole with less strain on the machine.

The formula used takes the number of milliseconds in a minute 60,000 divided by the RPM multiplied by .75. 60000/RPM equals the dwell time for one complete revolution. Multiplying by .75 gives 3/4 revolution to relieve the pressure.

18.12.6 PROGRAM NUMBER 5

This program will drill a circular bolt hole pattern as defined by the user. The Sine and Cosine functions are used to define the X and Y dimensions for the first hole. The program then rotates the hole an amount determined by the number of holes.

18.12.7 PROGRAM NUMBER 6

This sub program probes two bores in line on the X axis. It then resets the fixture offset to the center of the left hand bore and rotates the coordinate system to align with the C/

L of the bores. The left hand bore will be set to X0 Y0. The distance between bores is assumed to be 5". This dimension may be adjusted in line N14. Bores up to approximately 2.5" dia. may be checked. This value will be influenced by the probe diameter and may be changed by modifying the X and Y values in the L9101 R1+1 lines.

18.12.8 PROGRAM NUMBER 7

This program will cut a rectangular pocket with tapered sides. The pocket will have been roughed to the bottom finish size. The angle of the sides in this example is five degrees and the Z step will be .01. The calculation for the X and Y axis step over is Tan 5 * .01 = the step over distance. The tangent point for the bottom of the pocket must be calculated for the start dimensions. If the pocket is through, the Z depth will be equal to the depth of the bottom of the pocket plus 1/2 the diameter of the ball nose end mill that is being used. The X Y shift amount would be equal to the radius of the cutter divided by the COS of the angle. EX. .1875/COS 5 .1875/.99619=.1882 would be the correction factor for a 3/8 end mill at 5 degrees. If the pocket is not a through pocket, Z depth will be equal to the depth of the pocket minus 1/2 the diameter of the ball end mill. The X Y shift amount would use the same calculations as the through pocket plus a correction factor for the difference in depth, which would be TAN of the Angle times the Radius of the cutter.

EX: TAN 5 * .1875 = .087488*.1875 = .0164

This is the depth factor. Subtract this from the Angle factor .1882 for the total correction factor: .1882-.0164=.1718

18.12.9 PROGRAM NUMBER 8

Sine Wave - A sine wave is constructed by laying out a line that represents one 360 degree revolution. In the case of a cam wrap, this would be equal to the circumference of the part to be cut. An example would be a 1 1/2" diameter that has a circumference of PI*dia=3.1416*1.5=4.7124. This line is then divided into equal segments determined by the for requirements on the blue print. Many segments give a high resolution and better form; few segments give a low resolution. Two degree segments are acceptable for most applications. Once the number of segments is determined, the line is divided by this number to obtain the Y axis step. Since our line is 360 degrees long, we find the number of steps by dividing 360 by the angular increment, in this case two. 360/2=180 steps. We then divide the length of our line (4.7124) by the number of steps (180) 4.7124/180=.0262. This will be the amount of Y axis movement for each two degree increment on the Sine wave. The X axis movement is determined by multiplying the Sine of the accumulated angle by the height of the wave. The height of the wave will be half of the total X axis movement. If we require a total movement of 1.25 inches in the X axis our wave will be 1.25/2=.625.

Our formulas are:

PI*Dia = Circumference

360/Angular increment = Y axis move

Sine of the angle * Wave Height = X axis move

This program will ‘wrap’ a sine wave around a round part on a fourth axis.

18.12.10 PROGRAM NUMBER 9

Ellipse - An ellipse is defined as a collection of points whose locations are the sum of the distances from two fixed points such that the sum of the distances is always equal. The formula is given as (X Sq/ a Sq)+(Y Sq / b Sq)=1, where (*a)* is equal to the X radius of the ellipse and (*b)* is equal to the Y radius of the ellipse. By re-arranging the formula to solve for Y we obtain +-Y = +/- Square Root of (b2 - ((X2 * b2) / a2)). Because the formula solves for plus or minus Y we will need two loops to complete the ellipse, one for the Y+ and one for the Y-. The X axis will be incremented in .01 steps.

Formula: +-Y = +/- Square Root of (b2 - ((X2 * b2) / a2))

18.13 TUTORIAL PROGRAM EXPLANATIONS

18.13.1 PROGRAM NUMBER 1

This is a simple program to check the length of tool number 1 and enter this length into the tool offset table. Fixture offset number 23 must contain the distance from the Z zero surface to the top of the Tool Probe. Fixture offset number 24 must contain the X and Y location of the tool probe.

N1 O99 (CHECK TOOL
Contains the program number and a comment
N2 G90 G0 E24 X0 Y.25 S250. M4
Sets absolute and rapid modes, moves to X0 Y.25 on the Tool Probe and turns the spindle on backward at 250. RPM
N3 M65
Turns on the TS-20 tool setting probe
N4 H1 E23 Z1.
Applies the tool length offset for tool #1 plus the Z axis correction for the Fixture location plus one inch. This moves the tool tip to one inch above the tool probe
N5 G1 G31 Z-.1 F20.
Commands the probe skip function and feeds the tool into the tool probe
N6 G91 Z.05
Commands an incremental Z axis move .05 off of the tool probe
N7 G90 S500
Sets absolute mode and increases the RPM to 500
N8 G1 G31 Z-.1 F1.

Commands probe skip function and feeds the Z axis back into the tool probe at a slower feed for improved accuracy
N9 #R9=AZ
Reads the current Z location (AZ) into register 9 (R9). This is the actual Z value from the Home position
N10 #R8=R9-FZ23
This macro statement subtracts the Z fixture location in FIXTURE OFFSET 23 (FZ23) from the value in register 9 (R9) and places it in register 8 (R8) This compensates for the difference in Z from the Z zero position to the top of the tool probe
N11 G10 L10 P1 R0+R8
Uses the G10 function to store the value in register 8 (R8) into tool offset number 1
N12 M5 G0
Turns off the spindle and sets rapid mode
N13 G0 G90 H0 Z0
Sets absolute mode, cancels the tool offset and moves to Z zero
N14 E0 X0 Y0 Z0
Cancels the fixture offset and moves to X0 Y0 and Z0
N15 M99
Code to return from sub program. Will return control to the main or calling program

18.13.2 PROGRAM NUMBER 2

This program is similar to program 1 in that it uses the TS-20 or TS-27 tool setting probe to check for tool condition. The function of this program is to check for broken tools.

N1 O2(SUB TO CHECK FOR TOOL BREAKAGE
Contains the program number and a comment
N2 G90 G0 E24 X0 Y.25 S250. M4
Sets absolute and rapid modes, moves to X0 Y.25 on the touch probe and turns the spindle on backward at 250 RPM. Make sure the Y axis move will clear the tool probe
N3 #R8=TN
Sets register 8 (R8) equal to the tool number
N4 #H99=H(R8)
Sets tool offset number 99 (H99) equal to the tool offset indicated in register 8 (R8)
N5 #H99=H99+FZ23
Adds the correction factor for the difference from Z zero to the top of the probe

> NOTE
> Lines Four and Five may be combined to eliminate a step:

N4 #H99=H(R8)+FZ23

N6 M65
Turns on the tool probe
N7 H99 Z1. (TOOL LENGTH ENTERED IN OFFSET #99 TO PROBE
Applies the corrected tool length offset to one inch above the tool probe
N8 G1 Z-.1
Brings the tip of the tool below the edge of the probe
N9 G1 G31 Y0
Turns on the probe skip function and moves the tool into the tool probe
N10 #R9=AY
Reads the Y axis position into register number 9 (R9)
N11 G0 Y.25
Moves the tool off of the tool probe
N12 H0 Z0 G0 E0 X0 Y0
Cancels all offsets and sends the machine to the home position
N13 #IF R9>0 THEN GOTO :EXIT

Tests the value of R9. If the value of R9 is greater than Zero the tool is not broken. This is because once the tool makes contact with the probe, axis motion is stopped. If the tool is broken it will not make contact with the tool probe and will complete the move to Y0. If the value of R9 is greater than zero the THEN portion of the command will cause program execution to continue on the line with the :EXITlabel. If the value of R9 is zero, this line will cause no action. The program will continue with the next line.

N14 M0 Halts program execution. If the tool is broken the program will halt execution here. A #PRINT statement could be used here to alert the operator to a broken tool.
N15 #:EXIT This is the label for the jump from line 13 to continue program execution
N16 M99 Code to return from sub program. It will return control to the main or calling program

18.13.3 PROGRAM NUMBER 3

This program gets the tool number for the tool in the spindle then uses that tool to drill a set of holes. It then checks the time that the drill is cutting against the time entered in the tool table. If the time is less than the total time in the table, then the routine will continue. Once the time has exceeded the time allowed, the program will increment the tool number and load the next tool. This process will continue until all tools in the tool changer have been used. In order for this program to function properly, the timers must be set to 2)DO NOT CHECK. This function is found on page two of the parameters. In addition, all tools to be used must be loaded and their respective tool lengths entered in the tool table.

*N1 03*TOOL TIME**
Contains the program number and a comment.
N2 G0 G90 E1 X0 Y0

Sets the rapid and absolute modes then moves to X0 Y0 at fixture offset number 1
N3 #V9=TN ‘GET THE TOOL NUMBER
Assigns the number of the tool currently in the spindle to variable 9 (V9)
N4 #R9=V9 Assigns the tool number to register 9 (R9)

> NOTE
> Lines three and four may be combined to eliminate a step:

#R9=TN
N5 #:LOOP
Contains the label :LOOP. This is where an IF THEN loop will send the program
N6 M6 T+R9
Commands a tool change to the tool indicated by register 9 (R9)
N7 Z.1 H+R9
Applies the tool offset to .1 above Z zero
N8 G81 G98 Z-.1 R0+.1 F50. M45
Drills a hole
N9 G91 X.5 L20
Drills twenty holes in the X axis .5" apart
N10 G80 G90 M5M9
Cancels the drill cycle, turns off the spindle and coolant, resets the absolute mode
N11 X0 Y0
Moves back to X0 Y0.
N12 #IF TU(R9)<TT(R9) THEN GOTO :LOOP

Tests the time used against the time set in the Tool Time table. If the time used is less than the time in the table, program execution continues at the :LOOP label in line N5. If the tool time is expected to be the same for all of the tools TT(R9) may be changed to TT1. This will eliminate the need to set the tool time several places in the table, all times will be checked against the value in one table. If the time used is more than the time set in the Tool Time Table, program execution will continue on the next line.

N13 #R9=R9+1
Increments the tool number by one
N14 #TU(R9)=0
Resets the Time used in the Tool Time table to Zero
N15 #IF R9<22 THEN GOTO :LOOP

Tests the value of R9 to see if all of the tools have been used. If the tool number is 21 or lower, program execution will continue at :LOOP line N5. If the tool number is greater *than 21 program execution will continue on the next line. NOTE: The test value may be changed to test for more or fewer tools.*

N16 M2
End program code

18.13.4 PROGRAM NUMBER 4

This program takes the diameter of a large drill then calculates the amount of each peck and the minimum dwell time required to relieve pressure on the drill. This allows the machine to more efficiently drill a large hole with less strain on the machine.

The formula used takes the number of milliseconds in a minute 60,000 divided by the RPM multiplied by .75. 60000/RPM equals the dwell time for one complete revolution. Multiplying by .75 gives 3/4 revolution to relieve the pressure.

N1 O4 (BIG DRILL
Contains the program number and a comment
N2 L100
Subroutine number one
N3 G1 G91 F5. Z-R6
Sets incremental mode feed at 5. IPM to Z minus the value in register 6 (R6)
N4 G4 P+R4
Dwell for the amount of time indicated by the value in R4
N5 #V10=AZ
Enters the current Z position into V10
N6 #IF V10>V1 THEN GOTO N3
Tests for the current location of Z. If full depth has not been reached loop back to N3
N7 #V10=0
Reset V10 to zero
N8 G90 G1 Z.1F50.
Set absolute mode. Feed out to .1 above Z zero
N9 M17
N10 M30
Lines nine and ten end the subroutine definitions
N11 #CLEAR
The program starts here
This statement clears all of the values in the variable table
N12 R9-1. R8+1000. R7+.05 (CHANGE THIS LINE FOR NEW VALUES.
Sets the R values for the drill to be used
N13 #PRINT "R9=ABSOLUTE Z DEPTH"
N14 #PRINT "R8=RPM TO BE USED"
N15 #PRINT "R7=DIAMETER OF DRILL BEING USED"
Lines 13, 14 and 15 inform the operator what the R values are used for
N16 #V1=R9
Passes the value in R9 to V1
N17 #V2=R8
Passes the value in R8 to V2
N18 #R4=(6000/V2).75*
Calculates the Dwell time and stores it in R4

N19 #V3=R7
Passes the value in R7 to V3
N20 #V4=V3.3 ‘.3 = 30 PERCENT OF THE DRILL DIAMETER*
Calculates the peck distance
N21 #R6=V4
Passes the value in V4 to R6
N22 G90 G0 S+R8 M3 E1 X0 Y0
Sets the absolute mode, turns on the spindle and rapids to X0 Y0 of fixture offset #1
N23 H1 Z1. M8
Brings the spindle to 1. above Z zero and turns on the coolant
N24 Z.1
Brings the tool .1 above Z zero
N25 L101 G66
Calls subroutine L100 as a modal routine. From this point the subroutine will execute after every position move until canceled by a G67
N26 M45
Will execute the subroutine at this location
N27 X1.
Moves to X1. and executes the subroutine
N28 X2.
Moves to X2. and executes the subroutine
N29 G67
Cancels execution of the subroutine
N30 G0 G90 H0 Z0
Sets absolute and rapid modes. Cancels tool length offset and moves to Z0
N31 E0 X0 Y0
Cancels the fixture offset and moves to X0 Y0
N32 M2
End of program

18.13.5 PROGRAM NUMBER 5

This program will drill a circular bolt hole pattern as defined by the user. The Sine and Cosine functions are used to define the X and Y dimensions for the first hole. The program then rotates the hole an amount determined by the number of holes.

N1 O5 (BOLT PATTERN
Program number and comment
N2 #CLEAR
Clears all variable registers
N3 #PRINT “X0 Y0 IS THE CENTER OF THE PATTERN USING FIXTURE OFFSET #1"
N4 #PRINT “R5 IS THE NUMBER OF HOLES TO BE DRILLED”
N5 #PRINT “R4 IS THE DIAMETER OF THE HOLE PATTERN”

N6 #PRINT “R3 IS THE STARTING ANGLE”
Lines three through six tell the operator where everything is
N7 R5+10. R4+5. R3+30.
Defines the parameters for the part
N8 #R7=360/R5 ‘ANGLE BETWEEN HOLES
Sets R7 as the angle between holes. It is to be used in the G68 line to rotate the hole
N9 #V1=R4/2 ‘RADIUS OF THE BOLT PATTERN
Determines the radius of the bolt pattern for X Y calculations
*N10 #R9=SIN(R3)*V1 ‘X STARTING LOCATION*
*N11 #R8=COS(R3)*V1 ‘Y STARTING LOCATION*
N12 T1M6
N13 G90 G80 G0 S5000 M3 E1 X+R9 Y+R8
N14 Z5. H1 M8
N15 G81 X+R9 Y+R8 Z-1. R0.1 F10.G98
N16 #R6=R7
Set R6 equal to R7 as a value to increment the angle for each rotation
N17 #:LOOP
Label to loop to
N18 G68 R+R7 X0 Y0
Rotate the hole by the angle defined by R7
N19 #R7=R7+R6
Add to the angle for rotation (next hole)
N20 #IF R7<(360-R6) THEN GOTO :LOOP
Test for last hole
N21 G80 M5 M9
N22 G49 Z0
N23 G28 X0 Y0 Z0
N24 M2

18.13.6 PROGRAM NUMBER 6

This sub program probes two bores in line on the X axis. It then resets the fixture offset to the center of the left hand bore and rotates the coordinate system to align with the C/L of the bores. The left hand bore will be set to X0 Y0. The distance between bores is assumed to be 5". This dimension may be adjusted in line N14. Bores up to approximately 2.5" diameter may be checked. This value will be influenced by the probe diameter and may be changed by changing the X and Y values in the L9101 R1+1 lines.

N1 O6(SUB PROG TO CHECK BORE C/L AND ROTATE
Contains the program number and a comment
N2 G0 G90 G40 G49 G80 Z0
Safe start line
N3 X0 Y0 E1

Move to X0 Y0 at fixture offset 1
N4 Z0.1 H1
Moves the probe to .1 above Z zero
N5 G1 Z-0.25 F30.
Moves the probe .25 below Z zero
N6 M64
Turns on the probe
N7 L9101 R1+1. Y1.25 P1 F30.
Calls the probe routine to pick up the first point in the first bore
N8 L9101 R1+1. X-1.0825 Y-0.625 P2 F30.
Calls the probe routine to pick up the second point in the first bore
N9 L9101 R1+1 X1.0825 Y-0.625 P3 F30.
Calls the probe routine to pick up the third point in the first bore
N10 L9101 R1+2.
Calculates the diameter and center point of the first bore
N11 #V50=R1
Stores the X location of the first bore in V50
N12 #V51=R2
Stores the Y location of the first bore in V51
N13 G0 Z.1
Moves to Z.1
N14 X5. Y0
Moves to the estimated center point of the second bore
N15 G1 Z-0.25 F20.
Moves to Z-.25
N16 L9101 R1+1. X5. Y1.25 P1 F30.
Calls the probe routine to pick up the first point in the second bore
N17 L9101 R1+1. X3.9175 Y-0.625 P2 F30.
Calls the probe routine to pick up the second point in the second bore
N18 L9101 R1+1. X6.0825 Y-0.625 P3 F30.
Calls the probe routine to pick up the third point in the second bore
N19 L9101 R1+2.
Calculates the diameter and center point of the second bore
N20 #V60=R1
Stores the X location of the second bore in V60
N21 #V61=R2
Stores the Y location of the second bore in V61
N22 #V55=V60-V50
Calculates the X distance from the first bore to the second bore
N23 #V56=V61-V51
Calculates the Y distance from the first bore to the second bore
N24 #V57=ATN(V56/V55)
Calculates the angular rotation off of 0

N25 #FX1=FX1+V50
Sets fixture offset one X location to the center of the first bore
N26 #FY1=FY1+V50
Sets fixture offset one Y location to the center of the first bore
N27 #R9=V57
Sets R9 equal to the angular rotation of the second bore
N28 G49 Z0
Cancels tool length offset
N29 G0 X0 Y0 E1
Moves to X0 Y0 of fixture offset one
N30 G68 X0 Y0 R+R9
Rotates the coordinate system to align on the bores
N31 M99
Code to return to the main or calling program

18.13.7 PROGRAM NUMBER 7

This program will cut a rectangular pocket with tapered sides. The pocket will have been roughed to the bottom finish size. The angle of the sides in this example is five degrees and the Z step will be .01. The calculation for the X and Y axis step over is Tan 5 * .01 = the step over distance. The tangent point for the bottom of the pocket must be calculated for the start dimensions. If the pocket is through, the Z depth will be equal to the depth of the bottom of the pocket plus 1/2 the diameter of the ball nose end mill that is being used. The X Y shift amount would be equal to the radius of the cutter divided by the COS of the angle. EX: .1875/COS 5 .1875/.99619=.1882 would be the correction factor for a 3/8 end mill at 5 degrees. If the pocket is not a through pocket Z depth will be equal to the depth of the pocket minus 1/2 the diameter of the ball end mill. The X Y shift amount would use the same calculations as the through pocket plus a correction factor for the difference in depth, which would be TAN of the Angle times the Radius of the cutter. EX: TAN 5 * .1875 = .087488*.1875 = .0164. This is the depth factor. Subtract this from the Angle factor .1882 for the total correction factor. .1882-.0164=.1718.

N1 O7 (TAPER RECTANGLE
Contains the program number and a comment
N2 G0 G90 G80 G40 G49 Z0
Safe start line
N3 S7500 M3 M8
Spindle on 7500 RPM coolant on
N4 G8 M92
Set intermediate gain turn off ramping. This increases the accuracy and smoothes the moves
N5 G0 X0 Y0 E1
Move to the center of the pocket
N6 Z0.05 H1

Move to .05 above Z zero
N7 G1 Z-1.687 F50.
Move to the bottom of the pocket. The tangent point must be calculated for the X and Y locations
N8 R9+0.8736
Set R9 equal to 1/2 of the Y dimension at the bottom of the pocket plus the tangent point on the ball end mill

N9 R8+1.3556
Set R8 equal to 1/2 of the X dimension at the bottom of the pocket plus the tangent point on the ball end mill
N10 R7+1.687
Set R7 equal to the Z depth. The pocket depth plus 1/2 of the endmill diameter
N11 #:LOOP
Label to loop to for each step
N12 G1 Y-R9
Mill from the center to Y- dimension
N13 X+R8
Mill from the center to the X+ dimension
N14 Y+R9
Mill to the Y+ dimension
N15 X-R8
Mill to the X- dimension
N16 Y-R9
Mill to the Y- dimension
N17 X0
Mill to X0
N18 #R9=R9+.00087
Increment the Y dimension by .00087. The Tangent of the angle times the .01 Z step. In this case the angle is five degrees
N19 #R8=R8+.00087
Increment the X dimension by .00087. The tangent of the angle times the .01 Z step. In this case the angle is five degrees
N20 #R7=R7-.01
Increment the Z dimension .01
N21 Z-R7.
Move the Z axis up .01
N22 #IF R7 GT 0 THEN GOTO :LOOP
Test to see if the top of the part has been reached
N23 G0 Z.5 M5 M9
Rapid to Z.5 turn off the spindle and coolant
N24 G49 Z0 E0 X0 Y0 Z0
Cancel offsets and return to the home position

N25 M2
End of program

18.13.8 PROGRAM NUMBER 8

Sine Wave - A sine wave is constructed by laying out a line that represents one 360 degree revolution. In the case of a cam wrap, this would be equal to the circumference of the part to be cut. An example would be a 1 1/2" diameter that has a circumference of PI*dia=3.1416*1.5=4.7124. This line is then divided into equal segments determined by the for requirements on the blue print. Many segments give a high resolution and better form; few segments give a low resolution. Two degree segments are acceptable for most applications. Once the number of segments is determined, the line is divided by this number to obtain the Y axis step. Since our line is 360 degrees long we find the number of steps by dividing 360 by the angular increment, in this case two. 360/2=180 steps. We then divide the length of our line (4.7124) by the number of steps (180) 4.7124/180=.0262. This will be the amount of Y axis movement for each two degree increment on the Sine wave. The X axis movement is determined by multiplying the Sine of the accumulated angle by the height of the wave. The height of the wave will be half of the total X axis movement. If we require a total movement of 1.25 inches in the X axis our wave will be 1.25/2=.625.

Our formulas are: PI*Dia = Circumference, 360/Angular increment = Y axis move, Sine of the angle * Wave Height = X axis move

This program will ‘wrap’ a sine wave around a round part on a fourth axis.

N1O7(SINE WAVE DEMO
Program number and a comment
N2 L100
Sub routine label
N3 R1+0 R2+0 R3+0 R4+0 (ESTABLISH VALUES FOR R
WORDS
Zero R variables
N4 #V1=1.5 ‘DIAMETER OF THE PART
Enter the diameter of the part
N5 #V2=.625 ‘RISE OF THE SINE WAVE
Enter the rise of the sine wave
*N6 #V3=3.141593*V1 ‘CIRCUMFERENCE OF THE PART*
Calculate the circumference of the part
*N7 #V4=90/(5*V3) ‘CALCULATE THE Q WORD*
Calculate the Q word for Cam Wrapping
N8 #R1=V4 ‘TRANSFER Q WORD TO R1
Transfer the value of the Q word to R1
N9 G51.1 Y0 ‘SET Y AXIS MIRROR
Mirror the Y axis for Cam Wrapping

N10 G17 Q+R1 'TURN ON CAM WRAPPING
Turn on Cam Wrapping
N11 G90 F50.
Set absolute mode 50. IPM
N12 #V10=V3/180 'Y AXIS INCREMENT PER 2 DEG MOVE
Calculate the Y axis increment for each 2 deg
N13 #:LOOP
Label to loop to until finished
N14 #V12=V12+V10 'Y AXIS POSITION
Calculate the Y axis position
N15 #V13=V13+2 'ANGULAR COUNT
Increment the angle
*N16 #V14=SIN(V13)*V2 'X AXIS POSITION*
Calculate the X axis position
N17 #R4=V12 'Y AXIS PASS TO R4
Transfer the Y axis position to R4
N18 #R5=V14 'X AXIS PASS TO R5
Transfer the X axis position to R5
N19 G1 X+R5 Y+R4 F20. 'NEXT MOVE
Move to the next location
N20 #IF V13 LT 360 THEN GOTO :LOOP
Test if finished
N21 G0 Z1.
Move to clear part
N22 Y0
Unwrap move
N23 G50.1
Turn off Mirror
N24 G17
Turn off Cam Wrapping
N25 M17
N26 M30
End of subroutines
N27 G0 G80 G90 G40 G49 Z0
Safe start line
N28 T1 M6
Tool change
N29 S2500 M3 M8
Spindle and coolant on
N30 G0 X0 Y0 A0 Z0 E1
Move to position
N31 Z1. H1
Move to Z 1

N32 G1 Z0.65 F15.
Move to cut height. Z zero is the center of the part
N33 L101
Call Subroutine
N34 M5 M9
Turn off spindle and coolant
N35 G49 E0 X0 Y0 Z0
Cancel offsets
N36 G28
Go home
N37 M2
End of program

18.13.9 PROGRAM NUMBER 9

Ellipse - An ellipse is defined as a collection of points whose locations are the sum of the distances from two fixed points such that the sum of the distances is always equal. The formula is given as (X Sq/ a Sq)+(Y Sq / b Sq)=1 Where a is equal to the X radius of the ellipse and b is equal to the Y radius of the ellipse. By re-arranging the formula to solve for Y we obtain +- Y = Square Root of (b2 - ((X2 * b2) / a2)). Because the formula solves for plus or minus Y, we will need two loops to complete the ellipse, one for the Y+ and one for the Y-. The X axis will be incremented in .01 steps.

Formula: +-Y = +/- Square Root of (b2 - ((X2 * b2) / a2))

N1 O9 (ELLIPSE PROGRAM
Program number and a comment
N2 G0 G90 G80 G40 G49 Z0
Safe start line
N3 T1 M6
Call tool one
N4 S2500 M3 M7
Spindle on coolant on
N5 G0 X0 Y0 Z0 E1
Rapid to the middle of the part
N6 #:XDIM
Label to loop to if invalid data entered
N7 #PRINT “ENTER THE X DIMENSION”
Print statement asking the operator to enter data
N8 #INPUT V1
Input statement to accept the requested data
N9 #V1=V1/2
Calculates the X radius
N10 #IF V1 LE 0 THEN GOTO :XDIM

Tests for acceptable data
N11 #:YDIM
Label to loop to if invalid data is entered
N12 #PRINT "ENTER THE Y DIMENSION"
Print statement asking the operator for more information
N13 #INPUT V10
Input statement to accept requested information
N14 #V10=V10/2
Calculates the Y radius
N15 #IF V10 LE 0 THEN GOTO :YDIM
Tests for invalid information
N16 #V20=V1
Copies the value in V1 to V20
*N17 #V1=V1*V1*
Squares the contents of V1. This is the a value
*N18 #V10=V10*V10*
Squares the contents of V10. This is the b value

N19 #R9=V20
Transfers the X+ location to R9
N20 G0 X+R9 Y0
Moves to the start point on the Ellipse
N21 Z0.1 H1
Moves the tool to .1 above Z zero
N22 G1 Z-0.25 F20.
Feeds the tool to Z-.25
N23 #V25=V20
Copies the value of V20 into V25 for use as a counter
N24 #:LOOP
Label to loop to for each step
N25 #V20=V20-.01
Calculates the next X location
*N26 #V21=V20*V20*
Squares the X location for the X squared portion of the formula
*N27 #V30=(V21*V10)/V1*
Calculates the X squared times b squared divided by a squared portion of the formula
N28 #V31=SQR(V10-V30)
Calculates the square root of b squared minus V30 value
N29 #R9=V20
Copies the X location to R9
N30 #R8=V31
Copies the Y location to R8
N31 G1 X+R9 Y+R8 F20.

Moves to the next location
N32 #IF V20 > -V25 THEN GOTO :LOOP
Tests for the finish of the top portion of the Ellipse
N33 #:LOOP1
Label to loop to for the bottom portion of the Ellipse
N34 #V20=V20+.01
Reverses the X axis move
*N35 #V21=V20*V20*
Squares the X location for the X squared portion of the formula
*N36 #V30=(V21*V10)/V1*
First calculation of the formula
N37 #V31=SQR(V10-V30)
Final calculation of the formula
N38 #R9=V20
Copies the X location to R9
N39 #R8=V31
Copies the Y location to R8
N40 G1 X+R9 Y-R8 F20.
Moves to the next location
N41 #IF V20 < V25 THEN GOTO :LOOP1
Tests to see if the Ellipse is complete
N42 G0 Z0.1 M5 M9
Moves to Z.1 turns off the spindle and coolant
N43 G49 Z0 E0 X0 Y0
Cancels offsets returns to Zero
N44 G28
Returns home
N45 M2
End of program

18.14 TUTORIAL PROGRAM LISTINGS

18.14.1 PROGRAM NUMBER 1

N1 O99 (CHECK TOOL
N2 G90 G0 E24 X0 Y.25 S250. M4
N3 M65
N4 H1 E23 Z1.
N5 G1 G31 Z-.1 F20.
N6 G91 Z.05
N7 G90 S500
N8 G1 G31 Z-.1 F1.

```
N9 #R9=AZ
N10 #R8=R9-FZ23
N11 G10 L10 P1 R0+R8
N12 M5 G0
N13 G0 G90 H0 Z0
N14 E0 X0 Y0 Z0
N15 M99
```

18.14.2 PROGRAM NUMBER 2

```
N1 O2(SUB TO CHECK FOR TOOL BREAKAGE
N2 G90 G0 E24 X0 Y.25 S250. M4
N3 #R8=TN
N4 #H99=H(R8)
N5 #H99=H99+F23
N6 M65
N7 H99 Z1. (TOOL LENGTH ENTERED IN OFFSET #99 TO PROBE
N8 G1 Z-.1
N9 G1 G31 Y0
N10 #R9=AY
N11 G0 Y.25
N12 H0 Z0 G0 E0 X0 Y0
N13 #IF R9>0 THEN GOTO :EXIT
N14 M0
N15 #:EXIT
N16 M99
```

18.14.3 PROGRAM NUMBER 3

```
N1 03*TOOL TIME*
N2 G0 G90 E1 X0 Y0
N3 #V9=TN  'GET THE TOOL NUMBER
N4 #R9=V9
N5 #:LOOP
N6 M6 T+R9
N7 Z.1 H+R9
N8 G81 G98 Z-.1 R0+.1 F50. M45
N9 G91 X.5 L20
N10 G80 G90 M5M9
N11 X0 Y0
N12 #IF TU(R9)<TT(R9) THEN GOTO :LOOP
N13 #R9=R9+1
N14 #TU(R9)=0
N15 #IF R9<22 THEN GOTO :LOOP
N16 M2
```

18.14.4 PROGRAM NUMBER 4

```
N1 O4 (BIG DRILL
N2 L100
N3 G1 G91 F5. Z-R6
N4 G4 P+R4
N5 #V10=AZ
N6 #IF V10>V1 THEN GOTO N3
N7 #V10=0
N8 G90 G1 Z.1F50.
N9 M17
N10 M30
N11 #CLEAR
N12 R9-1. R8+1000. R7+.05 (CHANGE THIS LINE FOR NEW VALUES.
N13 #PRINT "R9=ABSOLUTE Z DEPTH"
N14 #PRINT "R8=RPM TO BE USED"
N15 #PRINT "R7=DIAMETER OF DRILL BEING USED"
N16 #V1=R9
N17 #V2=R8
N18 #R4=(6000/V2)*.75
N19 #V3=R7
N20 #V4=V3*.3     '.3 = 30 PERCENT OF THE DRILL DIAMETER
N21 #R6=V4
N22 G90 G0 S+R8 M3 E1 X0 Y0
N23 H1 Z1. M8
N24 Z.1
N25 L101 G66
```

```
N26 M45
N27 X1.
N28 X2.
N29 G67
N30 G0 G90 H0 Z0
N31 E0 X0 Y0
N32 M2
```

18.14.5 PROGRAM NUMBER 5

```
N1 O5 (BOLT PATTERN
N2 #CLEAR
N3 #PRINT "X0 Y0 IS CENTER OF PATTERN USING FIXTURE OFFSET #1"
N4 #PRINT "R5 IS THE NUMBER OF HOLES TO BE DRILLED"
N5 #PRINT "R4 IS THE DIAMETER OF THE HOLE PATTERN"
N6 #PRINT "R3 IS THE STARTING ANGLE"
N7 R5+10. R4+5. R3+30.
N8 #R7=360/R5   'ANGLE BETWEEN HOLES
N9 #V1=R4/2     'RADIUS OF THE BOLT PATTERN
N10 #R9=SIN(R3)*V1  'X STARTING LOCATION
N11 #R8=COS(R3)*V1  'Y STARTING LOCATION
N12 T1M6
N13 G90 G80 G0 S5000 M3 E1 X+R9 Y+R8
N14 Z5. H1 M8
N15 G81 X+R9 Y+R8 Z-1. R0.1 F10.G98
N16 #R6=R7
N17 #:LOOP
N18 G68 R+R7 X0 Y0
N19 #R7=R7+R6
N20 #IF R7<(360-R6) THEN GOTO :LOOP
N21 G80 M5 M9
N22 G49 Z0
N23 G28 X0 Y0 Z0
N24 M2
```

18.14.6 PROGRAM NUMBER 6

```
N1 O6(SUB PROG TO CHECK BORE C/L AND ROTATE
N2 G0 G90 G40 G49 G80 Z0
N3 X0 Y0 E1
N4 Z0.1 H1
N5 G1 Z-0.25 F30.
N6 M64
N7 L9101 R1+1. Y1.25 P1 F30.
```

```
N8 L9101 R1+1. X-1.0825 Y-0.625 P2 F30.
N9 L9101 R1+1 X1.0825 Y-0.625 P3 F30.
N10 L9101 R1+2.
N11 #V50=R1
N12 #V51=R2
N13 G0 Z.1
N14 X5. Y0
N15 G1 Z-0.25 F20.
N16 L9101 R1+1. X5. Y1.25 P1 F30.
N17 L9101 R1+1. X3.9175 Y-0.625 P2 F30.
N18 L9101 R1+1. X6.0825 Y-0.625 P3 F30.
N19 L9101 R1+2.
N20 #V60=R1
N21 #V61=R2
N22 #V55=V60-V50
N23 #V56=V61-V51
N24 #V57=ATN(V56/V55)
N25 #FX1=FX1+V50
N26 #FY1=FY1+V50
N27 #R9=V57
N28 G49 Z0
N29 G0 X0 Y0 E1
N30 G68 X0 Y0 R+R9
N31 M99
```

18.14.7 PROGRAM NUMBER 7

```
N1 O7 (TAPER RECTANGLE
N2 G0 G90 G80 G40 G49 Z0
N3 S7500 M3 M8
N4 G8 M92
N5 G0 X0 Y0 E1
N6 Z0.05 H1
N7 G1 Z-1.687 F50.
N8 R9+0.8736
N9 R8+1.3556
N10 R7+1.687
N11 #:LOOP
N12 G1 Y-R9
N13 X+R8
N14 Y+R9
N15 X-R8
N16 Y-R9
N17 X0
```

```
N18 #R9=R9+.00087
N19 #R8=R8+.00087
N20 #R7=R7-.01
N21 Z-R7.
N22 #IF R7 GT 0 THEN GOTO :LOOP
N23 G0 Z.5 M5 M9
N24 G49 Z0 E0 X0 Y0 Z0
N25 M2
```

18.14.8 PROGRAM NUMBER 8

```
N1O7(SINE WAVE DEMO
N2 L100
N3 R1+0 R2+0 R3+0 R4+0 (ESTABLISH VALUES FOR R WORDS
N4 #V1=1.5     'DIAMETER OF THE PART
N5 #V2=.625     'RISE OF THE SINE WAVE
N6 #V3=3.141593*V1 'CIRCUMFERENCE OF THE PART
N7 #V4=90/(5*V3)   'CALCULATE THE Q WORD
N8 #R1=V4         'TRANSFER Q WORD TO R1
N9 G51.1 Y0   'SET Y AXIS MIRROR
N10 G17 Q+R1   'TURN ON CAM WRAPPING
N11 G90 F50.
N12 #V10=V3/180    'Y AXIS INCREMENT PER 2 DEG MOVE
N13 #:LOOP
N14 #V12=V12+V10    'Y AXIS POSITION
N15 #V13=V13+2      'ANGULAR COUNT
N16 #V14=SIN(V13)*V2 'X AXIS POSITION
N17 #R4=V12        'Y AXIS PASS TO R4
N18 #R5=V14         'X AXIS PASS TO R5
N19 G1 X+R5 Y+R4 F20. 'NEXT MOVE
N20 #IF V13 LT 360 THEN GOTO :LOOP
N21 G0 Z1.
N22 Y0
N23 G50.1
N24 G17
N25 M17
N26 M30
N27 G0 G80 G90 G40 G49 Z0
N28 T1 M6
N29 S2500 M3 M8
N30 G0 X0 Y0 A0 Z0 E1
N31 Z1. H1
N32 G1 Z0.65 F15.
N33 L101
```

```
N34 M5 M9
N35 G49 E0 X0 Y0 Z0
N36 G28
N37 M2
```

18.14.9 PROGRAM NUMBER 9

```
N1 O9 (ELLIPSE PROGRAM
N2 G0 G90 G80 G40 G49 Z0
N3 T1 M6
N4 S2500 M3 M7
N5 G0 X0 Y0 Z0 E1
N6 #:XDIM
N7 #PRINT "ENTER THE X DIMENSION"
N8 #INPUT V1
N9 #V1=V1/2
N10 #IF V1 LE 0 THEN GOTO :XDIM
N11 #:YDIM
N12 #PRINT "ENTER THE Y DIMENSION"
N13 #INPUT V10
N14 #V10=V10/2
N15 #IF V10 LE 0 THEN GOTO :YDIM
N16 #V20=V1
N17 #V1=V1*V1
N18 #V10=V10*V10
N19 #R9=V20
N20 G0 X+R9 Y0
N21 Z0.1 H1
N22 G1 Z-0.25 F20.
N23 #V25=V20
N24 #:LOOP
N25 #V20=V20-.01
N26 #V21=V20*V20
N27 #V30=(V21*V10)/V1
N28 #V31=SQR(V10-V30)
N29 #R9=V20
N30 #R8=V31
N31 G1 X+R9 Y+R8 F20.
N32 #IF V20 > -V25 THEN GOTO :LOOP
N33 #:LOOP1
N34 #V20=V20+.01
N35 #V21=V20*V20
N36 #V30=(V21*V10)/V1
N37 #V31=SQR(V10-V30)
```

```
N38 #R9=V20
N39 #R8=V31
N40 G1 X+R9 Y-R8 F20.
N41 #IF V20 < V25 THEN GOTO :LOOP1
N42 G0 Z0.1 M5 M9
N43 G49 Z0 E0 X0 Y0
N44 G28
N45 M2
```

19.0 MISCELLANEOUS

19.1 FLOOD COOLANT THROUGH THE SPINDLE (OPTIONAL)

19.1.1 DESCRIPTION

The Coolant Through the Spindle option provides coolant flow through the spindle. The system has its own pump and electrical system, separate from the standard flood coolant system. The main power switch for this system is located in the rear of the machine and is labeled “Coolant Through Spindle.” The coolant through main disconnect switch must be in the on position to activate the coolant through system.

The SETP command allows the operator to select M7 or M8 for flood coolant. When M7 is selected for flood coolant, M8 is used to activate the coolant through the spindle option from the program. When M8 is selected for flood coolant M7 is used to activate the coolant through the spindle option from the program. The M9 code will cancel the M7 or M8 codes.

Special hollow retention knobs need to be fitted into the ends of tool holders, which are intended for use with special tools that allow for coolant through the tool. Because these retention knobs are hollow, their manufacturer recommends usage with tools that have no side cutting pressure. The thin side walls of these special retention knobs may fracture when using side cutting tools like end mills, fly cutters, face mills etc. When the retention knobs fracture and break, the tool will fall out of the spindle.

NOTE
Tools with side cutting pressure require solid retention knobs. Special filters may be required to be retrofitted to the system if micro sized grit is in the coolant. The grit can damage the rotating seals in the coolant through system. If this seal is destroyed, the coolant will leak out of the seal and damage the belt or other systems in the head area. These filters would be supplied and provided by the user as to suit their particular need.

If the coolant through the spindle system is activated when a tool is in the spindle, which has a solid designed retention knob, pressure will <u>build up</u> behind the retention knob. When the solid tool holder is removed from the spindle, either by hand of from a tool change, the pressurized coolant will spray out from the flange area. It is suggested to used the tool changer, with the doors closed, to remove a tool from the spindle when the coolant through the spindle system is pressurized.

NOTE
Always wear eye protection when operating the machine and all of its added equipment.

19.2 HYDRO SWEEP (OPTIONAL)

19.2.1 HYDRO SWEEP™ CHIP REMOVAL SYSTEM

The Hydro Sweep™ option is used to clear machined part chips from the machine tool. This system uses a flood coolant wash to transfer the chips to a recycling container.

The Hydro Sweep™ system is comprised of flood wash nozzles, a flood coolant tank, a rotating chip drain drum, and a chip ejector shoot. The flood wash nozzles are prepositioned to provide maximum wash capabilities. The wash flood is divided into three sections, the left side, the right side, and the cross wash.

The rotating drum is driven, on rollers, by a motor. The rollers are belt driven by a standard 110 vac motor. The motor is powered by the machine tool. The electrical plug for the motor is mounted to the rear of the VMC and marked “Conveyer”. The drum is activated by pressing the CYCLE START button on the Hydro Sweep™ control panel. The drum may also be plugged into a standard wall socket. This provides the drum motor with power without turning on the flood wash.

The flood system uses the machine coolant for the flood wash. The coolant tank capacity is 70 gallons (VMC 4020) and 85 gallons (VMC 6030) to provide an adequate supply for both the coolant system and the wash.

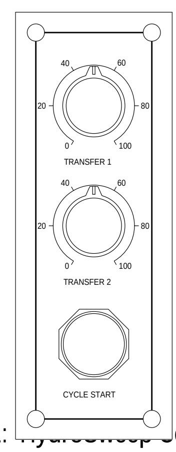


Figure 19-1: HydroSweep Control Panel

19.2.2 OPERATION

The Hydro Sweep™ is operated using the Hydro Sweep™ control panel. The control is mounted on the front of the VMC under the control pendant. The flood wash system is activated by pressing the CYCLE START button on the control panel. It may also be activated by the M20 code in the machine program.

The CYCLE START and M20 code are toggles for the system. Either can turn the system on or off. Press the button once to active the system and press it again to deactivate the system. The M20 operates the same way. The first time the code is performed in the program, the system is activated. The second time it is performed the system is deactivated. When the system is used, by either of these methods, the flood wash and drum are activated or deactivated. To activate the drum without the flood wash, the drum motor MUST be plugged into another power source.

> NOTE
> When an alternate power source is used, the drum is NOT activated by the CYCLE START or M20.

The knobs marked TRANSFER 1 and TRANSFER 2, are used to set the flood time. The TRANSFER 1 knob controls the cross wash. The TRANSFER 2 knob controls the left and right wash. The flood has four cycles. The cycles rotate in sequence from the cross wash, to the left wash, to the cross wash, to the right wash. This sequence is continual until the CYCLE START button is pressed. Only one wash cycle is on at any time. As one cycle starts the other stops. The flood wash duration for each cycle position is determined by the TRANSFER knob settings. The knobs are adjustable from 0 to 100 percent. With the knob at 0 the wash cycles for three seconds. At 100 percent the wash cycles for thirty seconds. The rotating drum lifts the chips to a drainage tray. Chips are moved from the drainage tray to a recycling bin by a ram. This ram moves at preset intervals when the system is activated.

19.3 AUTOMATIC DOORS (OPTIONAL)

19.3.1 DESCRIPTION

The Automatic Door option may be operated using the M80, (open), and the M81, (close). The doors ONLY operate with the M functions when they are in the interlocked position. The interlocked position is when the doors are locked into place on the pneumatic valves. When the doors are in the interlocked position, the M80 code activates the pneumatic valves to open the doors. When in the open position, the doors may NOT be closed manually. When in the closed position, the door interlock may be manually disengaged to open and close the doors. The door interlocks may be manually released by opening the doors by hand. This requires slightly more pressure than without the interlocks. The interlocks may be engaged manually by pulling the doors closed, by hand, until the interlock engages. The interlocks must be engaged manually to open the doors with the pneumatic valves.

> NOTE
> An M20 code is used to close the doors when the machine tool is not equipped with a general indexer.

NOTE
When the doors are closed and the M20 is coded, the machine will wait for a signal. To override this condition, open and close the doors manually.

NOTE
The M20 code is ONLY used when the machine is NOT wired for a general purpose indexer. Use the M81 code when there is an indexer.

The doors will come to a complete closed position before the program will continue. When a general purpose indexer is used, the doors will close with an M81.

19.3.2 M80 AUTOMATIC DOORS OPEN

This function is used to open the automatic doors. When the doors are in the interlocked position, this code activates the pneumatic valves to open the doors. When in the open position, the doors may NOT be closed manually. When in the closed position, the door interlock may be manually disengaged to open and close the doors. The interlocks must be engaged manually to open the doors with the pneumatic valves. The interlocks may be engaged manually by pulling the doors closed, by hand, until the interlock engages.

19.3.3 M81 AUTOMATIC DOORS CLOSE

This function is used to close the automatic doors. When the doors are in the interlocked position, this code activates the pneumatic valves to close the doors. When in the open position, the doors may NOT be closed manually. When in the closed position, the door interlock may be manually disengaged to open and close the doors. The interlocks must be engaged manually to open the doors with the pneumatic valves.

NOTE
When the doors are closed and the M81 is coded the machine will wait for a signal. To override this condition, open and close the doors manually.

NOTE
This function is ONLY used when the machine is wired for a general purpose indexer. Use the M20 code when there is no indexer.

19.4 SERVO COOLANT (OPTIONAL)

19.4.1 DESCRIPTION

The Servo Coolant is an option that requires the appropriate software version and hardware. The M7.1 or M8.1 activates the Optional Servo Coolant and Coolant one

(Flood). The SETP command allows the operator to select M7 or M8 for flood coolant. The H word must be specified before programming the M7.1 or M8.1.

The servo coolant option has been designed to allow the user to program a direction of coolant flow. This coolant flow may be directed onto the tool or directly onto the part to cool and wash away chips.

The servo coolant will be assigned to the A axis. If a rotary table is on the VMC, it will have the A axis and the servo coolant will then have the B axis. When in JOG mode, the servo coolant may be moved by jogging in the A or B axis (see SETP page, set parameter for A or B ratio option to COOLANT).

19.4.2 THE CS PROCEDURE

When the VMC is powered on the servo coolant nozzle will find its own COLD START zero position by seeking the spindle housing and backing off 10.0 degrees. The nozzle is moved towards the spindle until the movement is obstructed. When the user COLD STARTS the VMC the Servo Coolant will again seek the alignment position. At this point the nozzle is moved back by 10 degrees and Cold Start position is set. The range of motion is limited to 0 to 45 degrees from this CS position. The Cold Start will time out after 15 seconds. The external override potentiometer knob must be straight upward in the OFF position before Cold Starting the VMC.

The Servo Coolant will also have an external override potentiometer knob located on the right side of the doors on the Chip guard. When this override is straight up, the display will show OFF. If the knob is rotated, the amount overridden will be displayed to the right of the position display. This knob will be active after CS for all modes of operation and may be locked in or out with the M48.1 and M49.1 codes.

When the control is in AUTO or MDI the position display will be as follows:

X 0.0000   0
Y 0.0000   0   A0   OFF
Z 0.0000   0

The Axis Position Display is modified to allow display for override pot. The numeric display to the right the position display is now replaced by the value of the override pot. This display will be in degrees (A or B axis only). This display is active after Cold Starting the machine providing the servo coolant ratio is selected in the SETP parameter page. Override pot should be in OFF position at CS.

19.4.3 SETUP PROCEDURE

1. Return the nozzle to the HOME position by using the HO command or by using JOG mode to jog the nozzle back to the CS zero position. The override potentiometer must be in the off position.

2. From MDI, call the first tool to be used with the Servo Coolant. With this tool in the spindle turn on the coolant. Next position the coolant flow onto the **tip** of the tool by using the manual Pulse Generator. Adjustment may be made later for the nozzle to point to the outer diameter of the tool by using a P word and diameter value.

3. Note the degree value on the display. This will be the value for the L WORD on the M7.1 line for the first tool. No negative position may be programmed for the Servo Coolant.

4. Place this information in the program to use the Programmable Servo Coolant.

In addition to turning on the coolant, the nozzle may be directed by programming the following parameters:

L = Degrees to point to the first tool programmed. The L word initializes the coolant nozzle angle.

P = Optional diameter / radius of tools in program. This makes an adjustment outward or inward.

Q = Optional length override. This is to modify the nozzle adjustment up or downward. The calculation is normally made from the value in the tool length offset table.

For example, if Tool 3 needs the nozzle to be adjusted upward, the calculation must be changed. To change the calculation the M7.1 line must include a Q word. If the current value in the tool table offset 3 is -22.39 by adding one inch to this value, the Q word will be -23.39. The program will now use the Q word instead of the H word value to aim the programmable coolant upward 1 inch. This change now adjusts the original setup parameter programmed for the first tool. The next H word / M7.1 combination will return to the original programmed setting. The Q word does not affect the tool length offset or change any values in the tool table.

The program line will read as follows:

N124 H3 D3 Z1.5 The tool length offset is -22.39
N125 M7.1 Q-23.39

> NOTE
> M48.1 and M49.1 enables and disables the override pot. This is to allow maximum movement to the nozzle even when the override pot is not at zero.

EXAMPLE:

N1 O1 (PART 1234
N2 M6 T1 (TOOL #1 , 3 INCH SHELL MILL
N3 G0 G90 S3200 M3 E1 X0 Y0
N4 H1 D1 Z.1
The H word must be called before the M7.1 code

N5 M7.1 L10.0 P3.0
Servo coolant on, Flood coolant on M7.1Read length of tool 1 and place nozzle at the L of 10 degrees from CS position this offset now sets the origin of the nozzle for this program. All nozzle aim will be adjusted from here.
******cut part*******The P moves nozzle outward to the edge of inserts on the 3.0" shell mill*
N22 M6 T2 (.25 END MILL
N23 G0 G90 S3400 X0 Y0
N24 H2 Z1.5
N25 M7.1 Servo coolant on, Flood coolant on M7.1
******cut part*******Read length of tool 2 and point nozzle to center tip of tool 2*
N98 M9 Coolant off
N122 M6 T3 (.5 END MILL
N123 G0 G90 S3400 X0 Y0
N124 H3 D3 Z1.5
N125 M7.1 P.5 Q-23.39
*Servo coolant on as defined for the first tool called in program. Read length of tool 3 and move nozzle to center of tip of tool 3. The P changes*****cut part*******The Q-23.39 now moves the length upward 1.0 " to adjust preference*
N1100 M9 Coolant off

19.4.4 CONTINUOUS SWEEP MODE FOR THE SERVO COOLANT

Use of the reciprocation with the servo coolant will continuously sweep the nozzle across the tool and part clearing / cooling the entire tool and part.

M14 Reciprocation for the B (Continuous sweep mode -Servo coolant option)

M15 Reciprocation for the A (Continuous sweep mode -Servo coolant option)

Use of the reciprocation with the servo coolant will continuously sweep the nozzle across the tool and part. The servo coolant nozzle has a axis range of 0 to 45 degrees only.

EXAMPLE:

G90 A10. X-3.0 M7
G1 F1000.0 A15.
M15
X6.0 G1 F25.0 The smaller the feedrate, the more times A will reciprocate
M10

19.5 CHIP AUGER AND WASH DOWN (OPTIONAL)

19.5.1 DESCRIPTION

The chip auger and wash down option is used to clear machined part chips from the machine tool. This system uses a flood coolant wash to transfer the chips to the front of the machine where a mechanical screw (auger) removes the chips.

The chip auger and wash down system is comprised of flood outlets, a flood coolant tank, a rotating mechanical auger and a chip shoot. The flood wash outlets are pre-positioned to provide maximum wash capabilities. The flood is divided into two sections: the right side and the left side.

The chip auger is driven by a shaft that is pinned to the auger. The shaft is driven by a ¾ HP standard 240 VAC 3 phase motor with a ninety degree angle head adapter. The motor is powered through the machine. The electrical plug for the motor is mounted at the bottom of the CNC Box. The flood pump is also a standard 240 VAC 3 phase motor and the electrical plug is mounted at the bottom of the Junction Box.

The wash system uses the fluid in the coolant tank. The coolant tank capacity is approximately 70 gallons to provide an adequate supply for both the coolant system and the wash. The coolant tank has a set of baffles (screens) inside the tank to filter out chips that get past the bottom pan screen. Therefore, the coolant tank will have to be cleaned periodically to remove any accumulation of chips inside the coolant tank.

19.5.2 OPERATION

The chip auger and wash system can run together (Standard Mode) or the wash separately (Wash Only Mode). The normal operating mode is with the switch in the Auger Wash mode. A control box mounted on the front of the machine (underneath the pendant control) has a two position switch to select the desired mode, Auger Wash or Wash Only. In both conditions, the side door panels must be installed and the front doors closed for the system to operate. The coded magnetic interlocks on the side panels and front doors must be in their closed state (normal operation when the front doors are closed and the side panels are installed).

Figure 19-2: Control Box

Figure 19-3: Coded Magnetic InterLocks

The chip auger wash system is activated and deactivated by the lower button on the control panel (labeled Start/Stop). The button is pushed once to activate the auger wash system and pushed again to deactivate the auger wash system. Either mode, Auger Wash or Wash Only, can be stopped immediately by opening the front doors or removing one of the side panels.

The chip auger/wash system cycle can also be activated by programming M20. Programming M20 again turns off the chip auger/wash system.

In the auger mode (Auger Wash), the auger continuously runs and the wash system cycles. The wash cycle time is a fixed off cycle time (time between when the wash shuts off and when the next wash cycle starts). Both sides of the machine are washed at the same time. The wash duration time (on time) can be adjusted by the cycle wash control (middle knob on the control box panel). The shortest duration time occurs with the knob rotated fully counterclockwise. The longest duration time is when the wash cycle time knob is fully rotated clockwise.

In the wash only mode, the auger does not operate and the wash system operates normally. The wash cycle time can be manipulated to adjust the wash duration (on/off time) by rotating the knob either clockwise or counter clockwise, respectively.

The auger operates automatically when the Start/Stop button is activated. The auger will move the chips across the bottom of the machine as the wash system washes the chips into the auger. The auger pushes the chips out of the push-out shoot where they are collected for removal. A standard 55-gallon drum can be placed on a pallet underneath the chip shoot to collect the chips.

If a jam condition occurs, the auger motor will reverse to clear the jam condition. The auger reversal is controlled by an overload current condition. The current supplied to the motor is monitored through the auger board. When a jam condition occurs and the supply current to the motor exceeds the current limit, the auger motor will reverse. The auger will reverse approximately 3-5 turns and then will resume normal operation. If the overload condition still exists (the reversal did not clear the jam the first time), the auger will repeat the reversal/forward clearing procedure up to 5 times per minute.

If the condition still exists, the auger board will trip and the obstruction will have to be cleared manually, the auger board reset (reset switch on auger board), and the normal operation resumed.

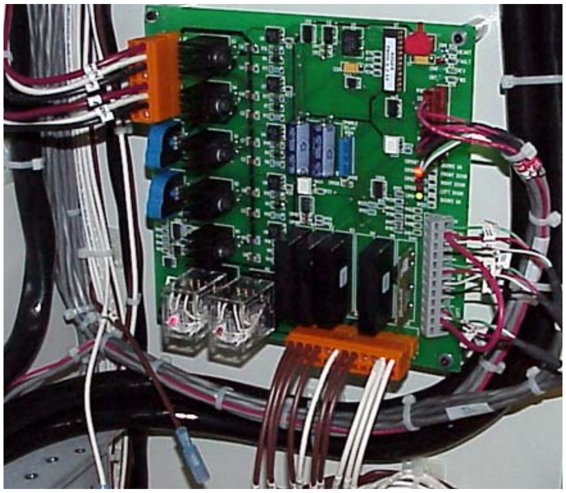


Figure 19-4: Auger Board

19.6 DUAL ARM TOOL CHANGER

19.6.1 COMMANDS

19.6.2 DD

Displays the bucket number and the 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 bucket ready position.
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.

19.6.3 SETTO

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 the 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.

19.6.4 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 least 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 Motor 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.

19.6.5 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 M6T3, 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.

19.6.6 OPERATING THE DATC FROM A PROGRAM

The commands used within a program to operate the Dual Arm Tool Changer are the same codes as the Geneva and Servo-Turret toolchangers. The call for a tool change can take advantage of the ability of the DATC to pre-stage the next tool into the bucket ready position. This is accomplished by commanding Txx and M6 independently. (Do not use T-xx as in previous versions.)

19.7 MISCELLANEOUS CONTROL OPTIONS

19.7.1 INPUT FROM A TAPE PUNCH OR COMPUTER

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

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

Option 1:

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

Option 2:

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

Option 3:

For # type 3. This option is used when the program to be transmitted is from another control. The FADAL CNC 88 will accept the program but may not be able to run the program in its present form. Some of the coding may have to be edited to the coding format required by the FADAL CNC 88.

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

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

19.7.2 KEY LOCK

The KEY LOCK on the pendant is used to protect the program from unauthorized editing. When any of the commands that can alter the program are used, the message “COMMAND PROHIBITED BY KEY LOCK” appears.

To prohibit program editing:

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

To allow program editing:

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

19.7.3 NOEDIT FEATURE

On the line with the O word type a comment with the word NOEDIT within the first sixteen characters, and this will prevent any changes to the program.

EXAMPLE: *N1 O6 (NOEDIT FACE PLATE*

19.8 ANALYZER SOFTWARE

19.8.1 DESCRIPTION

Analyzer software is a DOS-based program written by FADAL for a user that machines surfaces on the Fadal VMC. The VMC must be equipped with 1010-5 axis controller boards and the program must have been written to machine a part path over 50 ipm. Machining at 1000 BPS (blocks per second) requires machining at high feed rates. When a normal DNC program uses high feed rates, the movement may violently shock the slides during axis reversals. The intention of the FADAL Analyzer software is to smooth high-speed motion during reversals. This will utilize the same basic feed-forward functions of the M94 (severe direction changes) and M94.1 (incremental direction changes) codes. The Analyzer was written for a DOS-based computer, for the distinct advantages of off-line computing. The most important is the computational requirements (10,000 block look-ahead) of this program and the potential size of data to be processed.

19.8.2 ADVANTAGES OF ANALYZER

1. CAD systems generally do not address severe or incremental direction changes. This results in sudden drops or rises in feeds that can cause banging or marks. The Analyzer looks ahead thousands of moves and computes effective points at which to smooth feed changes.
2. Smoothing feeds is a one-time process.The Analyzer assumes this duty so that the CNC can spend time machining, not recomputing what it is going to machine, each time a part is machined. This results in significantly faster DNC rates then conventional methods.

19.8.3 USING THE ANALYZER

The Analyzer software converts and performs a binary compression of a program into a file that the DNC can run at 1000 BPS.

Simple Rules:

1. G91.3 starts the process.
2. G91.2 cancels (Format 2).
3. The high-speed moves must be absolute.
4. The block before the G91.3 must contain an absolute X, Y, and Z positioning move (contouring moves only).
5. Only high-speed moves between a G91.3 and G91.2 will be analyzed and processed.
6. A & B moves are currently not allowed.

The Analyzer displays 3 different views of the part (upper left: side view; upper right: front view; lower left: top view). The part must be scaled to display properly. Scale the part the first time you analyze it.

Scaling scans the program to check the maximum dimensions and far left corner position. The values generated by scaling may be altered. Scaling only affects the graphics display and does not change the analyzing process in any way.

Axis reversals are called “Severe Direction Changes.” The greater the angle chosen, the more severe the direction change must be to be smoothed.

Incremental direction changes are smoothed by "Angular Reduction." A severe angle broken up over many moves will cause a reduction.

The feedrate of the tool is controlled by the size of the high-speed moves; the larger the moves, the faster the feed. For example, if the maximum feed is 150 ipm, the maximum move will be .0025 inches. (Feed = maximum move * 1000 BPS * 60 seconds). During incremental direction changes, the maximum move will be reduced .0024, .0023, .0022, etc. depending upon the total angular change and the distance over which this change is made.

Where the tool is moving quickly, the color will be blue. Where the tool is slowing for an incremental direction change, the color may be green or red. Where the tool makes a severe direction change, the color will be red.

Long moves (those moves much longer than the maximum move size) will be broken into moves equal to or smaller than the maximum move size. For example, a 1 inch Z position move will begin red, then change green then blue as the tool gets up to speed. When approaching a severe direction change, the maximum move will reduce through green to red.

19.8.4 SAMPLE FILE: DATA.ABS

Included with the Analyzer is the sample program, DATA.ABS. Changing the parameters on this program will help you understand how to use the Analyzer effectively.

Each program you analyze will have a .PAR parameter file, saving the parameters you last used on that program. This allows you to optimize the parameters for each program, accounting for different tolerances and needs, without having to try to remember the settings you used last.

19.9 CNC88 HS OPTIONAL DISKETTE DRIVE

19.9.1 SPECIFICATIONS

3½" diskette medium.

1.44 Megabyte capacity; will read/write 720K diskettes.

MS-DOS format.

19.9.2 ACCESS FROM CONTROL

The drive is accessed from the Functions menu on the CNC by selecting option 0-DISK. A submenu is displayed with the following options:

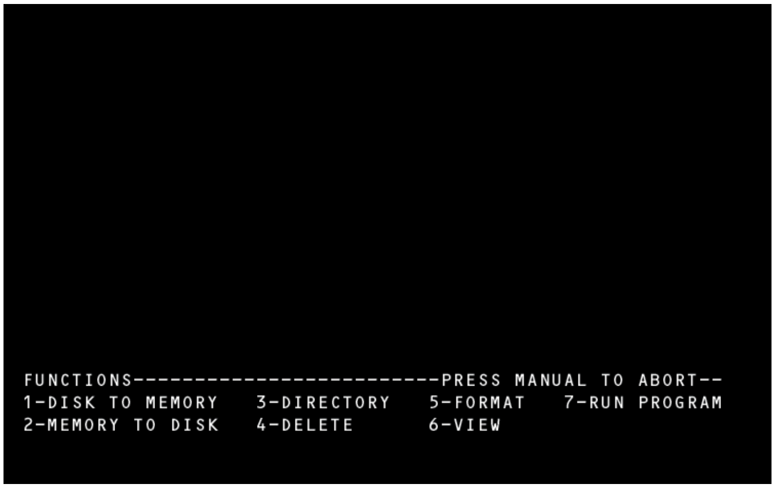


Figure 19-5: 0-Disk Submenu

Disk to Memory	*Loads a program from a diskette into CNC memory.*
Memory to Disk	*Saves a program from CNC memory onto a disk.*
Directory	*Displays the files and directories on a disk.*
Delete	*Deletes a file from the diskette.*
Format	*Formats a diskette (all files on diskette are deleted).*
View	*Displays the contents of a file from a diskette (one page at a time) on the CNC monitor.*
Run DNC	*Runs the VMC from a file on a diskette using DNC.* *Options are:* *1) Error checking* *2) Dry run* *3) Mid-program start by block #*

> NOTE
> The % character is not needed for running DNC from a diskette, but can be included in the file on the diskette.

19.9.3 ACCESS FROM PROGRAM USING MACRO STATEMENT #DISK

Macro language statement #DISK invokes a program on disk from within a program in the CNC memory. After processing the file from disk, control is routed back to the line of the program in memory that follows the #DISK statement.

EXAMPLE:

Macro Statement Format:

#DISK filename

EXAMPLE:

N1 G90 G0 S10000 M3 E1 X0 Y0
N2 H1 Z.1 M7
N3 #DISK CAVRUF This calls the disk file named CAVRUF and runs it relative to Fixture 1
N4 G0 Z.1   Program execution continues with this line, positioning the head to Z.1 after CAVRUF finishes running the first time
N5 E2 X0 Y0   This positions to X0 Y0 of Fixture 2
N6 #DISK CAVRUF This calls the disk file named CAVRUF and runs it again, this time relative to Fixture 2

Restrictions on File Content and Format:

The file needs to be in text file format, without any formatting characters or codes, such as those a Word Processor might use. In addition, the following guidelines apply:

1. The file needs to consist of standard CNC program words, similar to what would be typed or transmitted into the memory of the machine.
2. The file must not contain an O word.
3. The file should not contain an M30, M99, or an M2. The CNC will return to the program in memory automatically at the end of the file.
4. The lines can be numbered, but do not need to be.
5. The file itself cannot contain any #DISK statements.

> NOTE
> The % character is not needed for running a program from disk, although it can be included in the file.

19.9.4 RESTRICTIONS ON USER-DEFINED SUBROUTINES

The file executes in the CNC memory just as if it were a part of the program that executes the #DISK statement, with the following restrictions on user-defined subroutines.

1. The program on the disk cannot define a user-defined subroutine.
2. The disk file cannot call a user-defined subroutine.

19.9.5 SUBPROGRAMS AND FIXED SUBROUTINES ARE ALLOWED

Subprograms and fixed subroutines can be called from the disk file. Use the standard subprogram call format: M98 P# (where # is the subprogram to call). After the subprogram has executed, the program returns to the line after the subprogram call.

For example, L100 might be used in memory to define subroutine one, but this cannot be accomplished in the disk file.

Table 19-1: Diskette Drive Error Codes

CODE	DESCRIPTION
1002	FILE ALREADY EXISTS: Attempt made to define file with file already in directory.
1003	FILE DOES NOT EXIST: File selected for reading or deletion not in directory.
1004	ILLEGAL COMMAND: Command not recognized.
1005	DISKETTE FULL: Diskette has no room for transmitted data, or directory has no room for another entry.
1009	COMMUNICATIONS ERROR: Check character error detected, or other software or hardware communications problem.
1100	DISK READ ERROR: Unable to read directory without error, usually caused by unformatted diskette.
1101	BLOCK CHECK ERROR: Four attempts failed to read diskette block without error.
1103	SEEK ERROR: Track could not be verified while reading, or before writing, data block.
1104	DISKETTE TIMEOUT: Diskette could not be accessed, usually due to no diskette in drive, or diskette not fully inserted in drive.
1105	DISK WRITE PROTECTED: Write attempted on write-protected diskette.

20.0 GRAPHICS

20.1 CNC 88HS GRAPHICS

20.1.1 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 pressed while plotting is taking place.

20.1.2 PLOTTING OPTIONS

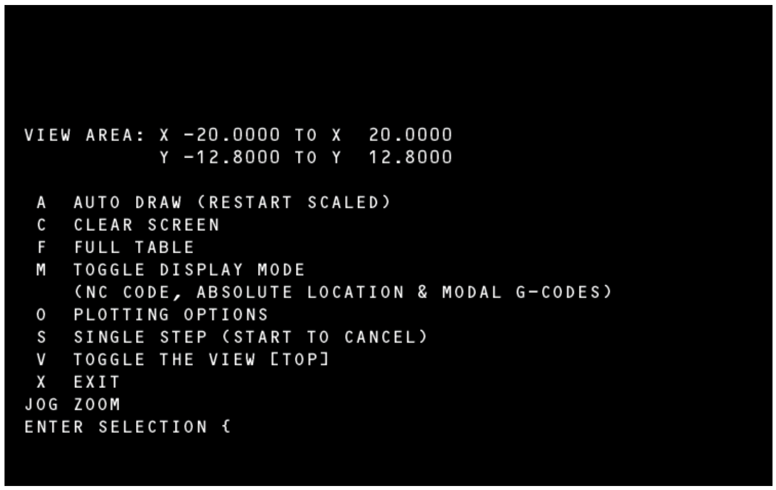


Figure 20-1: Plotting Options Menu

A = AUTO

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

C = CLEAR

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

F = FULL TABLE

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

M = TOGGLE DISPLAY MODE

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

O = OPTIONS PLOTTING

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

I Ignore G41 G42 compensation

L Plot only subroutines

O Plot only subprogram

P Plot total program

T Plot tool

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

S = SINGLE STEP

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

V= VIEW TOP OR ISOMETRIC

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

JOG = ZOOM

During the plotting process, or after or during the 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 functionthe part path the user wishes 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.

The Graphics screen will display a position located in the upper left hand portion. Located to the right of the position is a + (or -) and a value. This value is the Pixel resolution. The further the user ZOOMs into the graphically displayed part the smaller the Pixel resolution will become.

The ZOOM box may be decreased in the Z to show a cross hatch. The user may position this crosshatch using the X and Y. The position display will show a value and this value’s tolerance depends on the Pixel amount displayed.

20.2 FUNCTION MENU

20.2.1 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. Also it is designed for creating TOOL CALL or END OF PROGRAM coding and for defining FIXED CYCLES or SUBROUTINES.

20.2.2 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 the incorrect data is removed. When all the data has been entered, press the C key to compute the geometry.

20.2.3 GETTING STARTED

The user should always be aware of what position the cursor is at in the current program. 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 written to indicate which function was used.

20.2.4 THE MENUS

Once in the function tile listing, select the number of the function titles until you arrive at the individual function menu. The cursor is used to locate 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 (UP) to move the cursor upward to the data then use the Backspace button to back over the information. Retype the data.

When all data has been successfully enter press the C key to compute. The geometry will automatically be computed and displayed at the bottom portion of the screen. Pressing the D key (DRAW) will enlarge the graphics 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 expected solution is found.

When the solution is accepted, the data may be entered 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, speed and Z milling values. View the program on the Graphics display before machining. Dry run the program before cutting the part.

20.2.5 COORDINATE SYSTEM

The function menu uses the X Y plane (G17) of the cartesian coordinate system. All point descriptions are in absolute mode(G90), where all points are described as their distance from HOME (X0,Y0).

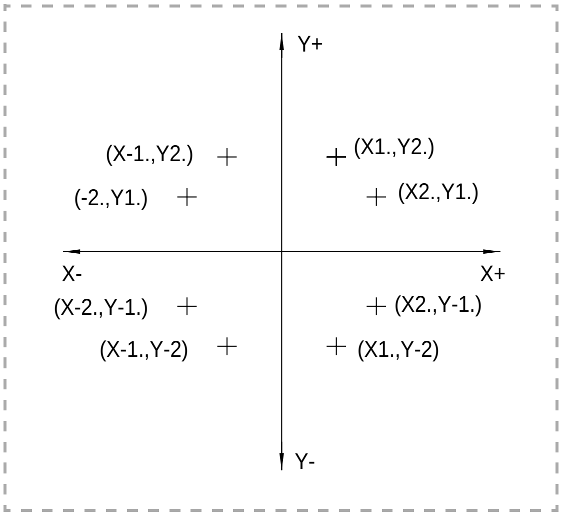


Figure 20-2: Coordinate System

20.2.6 POINTS

Describe a point by specifying its X and Y distance from home. The X is measured right (+) or left (-) from the X axis zero location. The Y is measured up (+) or down (-) from the Y axis zero location.

20.2.7 ANGLES

Angles are measured from the positive X axis using decimal degrees. A positive angle is measured counterclockwise from the X positive axis, and the negative angle is measures clockwise from the X positive axis.

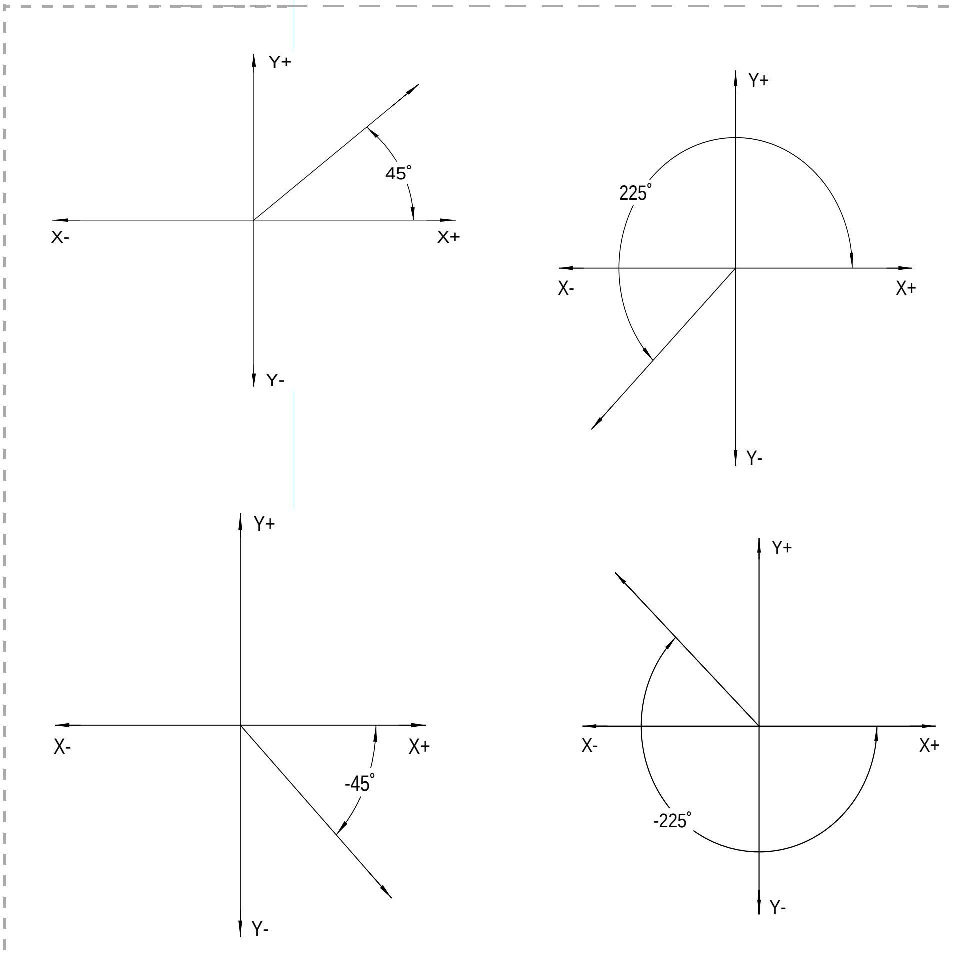


Figure 20-3: Angles

Angles describe direction of motion on a line. Moving on a line to the right describes the line to be at an angle of 0 deg. Moving on a line to the left describes the line to be at an angle of 180 deg.

20.2.8 LINES

Lines are described by specifying a point on a line and the angle of the line. The point does not need to be on the part. It can be on an extended portion of the line. The angle defines the direction of the motion on the line (see Angles).

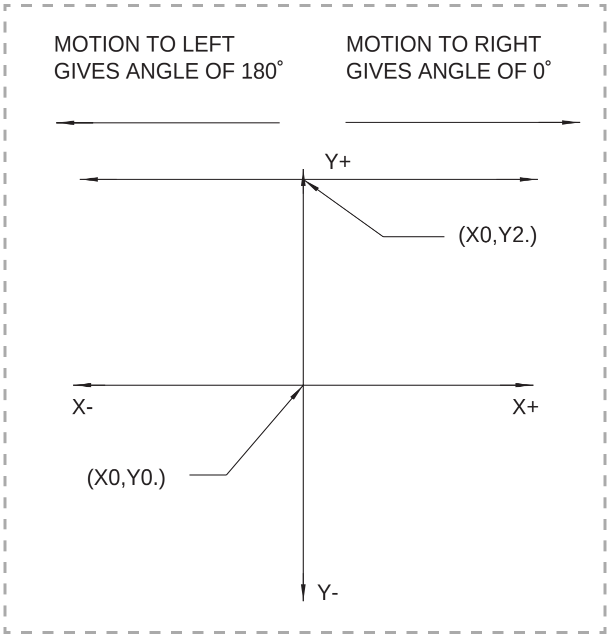


Figure 20-4: **Lines**

20.2.9 CIRCLES

A circle is defined by the X, Y location of its center, a Radius, and a direction of motion. The circle direction of motion is described by clockwise or counterclockwise. To define a circle as a point, describe the circle as having a zero radius. If the X, Y center location is not known, then it is to be considered a blend radius.

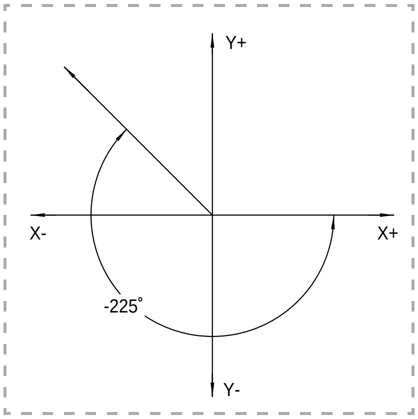


Figure 20-5: **Circles**

20.2.10 BLEND RADIUS

A blend radius is a circle that is situated between two known geometric elements. It is defined by a radius value and a direction of motion. The blend radius direction of motion is described by clockwise or counterclockwise.

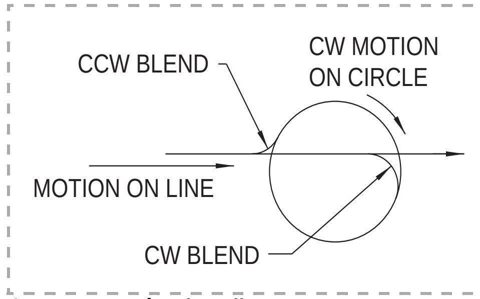


Figure 20-6: Blend Radius

20.3 THE FUNCTION MENU DISPLAY

20.3.1 FUNCTION MENUS

The Function menu display and each of the graphics menus are shown as they appear on the monitor.

20.3.2 MAIN MENU

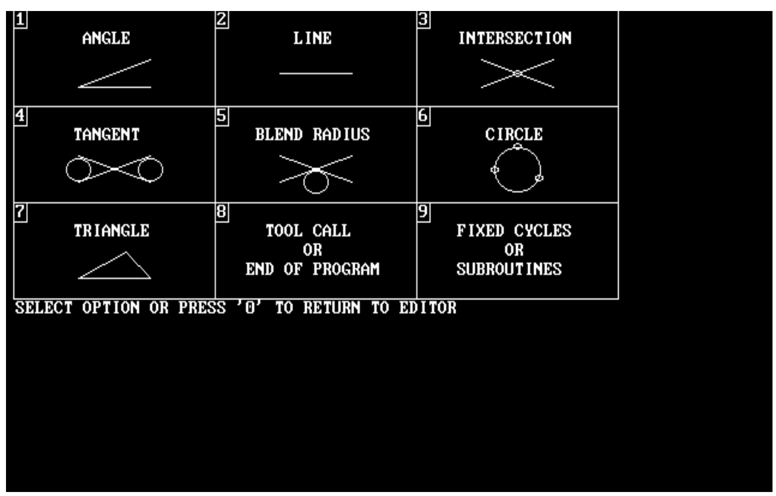


Figure 20-7: Main Menu

20.3.3 ANGLE OF A LINE

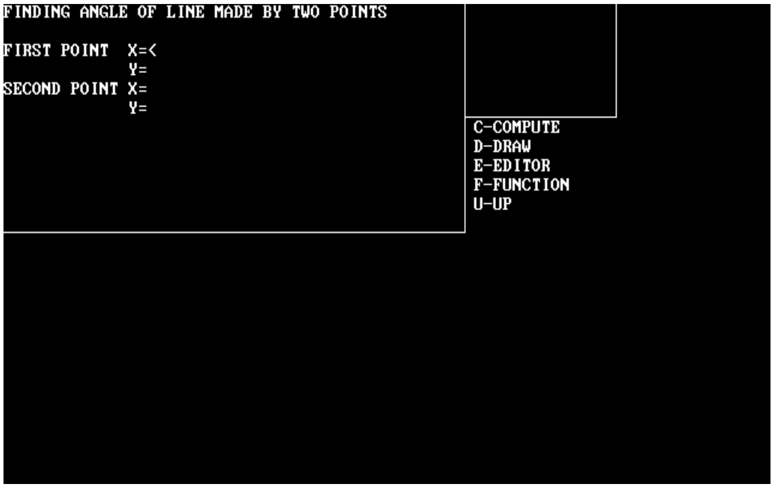


Figure 20-8: Angle of a Line

20.3.4 TWO LINE FUNCTIONS

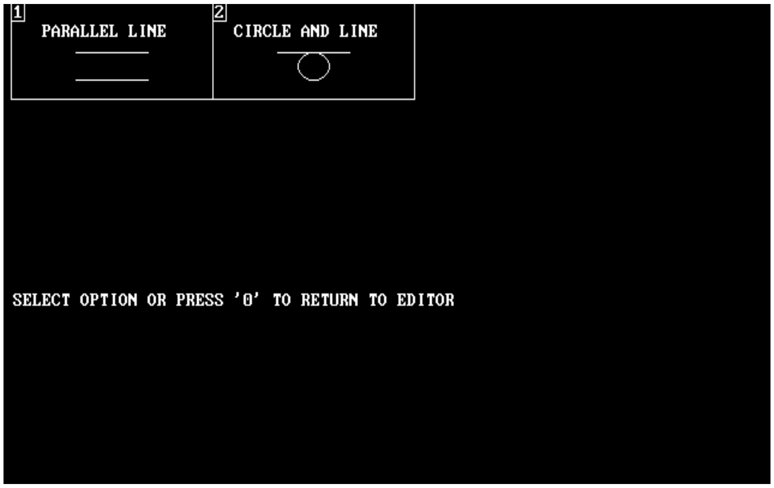


Figure 20-9: Two Line Functions

20.3.5 FINDING A PARALLEL LINE USING A LINE OR CIRCLE

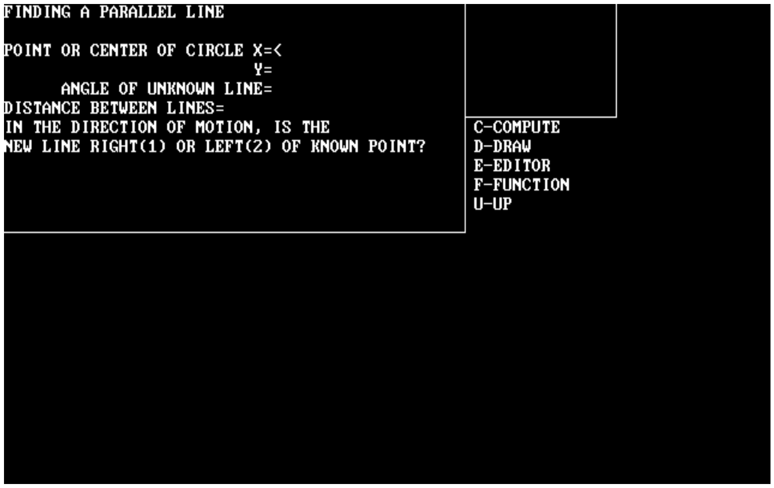


Figure 20-10: Finding a Paralell Line

20.3.6 THREE INTERSECTION FUNCTIONS

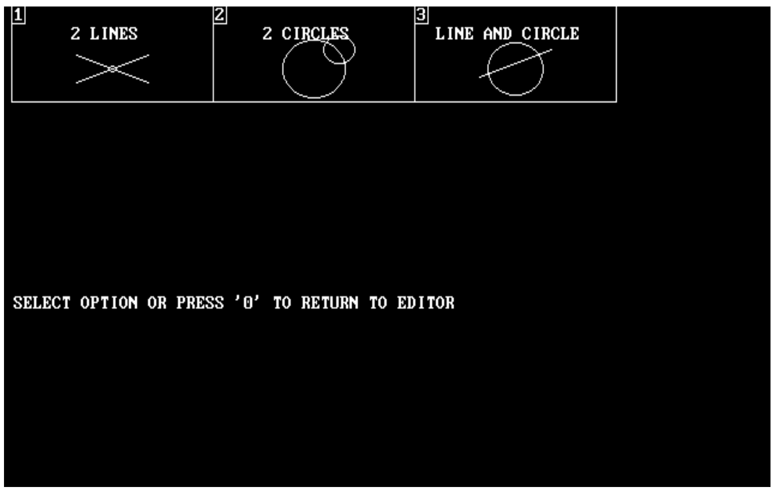


Figure 20-11: Three Intersection Functions

20.3.7 INTERSECTION OF 2 LINES

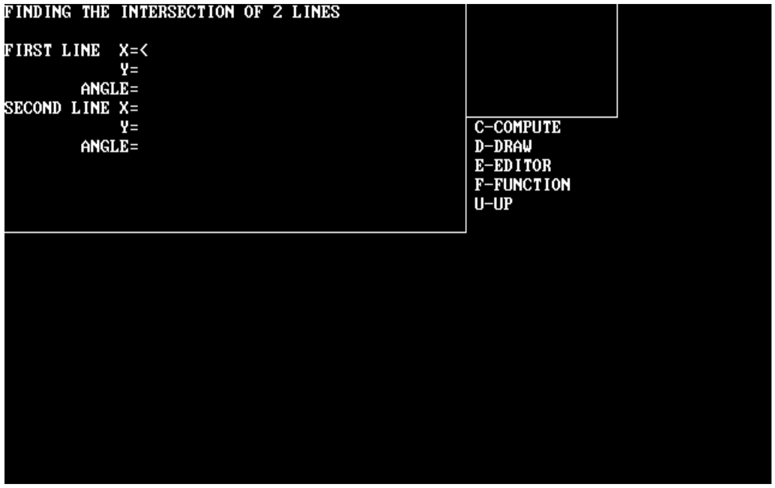


Figure 20-12: Intersection of 2 Lines

20.3.8 INTERSECTION OF 2 CIRCLES

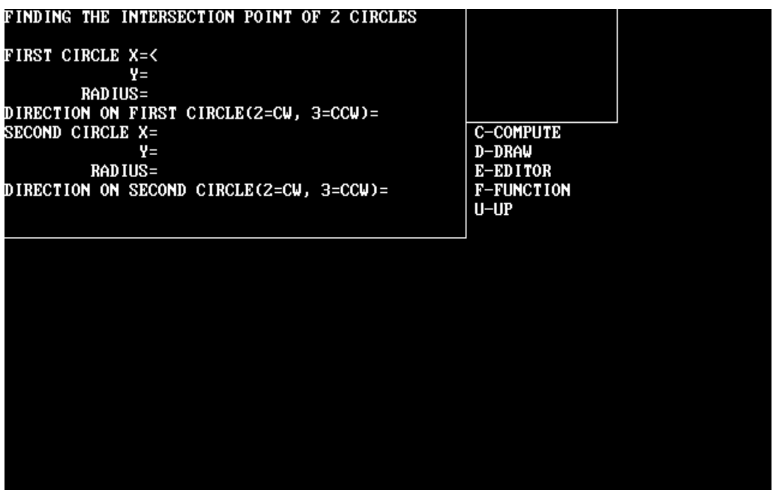


Figure 20-13: Intersection of 2 Circles

20.3.9 INTERSECTION OF A LINE AND CIRCLE

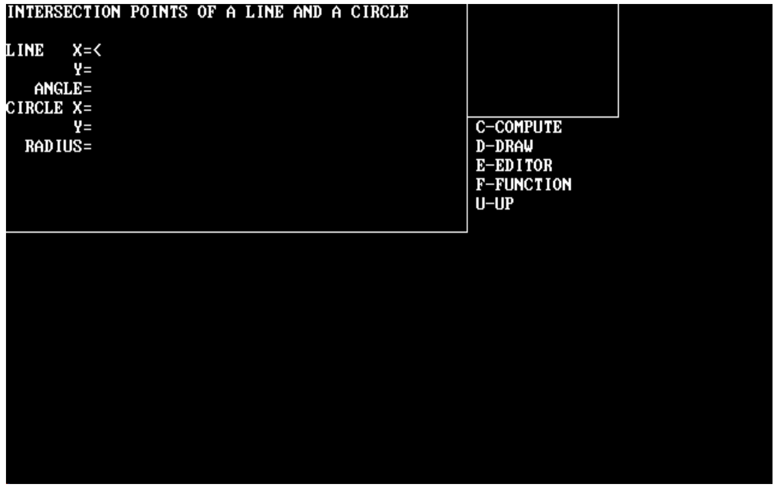


Figure 20-14: Intersection of a Line and Circle

20.3.10 TWO TANGENT FUNCTIONS

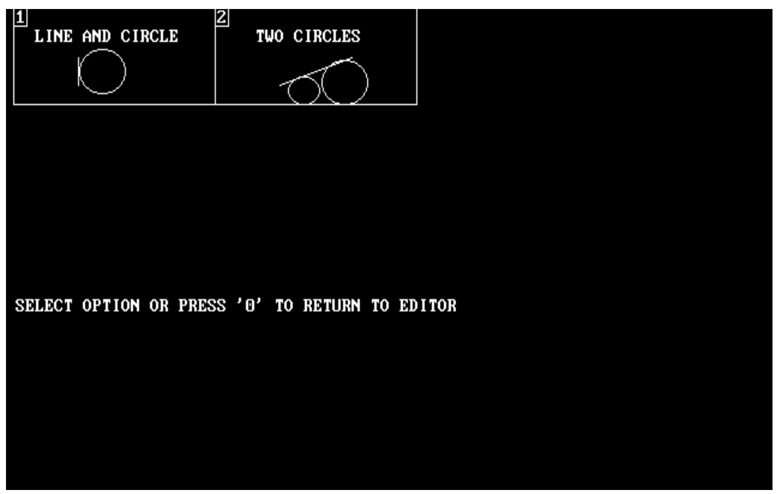


Figure 20-15: Two Tangent Functions

20.3.11 TANGENT POINT OF A LINE AND CIRCLE

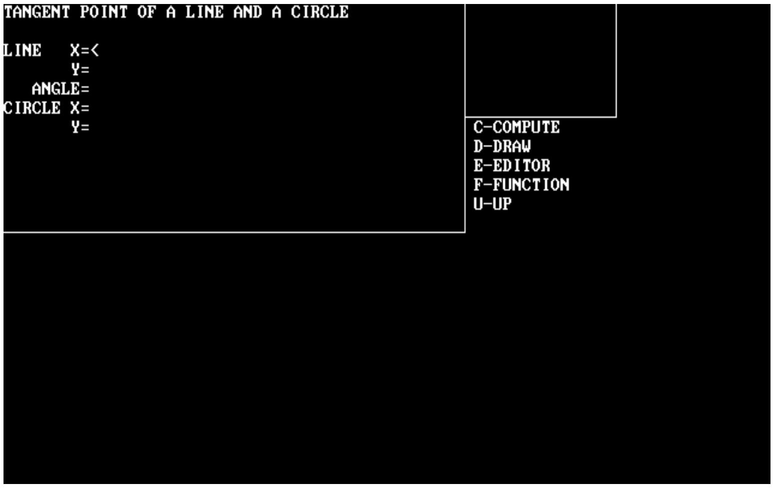


Figure 20-16: Tangent Point of a Line and Circle

20.3.12 TANGENT POINTS OF TWO CIRCLES

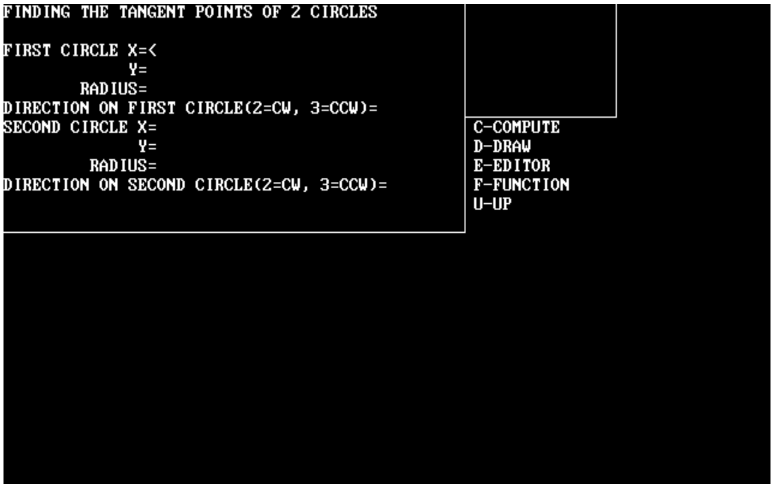


Figure 20-17: Tangent Points of Two Circles

20.3.13 NINE BLEND RADIUS FUNCTIONS

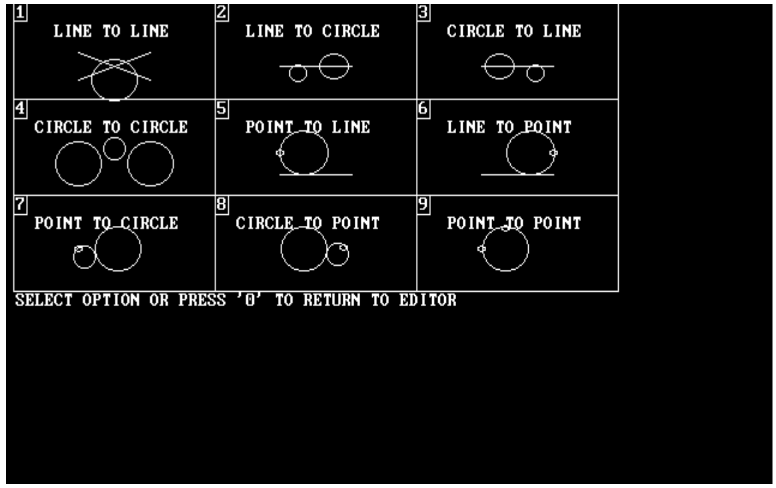


Figure 20-18: Nine Blend Radius Functions

20.3.14 BLEND RADIUS FROM A LINE TO A LINE

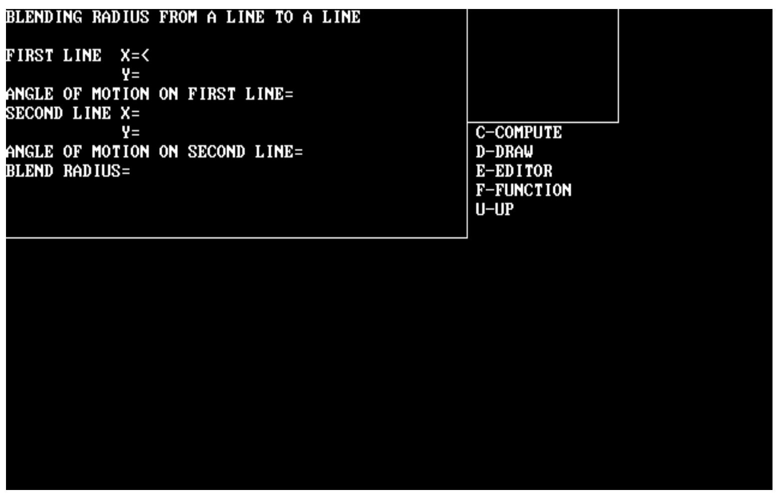


Figure 20-19: Blend Radius from a Line to a Line

20.3.15 BLEND RADIUS FROM A LINE TO A CIRCLE

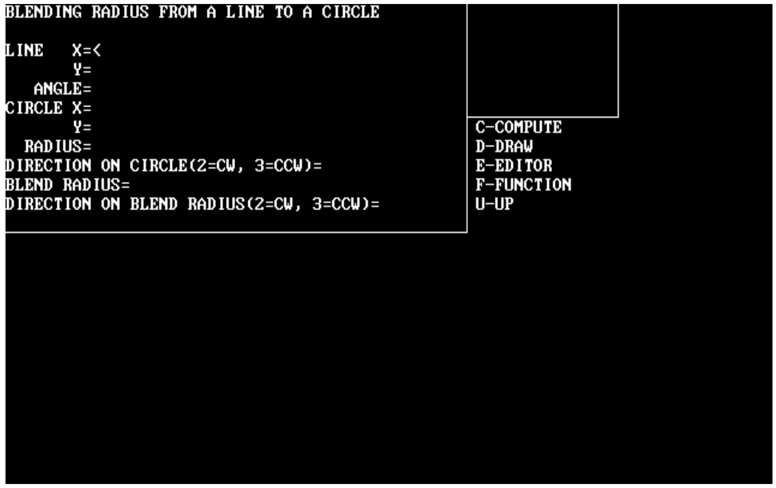


Figure 20-20: Blend Radius from a Line to a Circle

20.3.16 BLEND RADIUS FROM A CIRCLE TO A LINE

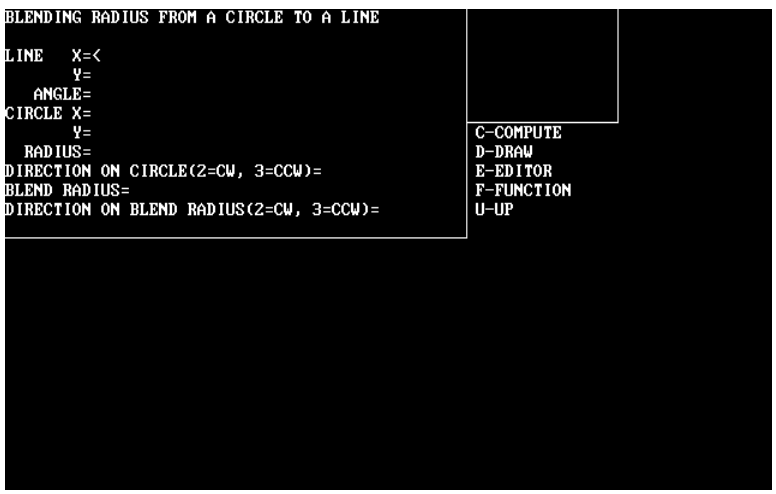


Figure 20-21: Blend Radius from a Circle to a Line

20.3.17 BLEND RADIUS FROM A CIRCLE TO A CIRCLE

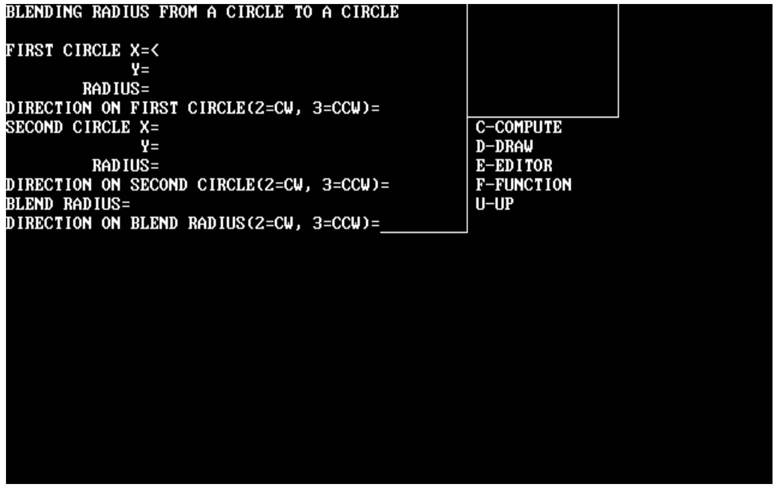


Figure 20-22: Blend Radius from a Circle to a Circle

20.3.18 BLEND RADIUS FROM A POINT TO A LINE

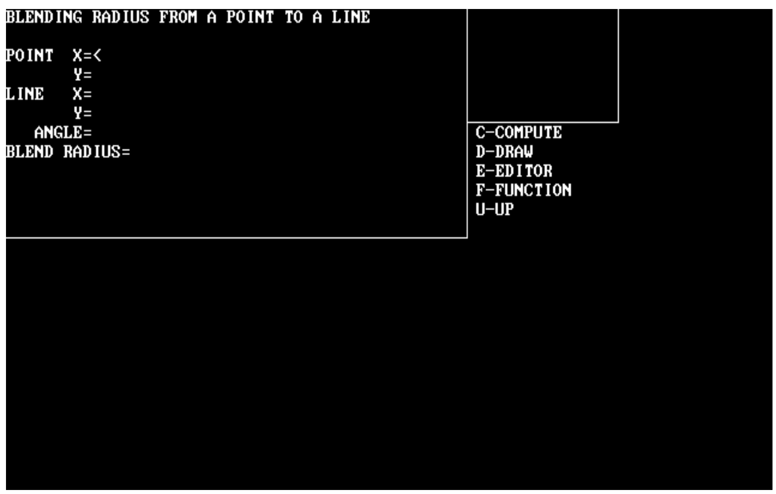


Figure 20-23: Blend Radius from a Point to a Line

20.3.19 BLEND RADIUS FROM A LINE TO A POINT

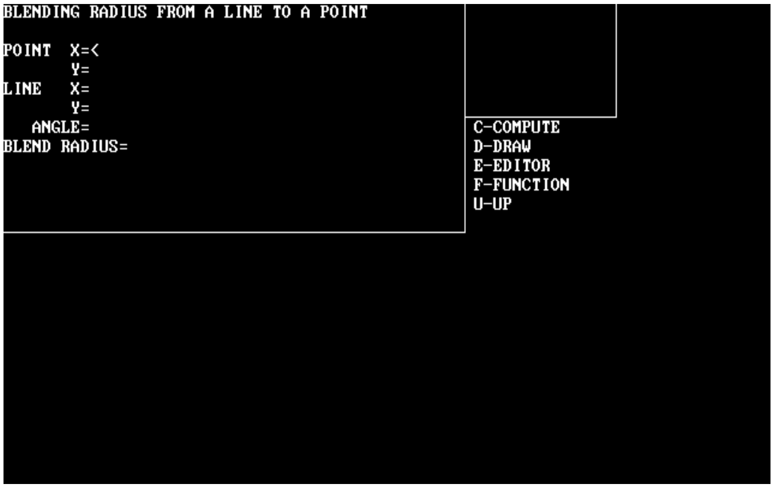


Figure 20-24: Blend Radius from a Line to a Point

20.3.20 BLEND RADIUS FROM A POINT TO A CIRCLE

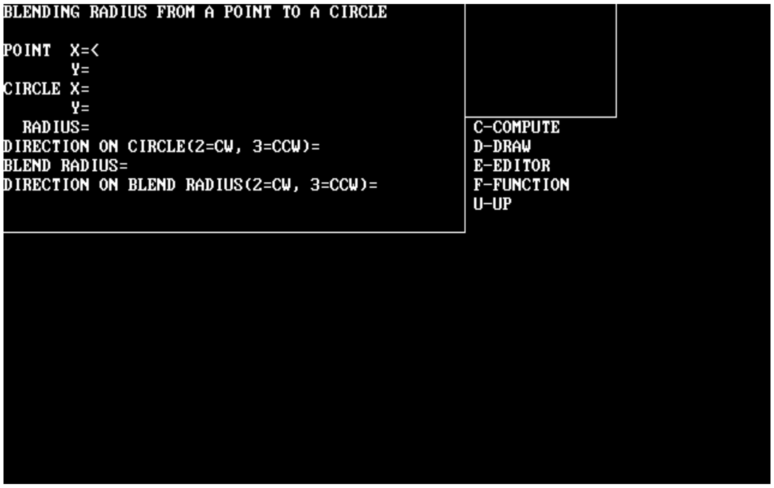


Figure 20-25: Blend Radius from a Point to a Circle

20.3.21 BLEND RADIUS FROM A CIRCLE TO A POINT

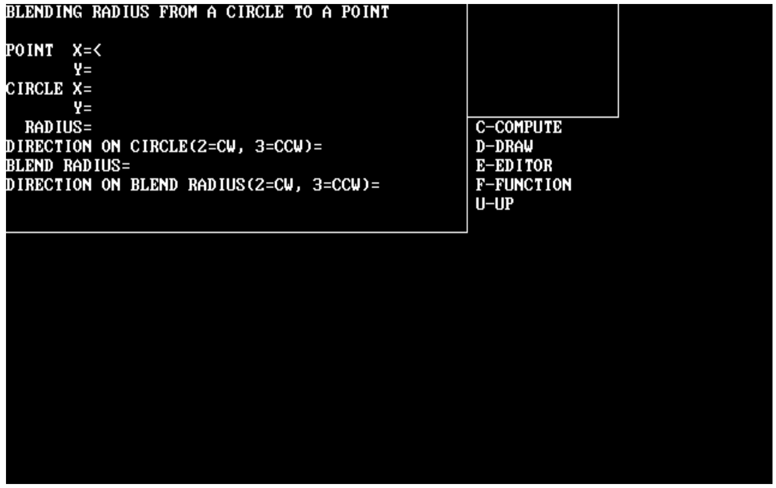


Figure 20-26: Blend Radius from a Circle to a Point

20.3.22 BLEND RADIUS FROM A POINT TO A POINT

Figure 20-27: Blend Radius from a Point to a Point

20.3.23 CIRCLE FUNCTION

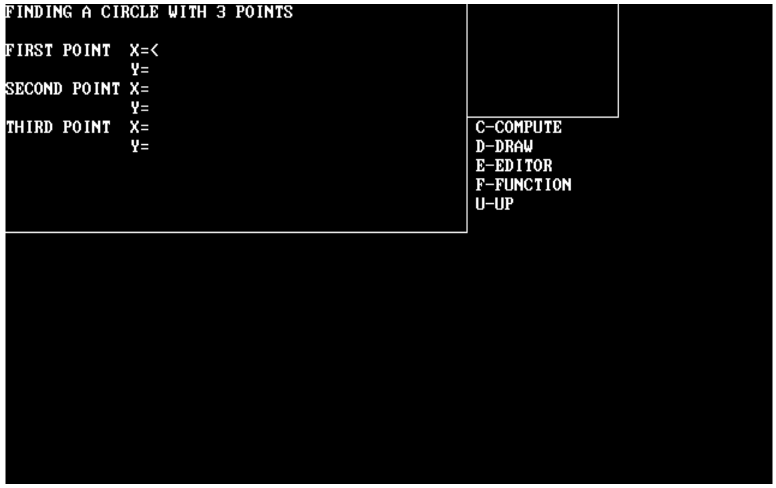


Figure 20-28: Circle Function

20.3.24 TRIANGLE SOLVER

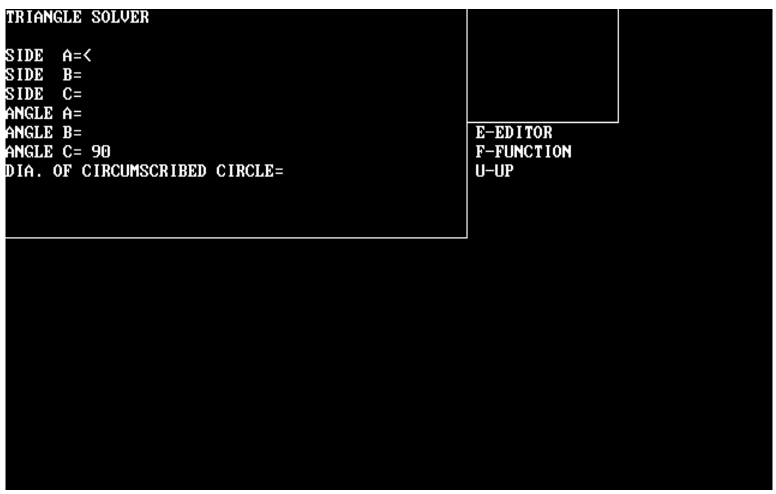


Figure 20-29: Triangle Solver

20.3.25 TOOL CALL AND END OF PROGRAM

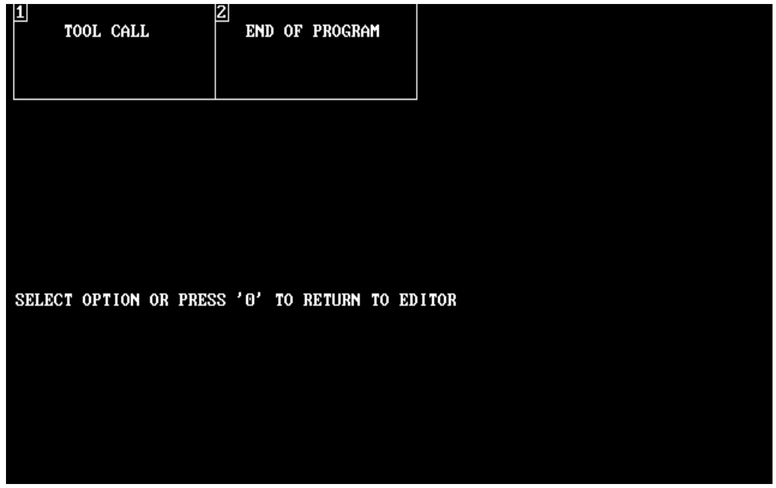


Figure 20-30: Tool Call and End of Program

20.3.26 TOOL CALL

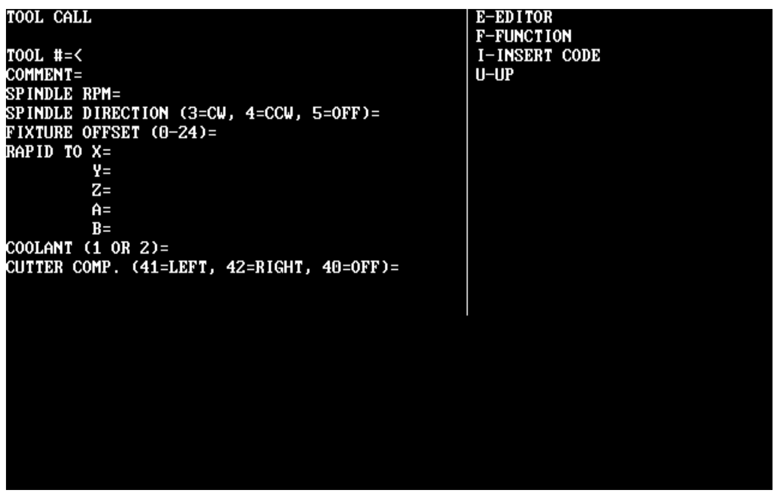


Figure 20-31: Tool Call

20.3.27 END OF PROGRAM

WARNING
End of program will insert at cursor!

The following will be inserted at the cursor for an “end of program”:

(ENDING PROGRAM
G0 G80 G90 M5 M9
Z0 G53
X0 Y0 Z0 E0 H0
M30

20.3.28 FIXED CYCLES AND SUBROUTINE FUNCTIONS

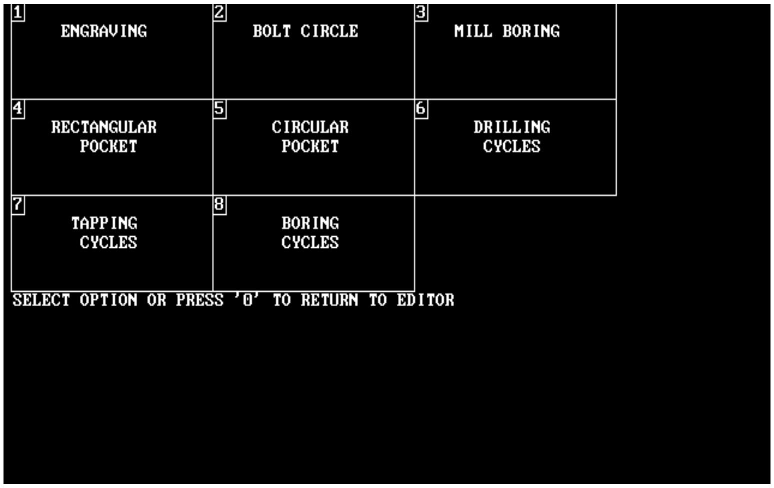


Figure 20-32: Fixed Cycles and Subroutine Functions

20.3.29 ENGRAVING

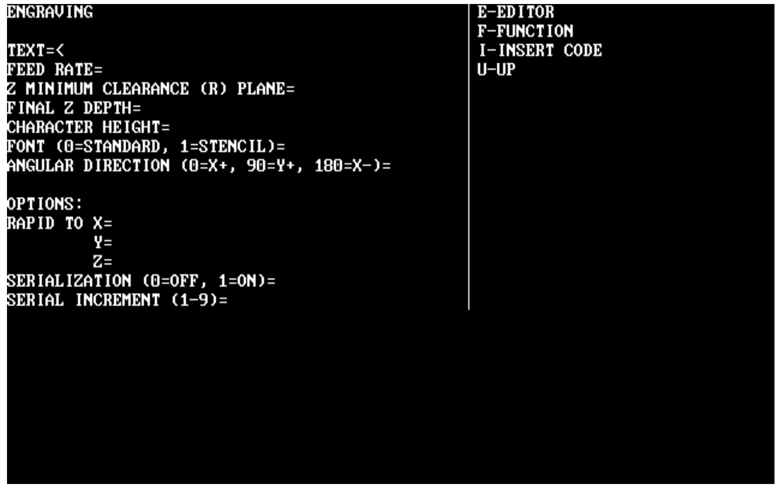


Figure 20-33: Engraving

20.3.30 BOLT CIRCLE

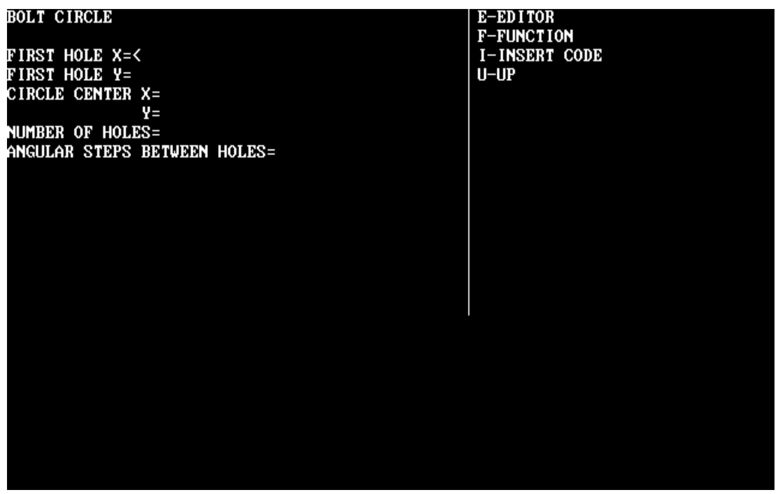


Figure 20-34: Bolt Circle

20.3.31 MILL BORING

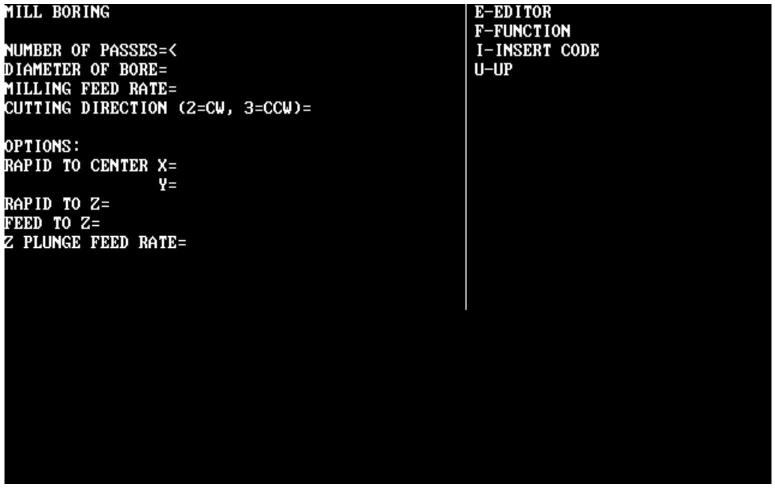


Figure 20-35: Mill Boring

20.3.32 RECTANGULAR POCKET

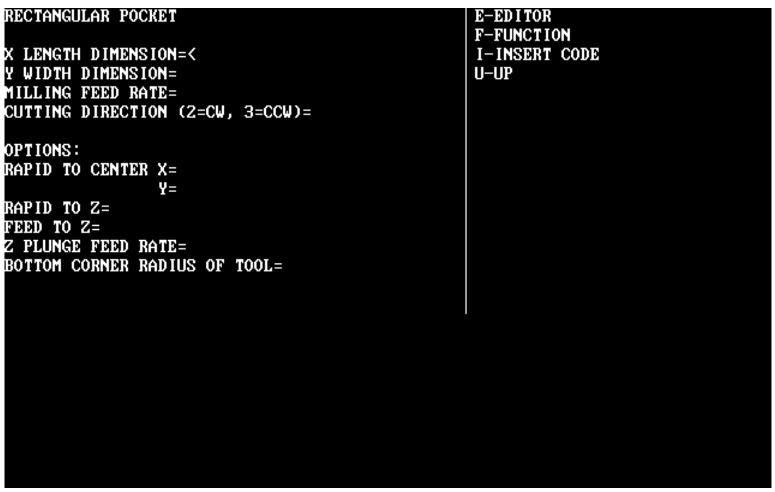


Figure 20-36: Rectangular Pocket

20.3.33 CIRCULAR POCKET

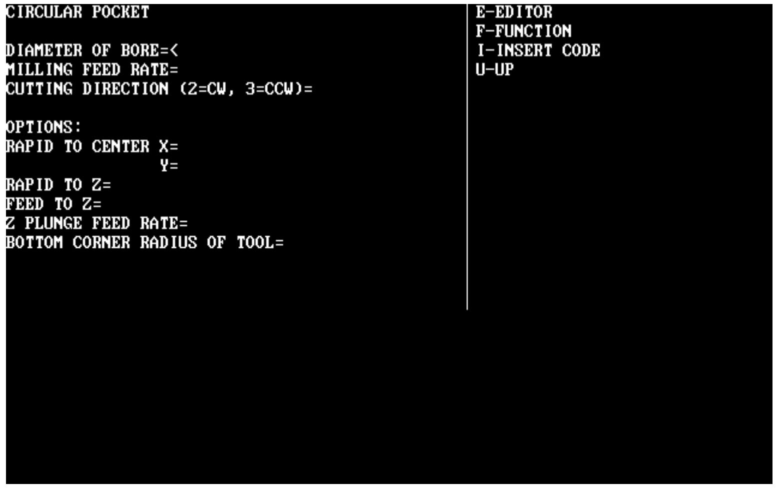


Figure 20-37: Circular Pocket

20.3.34 DRILLING CYCLES

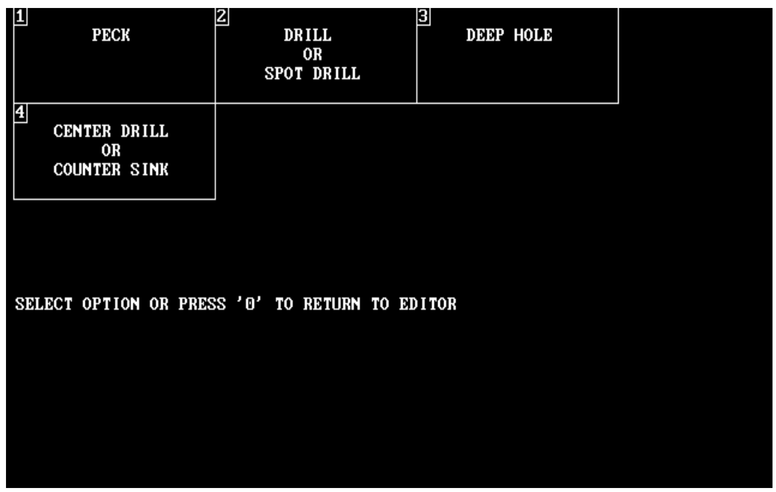


Figure 20-38: Drilling Cycles

20.3.35 TAPPING CYCLES

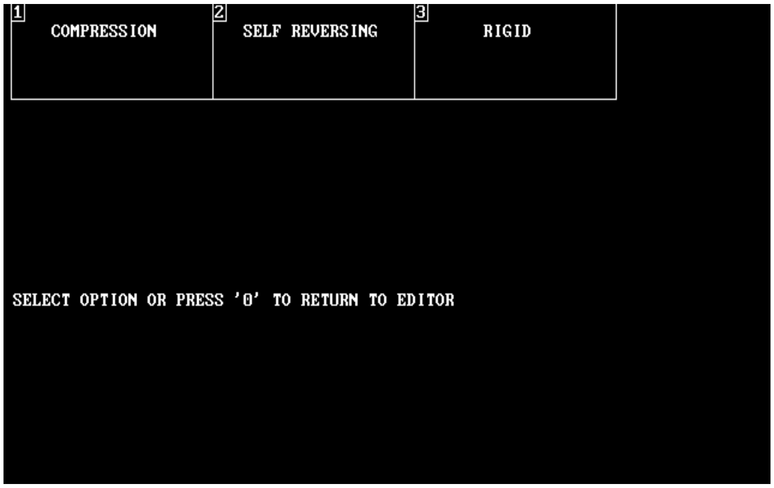


Figure 20-39: Tapping Cycles

20.3.36 BORING CYCLES

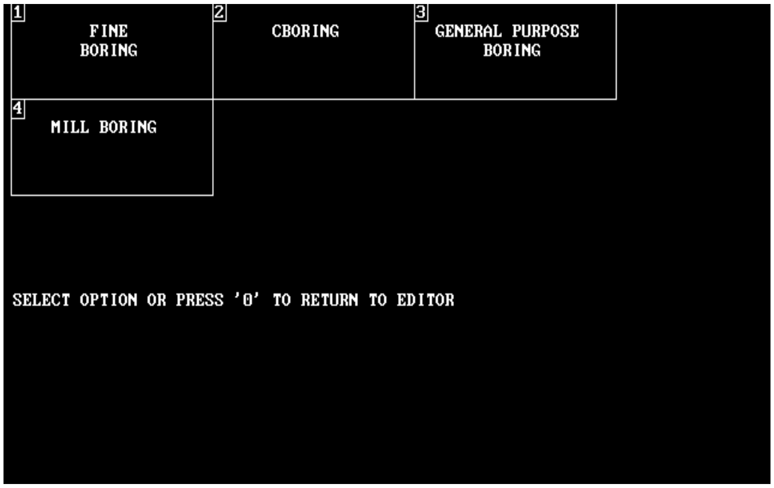


Figure 20-40: Boring Cycles

21.0 FORMATS

21.1 OVERVIEW

From the SETP Command the control has the option to use either FORMAT 1 or FORMAT 2.

21.1.1 FORMAT 1

Format 1 is FADAL style programming and machine operation.

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

21.1.2 FORMAT 2

Format 2 allows a 6M/ 10M /11M style programming and machine operation.

Format 2 follows the design to allow the programmer to completely command the CNC control. All resets therefore must be done by the programmer in the program and machine operation. Some codes are unique for FORMAT 2 and FORMAT 1. Listed in this chapter are examples for each format's programming style.

21.2 FORMAT 1 PROGRAMMING

21.2.1 NOTES ON FORMAT 1 STYLE PROGRAMMING

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

EXAMPLE:

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

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

3. The O word on the first line is not required. The O word is used only for identification of the program in the program library for multiple program storage.

The first sixteen characters of the comment with the O word will appear when the program library directory is displayed. This comment can be longer than sixteen characters but the remainder of the comment will only appear when the program is listed or printed (See also the section on program library, the PR command).

4. If the word NOEDIT is used within the first sixteen characters of the comment on the line with the O word, the program cannot be edited. This is the same function as using the key lock on the pendant. Program can only be deleted with the PR command.

21.2.2 FORMAT 1 PROGRAM EXAMPLE

EXAMPLE 1, without subroutines or subprogram calls with fixture offsets:

O# (COMMENT
M6 T1
(TOOL #1 ID
G0 G90 S##### M3 E## X###.#### Y###.####
H# M7 Z###.####
G4 P1000
Math work
M5 M9
G0 G90 80
Z0 G53
M6 T2
(TOOL #2 ID
G0 G90 S##### M3 E## X###.#### Y###.####
H# M7 Z###.####
G4 P1000
Math work
M5 M9
G0 G90 G80
Z0 G53
E0 X0 Y0 H0
M2

21.2.3 FORMAT 1 PROGRAM WITH SUBROUTINES AND SUBPROGRAM CALLS

O# (COMMENT
(SUBROUTINE SECTION
L100 (SUB #1 FOR _______ Each sub begins with an L word, a sub
Math worknumber (1-99), and a two digit zero repetition (00)
M17
L200 (SUB #2 FOR _______) Math work
M17
L300 (SUB #3 FOR _______) Math work

M17 This M17 marks the end of the last sub
M30 The M30 marks the end of the program's sub section
(MAIN PROGRAM)
M6 T1(.........TOOL #1 ID
G0 G90 S##### M3 E## X###.#### Y###.####
H# M8 Z###.####
G4 P1000
Math work

L101 This line calls sub #1 one time
M98 P5 L1 This line calls subprogram 5 one time
L215 This line calls sub #2 fifteen times
G91 X##.#### L9 The L9 repeats this line nine times
M5 M9
G0 G90 G80
Z0 G53
M6 T2(..........TOOL #2 ID
G90 G0 S##### M3 E## X###.#### Y###.####
H# M8 Z###.####
G4 P1000
Math work

M5 M9
G0 G90 G80
Z0 G53
E0 X0 Y0
M2

O5 (SUBPROGRAM #5 FOR _______________ Math work
M99 Ending this program with an M99 makes it a subprogram

21.3 FORMAT 2 PROGRAMMING

21.3.1 NOTES ON FORMAT 2 STYLE PROGRAMMING

1. The safe block is mandatory at the beginning of the program.

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

2. The G28, with the axis designated to move, will bring the machine to the zero position established by the cold start command (CS) or to the zero position established by the SETX, SETY, SETZ, or SETH commands and/or fixture offsets, whichever was used last. The SET commands can be used to establish a zero position at any desired location. The program does not need to start from the machine home position. See also the section on establishing home position.

3. Fixed subroutines such as bolt hole pattern subroutines or pocket cutting routines from other machines cannot be used on this control. (See section 6.0 FIXED SUBROUTINES, MAN-0131.)

4. The SETP command allows certain changes to default codes, diameter or radius input, and M7 M8 preferences.

5. When typing any variable (R0,R1 - R9) always type the positive or negative symbol before the value, for example, R1+.5 R2+2.34 R3+5.7124 R4-6.765. (See section 6.0 FIXED SUBROUTINES, MAN-0131.)

6. The H word will only pick up the tool length offset. The D word must be used to pick up the diameter or radius from the tool table.

7. Tool changes can be made from any XY position. The M6 code change will move the head to the tool position and make the exchange. The T word and the M6 can appear together, M6T# or T# M6, and usually do with this type of tool changer; however they may appear on separate lines.

8. Each line requires a sequence number. Sequence numbers use an N####.### word. Decimal points can be used to input lines between existing line numbers. The sequence numbers can be added when the program is typed at the computer, but the numbers are added automatically when the program is transferred to the control with the TA (tape input) command. So it is possible to type the program without line numbers and have the control add the line numbers automatically. The NU command is used to renumber the lines.

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

EXAMPLE:

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

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

10. The O word on the first line is not required. The O word is used only for identification of the program in the program library for multiple program storage. The first sixteen characters of the comment with the O word will appear when the program library directory is displayed. This comment can be longer than sixteen

characters but the remainder of the comment will only appear when the program is listed or printed (see also the section on program library, the PR command).

11. If the word NOEDIT is used within the first sixteen characters of the comment on the line with the O word, the program cannot be edited. This is the same function as using the key lock on the pendant.

12. When transferring a program from another control through the RS232 port, the original line numbers can be maintained by selecting #2 from the N WORDS ORDERED: parameter. Selecting this will cause the control to maintain the original line numbers. The first row is from the original program. When the program is output from the machine, only the original numbers are outputted.

21.3.2 FORMAT 2 PROGRAM EXAMPLE

```
O# (COMMENT
(COMMENT
G53 Z0
G0 G17 G40 G70 G80 G90 H0 Z0  This safety line is a  RESET
E0 X0 Y0   This moves to the SETH position or cold start
G54 X##.#### Y##.####  (or E1 X# Y# or G92 X# Y# are alternative
M6 T1
(TOOL #1 ID
G0 G90 S##### M3 X###.#### Y###.####
H# M8 Z###.#### D#
G4 P1000
Math work
M98 P4 L4   This calls subprogram #4 four times
M98 P3 L27   This calls subprogram #3 twenty seven times
G91 X##.#### G90 L9   The L9 repeats this line nine times
M5 M9
G0 G90 H0 Z0
M6 T2
(TOOL #2 ID
G0 G90 S##### M3 X###.#### Y###.#### E## (or G54-G59)
H# M8 Z###.#### D#
G4 P1000
Math work
M5 M9
G0 G90 H0 Z0
E0 X0 Y0   Use this line to cancel all fixture offsets
M30
O3 (SUBPROGRAM #3 FOR _______________
Math work
```

M99 Ending this program with an M99 makes it a subprogram. It is called with an M98 P3 L#. The L word is used for number of repeats
O4 (SUBPROGRAM #4 FOR ______________
Math work
M98 P3 L1 Sub #4 is calling sub #3 one time (nesting)
M99 Ending this program with an M99 makes it a subprogram

21.4 DEFAULT PARAMETERS, SETP COMMAND

21.4.1 FORMAT 1 & FORMAT 2

Changing from FORMAT 1 (FADAL mode), to FORMAT 2 (6M, 10M, 11M mode), is done from the SETP mode.

Format 2 allows a 6M/ 10M /11M style programming and machine operation.

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

WARNING
When operating in the FORMAT 2 mode, All RESETS must be commanded by the user.

1. The automatic move to home, upon program initiation, must be commanded in the program. Format 2 requires commands to reset. All reset commands must be used with any AUTO command.
    a. In most cases it is required to use the SETCS then the HO command to place the machine to the CS alignment position before typing SETP. Verify that the alignment markers all line up. User may be instructed to COLD START CNC.

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

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

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

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

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

21.4.2 FORMAT 1 PARAMETER SETTINGS

The following is the screen's representation upon entering the SETP command. The parameters FORMAT, DEFAULTS, IMM> FIXED CYCLE and CRC MODE should be set as shown. All other parameters should be set as they are from the factory.

Figure 21-1: Format 1 Parameter Settings

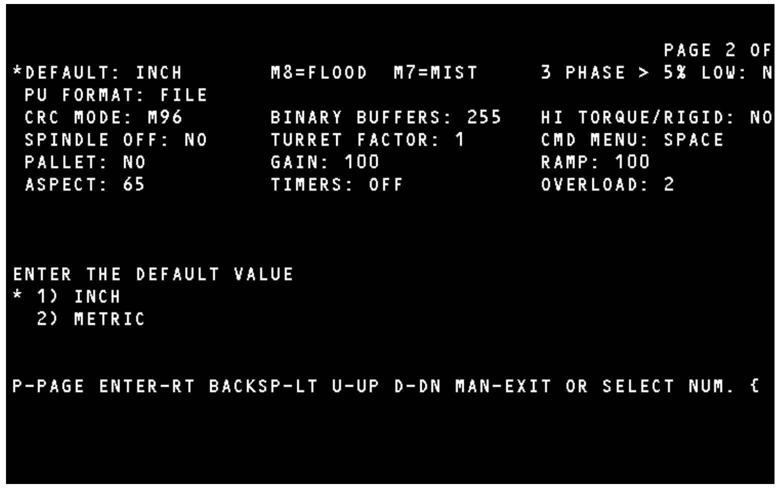


Figure 21-2: Format 1 Parameter Settings (Continued)

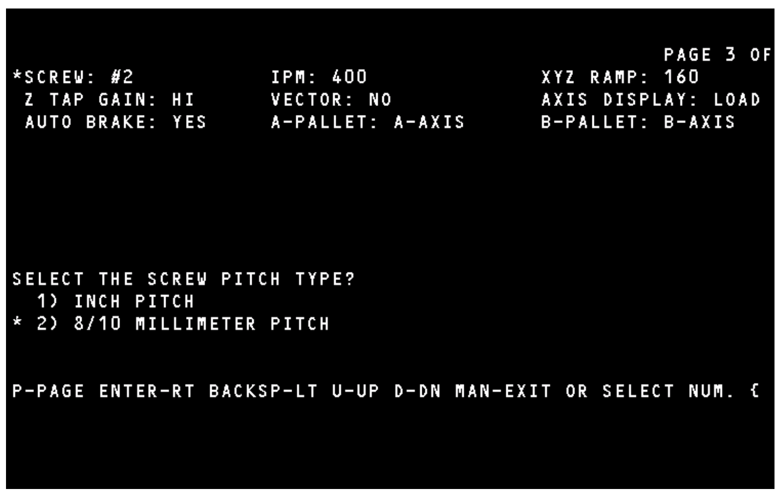
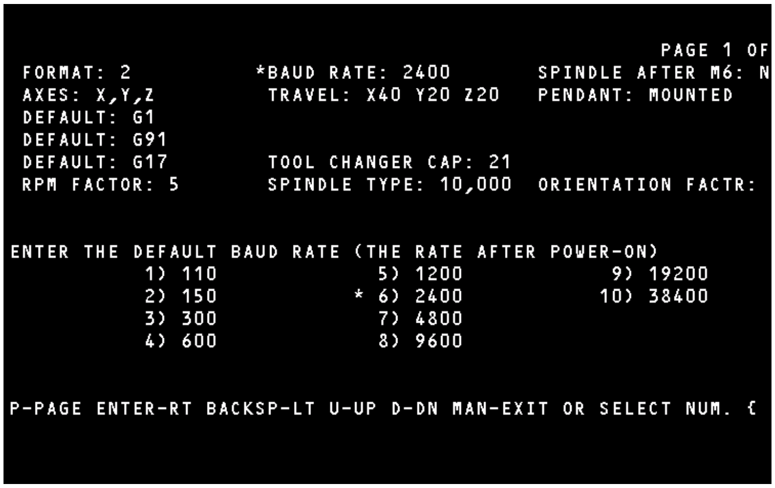


Figure 21-3: Format 1 Parameter Settings (Continued)

21.4.3 FORMAT 2 PARAMETER SETTINGS

The following is the screen representation upon entering the SETP command. The parameters FORMAT, DEFAULTS, CRC MODE, N-WORDS ORDERED and TOOL

TABLE should be set as shown. All other parameters should be set as they are from the factory.

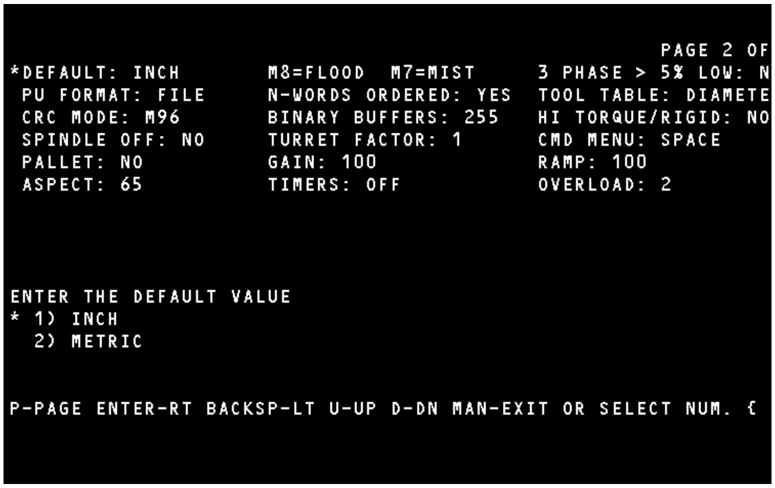


```
                                                       PAGE 1 OF
 FORMAT: 2          *BAUD RATE: 2400      SPINDLE AFTER M6: N
 AXES: X,Y,Z         TRAVEL: X40 Y20 Z20   PENDANT: MOUNTED
 DEFAULT: G1
 DEFAULT: G91
 DEFAULT: G17        TOOL CHANGER CAP: 21
 RPM FACTOR: 5       SPINDLE TYPE: 10,000  ORIENTATION FACTR:

ENTER THE 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 21-4: Format 2 Parameter Settings



```
                                                       PAGE 2 OF
*DEFAULT: INCH       M8=FLOOD  M7=MIST     3 PHASE > 5% LOW: N
 PU FORMAT: FILE     N-WORDS ORDERED: YES  TOOL TABLE: DIAMETE
 CRC MODE: M96       BINARY BUFFERS: 255   HI TORQUE/RIGID: NO
 SPINDLE OFF: NO     TURRET FACTOR: 1      CMD MENU: SPACE
 PALLET: NO          GAIN: 100             RAMP: 100
 ASPECT: 65          TIMERS: OFF           OVERLOAD: 2

ENTER THE DEFAULT VALUE
* 1) INCH
  2) METRIC

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

Figure 21-5: Format 2 Parameter Settings (Continued)

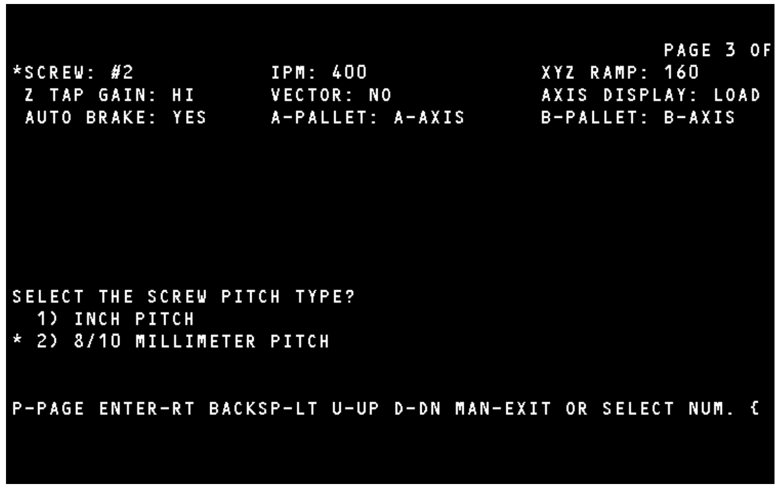


Figure 21-6: Format 2 Parameter Settings (Continued)

21.4.4 FORMAT

THERE ARE TWO PROGRAM OPERATION FORMATS AVAILABLE

DO YOU WANT MAXIMUM 6MB/10M/11M CAPABILITY?
1) NO, ORIGINAL FADAL STYLE - FORMAT 1
2) YES, FORMAT 2

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

21.4.5 AXES: X,Y,Z

ENTER THE AXIS CONFIGURATION

1) A
2) A, B
3) X,Y,Z
4) X,Y,Z,A
5) X,Y,Z,A,B
6) X,Y,Z,B
7) C only

When selecting the A or B axes, the machine should be powered off. When the power is returned the axes will be active. COLD START must then be performed.

21.4.6 DEFAULT: G0

ENTER THE DEFAULT VALUE

1) G0
2) G1

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

21.4.7 DEFAULT: G90

ENTER THE DEFAULT VALUE

1) G90
2) G91

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

21.4.8 DEFAULT: G17

ENTER THE DEFAULT VALUE

1) G17
2) G18
3) G19

This parameter is used to select the default machine plane.

21.4.9 RPM FACTOR

ENTER THE SPINDLE RPM ADJUSTMENT FACTOR

THE FACTOR MUST BE BETWEEN 0 AND 31

This parameter should only be adjusted by trained maintenance personnel.

21.4.10 BAUD RATE

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*	*11) 57600*
4) 600	*8) 9600*	*12) 115200*

The operator may select the desired communications baud rate.

21.4.11 TRAVEL

ENTER X,Y,Z TRAVEL

1) X22 Y16 Z20	*8) X40 Y20 Z28*	*15) X50 Y20 Z20*
2) X20 Y12 Z20	*9) X60 Y30 Z30*	*16) X50 Y20 Z28*
3) X20 Y13.5 Z20	*10) X22 Y16 Z28*	*17) X80 Y30 Z30*
4) X22 Y13.5 Z20	*11) X20 Y16 Z20*	*18) X30 Y20 Z24*
5) X22 Y13 Z20	*12) X20 Y16 Z28*	*19) X30 Y20 Z32*
6) X20 Y13 Z20	*13) X30 Y16 Z20*	*20) X20 Y20 Z24*
7) X40 Y20 Z20	*14) X30 Y16 Z28*	*21) X20 Y20 Z32*

The machine travel is selected with this parameter.

21.4.12 A-AXIS RATIO

ENTER A-AXIS RATIO

1) 90:1
2) 180:1
3) 360:1
4) 120:1
5) 72:1
6) COOLANT
7) 60:1
8) 144:1

Select the appropriate option for the rotary table being used. See servo coolant option.

21.4.13 B-AXIS RATIO	ENTER B-AXIS RATIO *1) 90:1* *2) 180:1* *3) 360:1* *4) 120:1* *5) 72:1* *6) COOLANT* *7) 60:1* *8) 144:1* Select the appropriate option for the rotary table being used. See servo coolant option.
21.4.14 TOOL CHANGER CAP	ENTER THE TOOL CHANGER CAPACITY *1) 16 TOOL ATC* *2) 21 TOOL ATC* *3) 30 TOOL ATC* Select the appropriate tool changer capacity.
21.4.15 SPINDLE TYPE	ENTER THE SPINDLE DRIVE TYPE & RPM *1) 10,000 HI/LOW* *2) 10,000 3 STEP* *3) 15,000 DIRECT* *4) 5,000 HI/LOW* *5) 6,500 HI/LOW* *6) 3750 WYE/DEL.* *7) 7500 WYE/DEL.* *8) 5000 DIRECT* *9) 15K WYE/DEL.* Select the correct spindle type for the machine. This parameter is set at the factory.
21.4.16 SPINDLE AFTER M6	SHOULD 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 OVER SPEEDING A TOOL BEFORE THE NEW S-WORD IS ENCOUNTERED*

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.**

21.4.17 PENDANT

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

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.

21.4.18 M60/A-AXIS BRAKE

DOES M60 TURN ON THE A-AXIS BRAKE?

1) YES
2) NO

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

21.4.19 M62/B-AXIS BRAKE

DOES M62 TURN ON THE B-AXIS BRAKE?

1) YES
2) NO

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

21.4.20 IMM. FIXED CYCLE

SHOULD A FIXED CYCLE EXECUTE IMMEDIATELY?

1) YES
2) NO, ONLY IF X OR Y DIMENSION WORDS ARE IN THE DEFINITION LINE

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.

21.4.21 ORIENTATION FACTOR

ENTER THE SPINDLE RPM ADJUSTMENT FACTOR

THE FACTOR MUST BE BETWEEN 0 AND 31

This parameter should only be adjusted by trained maintenance personnel.

21.4.22 DEFAULT: INCH

ENTER THE DEFAULT VALUE

1) INCH
2) METRIC

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.

21.4.23 PU FORMAT

SELECT PUNCH OUTPUT FORMAT

1) PUNCHED TAPE FORMAT (TELETYPE STYLE)
2) COMPUTER FILE FORMAT (NO NULLS)

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

21.4.24 CRC MODE

ENTER THE DEFAULT OUTSIDE CORNER MOVEMENT

1) M96 - ROLL
2) M97 - INTERSECTIONAL

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

21.4.25 SPINDLE OFF

DO YOU WANT THE SPINDLE TO TURN OFF WHEN EXITING JOG OR MANUAL DATA INPUT?

1) NO
2) YES

This parameter is set at the factory and should not be changed.

21.4.26 PALLET	DO YOU HAVE A PALLET CHANGER? *1) NO* *2) YES* Select the option appropriate for the machine.
21.4.27 ASPECT	ENTER THE ASPECT RATIO OF Y TO X THE RATIO MUST BE BETWEEN 25 AND 150
21.4.28 M7-FLOOD M8-MIST	ENTER M7, M8 PREFERENCE *1) M7 IS FLOOD COOLANT, M8 IS MIST* *2) M8 IS FLOOD COOLANT, M7 IS MIST* The operator may select either M7 or M8 as the flood coolant code.
21.4.29 BINARY BUFFERS: 255	SELECT THE NUMBER OF BINARY BUFFERS FOR CNC LOOK-AHEAD. *1) 15 3) 50 5) 255* *2) 30 4) 100* The BINARY BUFFERS parameter can be changed to increase or decrease the control look ahead. The factory sets the buffers at 255. This is the most effective for programs with many small moves that must be executed rapidly. This parameter is used to more efficiently utilize the Run Time Menu. The smaller the buffers the more quickly the Run Time Menu changes will take effect in the program.
21.4.30 TURRET FACTOR	ENTER THE ENGAGEMENT FACTOR FOR THE TOOL TURRET GENEVA GEAR THE FACTOR MUST BE BETWEEN 1 AND 50 (1 FOR SERVO TURRET) This parameter is set at the factory. For VMCs equipped with the Servo Turret, this factor MUST always be 1.
21.4.31 GAIN	ENTER THE GAIN FACTOR FOR RIGID TAPPING THE FACTOR MUST BE BETWEEN 0 AND 255 This parameter effects 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.

21.4.32 TIMERS

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

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

21.4.33 3 PHASE 5% LOW: NO

IS YOUR 3 PHASE POWER MORE THAN 5% LOW?

1) YES
2) NO

The selection chosen is based upon the building power supply.

21.4.34 HIGH TORQUE

DO YOU HAVE THE HIGH TORQUE OPTION OR RIGID TAP OPTION?

1) NO
2) YES

This parameter is set at the factory.

21.4.35 CMD MENU

TURN COMMAND MENUS:

1) OFF
2) ON
3) TOGGLE ON WITH SPACE BAR

The operator may select the command menu structure.

21.4.36 RAMP

ENTER THE RAMP FACTOR FOR RIGID TAPPING

THE FACTOR MUST BE BETWEEN 0 AND 255

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

NOTE
Only a factory trained representative should modify this parameter.

21.4.37 OVERLOAD	ENTER THE MOTOR OVERLOAD FACTOR (2 IS STANDARD) THE FACTOR MUST BE BETWEEN 1 AND 8
21.4.38 SCREW	SELECT THE SCREW PITCH TYPE? *1) INCH PITCH* *2) 8/10MILLIMETER PITCH*
21.4.39 IPM	SELECT THE MAX RAPID FEED RATE? *1) 400 IPM 4) 800 IPM* *2) 500 IPM 5) 900 IPM* *3) 700 IPM*
21.4.40 XYZ RAMP	ENTER THE X, Y, & Z RAMP LENGTH FACTOR (160 IS STANDARD) THE FACTOR MUST BE BETWEEN 50 AND 200 This parameter sets the speed at which the axes accelerate and decelerate during travel.
21.4.41 Z TAP GAIN	SELECT THE Z GAIN DURING RIGID TAPPING? *1) NORMAL* *2) MEDIUM* *3) HIGH* Only a factory trained representative should modify this parameter.
21.4.42 VECTOR	DO YOU HAVE VECTOR DRIVE (10 RPM MIN)? *1) YES* *2) NO*
21.4.43 AXIS DISPLAY	DISPLAY THE AXIS FOLLOW ERROR OR LOAD? *1) LOAD PERCENTAGE*

2) FOLLOWING ERROR

This option is used to select what the axis display will show on the position display while auto running a program.

21.4.44 A-PALLET

WHICH ROTARY AXIS IS INSTALLED ON THIS PALLET?

1) NOT APPLICABLE
2) A-AXIS
3) B-AXIS
4) A-AXIS & B-AXIS

Select the appropriate option for the rotary axis being used.

21.4.45 B-PALLET

WHICH ROTARY AXIS IS INSTALLED ON THIS PALLET?

1) NOT APPLICABLE
2) A-AXIS
3) B-AXIS
4) A-AXIS & B-AXIS

Select the appropriate option for the rotary axis being used.

21.4.46 AUTO BRAKE

AUTOMATICALLY APPLY THE A OR B BRAKE DURING IDLE TIME?

1) YES
2) NO

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

21.4.47 5TH AXIS PROGRAMMABLE HEAD

DO YOU HAVE THE 5TH AXIS PROGRAMMABLE HEAD ATTACHMENT?

1) NO
2) YES

Answer YES if a 4th or 5th axis rotary head attachment is installed in the spindle.

21.4.48 G0 DETAIL

ENTER THE DETAIL DESIRED FOR G0 MOVES

BETWEEN 2 TO 250 TENTHS (251=OFF)

Enter a value between 2 and 250 to set the maximum distance to allow before the control will continue with the next motion block. Smaller values may have the tendency to cause the machine to wait at the end of G0/G5 moves. The value of 251 can be set to turn off the feature. This parameter should be set to any preference, as it will not affect contouring motion.

21.4.49 AIR VALVE FEEDBACK

DO YOU HAVE AIR VALVE FEEDBACK?

1) NO
2) YES

Answer NO only for machines built previous to having feedback on the drawbar.

21.5 PARAMETERS APPLICABLE TO FORMAT 2 ONLY

21.5.1 N-WORDS ORDERED

ENTER THE N-WORD SEQUENCE CONFIGURATION

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.

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.

21.5.2 TOOL TABLE

WILL THE TOOL COMPENSATION TABLE HAVE THE RADIUS OR DIAMETER?

1) DIAMETER
2) RADIUS

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 may require that you perform the Cold Start procedure (see CS command).

22.0 OPERATION OF THE TRM

22.1 INTRODUCTION

Fadal's smallest VMC model is the Tool Room Mill. Although It has many similarities with Fadal's larger models, it has a few operational differences that warrant a separate section in the User Manual.

22.1.1 EXISTING PROGRAMS

An important feature of the Tool Room Mill is its capabilities to run existing programs that run currently on the standard Fadal CNC control. The Feeds and Speeds of the program will have to match the TRM specifications, but as a whole the program will run without changes. Since the machine does not have a tool changer, a message to make a manual tool change will be requested when a tool change is executed.

22.1.2 TOOL CHANGES

The TRM does not have a tool changer, but tool changes in Auto and MDI can still be executed. This will allow programs to run unmodified if they were written for machines with a tool change present. When the control is asked for a tool change, a waiting message "TOOL CHANGE –" will blink on the screen. Before the machine will proceed with the program, the tool has to be changed manually. Use the TOOL IN/OUT button to release the tool in the spindle and then put the requested correct tool into the spindle. When the correct tool is in the spindle, the START button can be pressed to continue the program with the new tool inserted in the spindle.

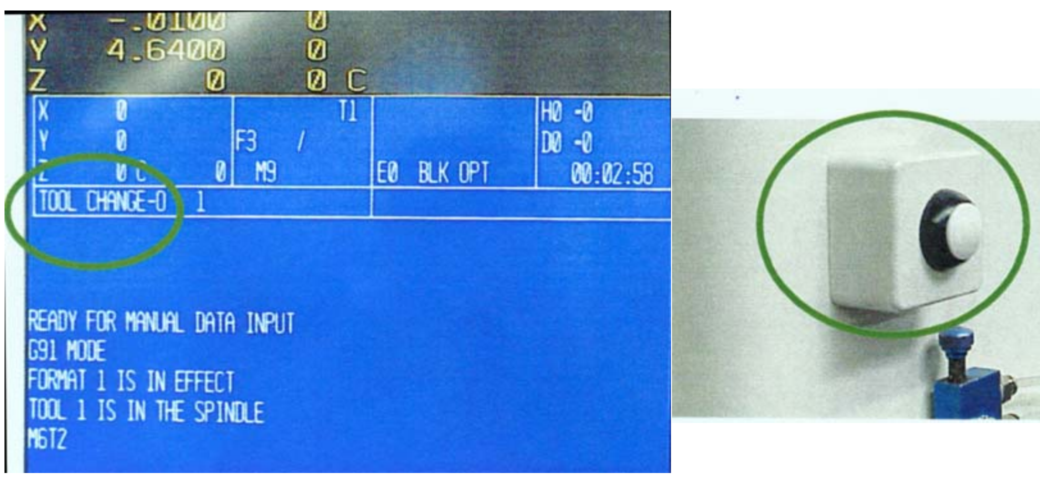


Figure 22-1: Tool Change

22.1.3 MANUAL MODE USING THE MPG HAND WHEELS

The Tool Room Mill uses a fully manual mode to move the machine's axes. Each axis has its own dedicated MPG, Manual Pulse Generator, hand wheel. For this reason, there is no axis selector switch on the machine. The X and Y axis MPG hand wheels are located parallel to the axis that they operate. The Z axis MPG hand wheel is located on the pendant of the machine. The Z axis MPG hand wheel has a dual function; it will also operate when MPD, Manual Pulse Dry run, is used. To see how to use the MPD function, read about it in the appropriate section of the User Manual.

To enter the Manual Mode of the machine, press the JOG key. The full display will be used to show current position. Using the axis increment selector switch, the stepping resolution of each of the MPG hand wheel can be modified from .0001”, .001” or .01” The axis load is display to the right of the current position of the axis as a percentage.

Each of the axes position can be zero while in this mode. By pressing the associated letter for the axis, the position display will be zeroed out. For example, pressing the Y key will result in the Y axis position to be cleared to zero. This feature can be helpful to set reference points as the machine is positioned to points on a part print.

Figure 22-2: Full position display during manual mode (Jog)

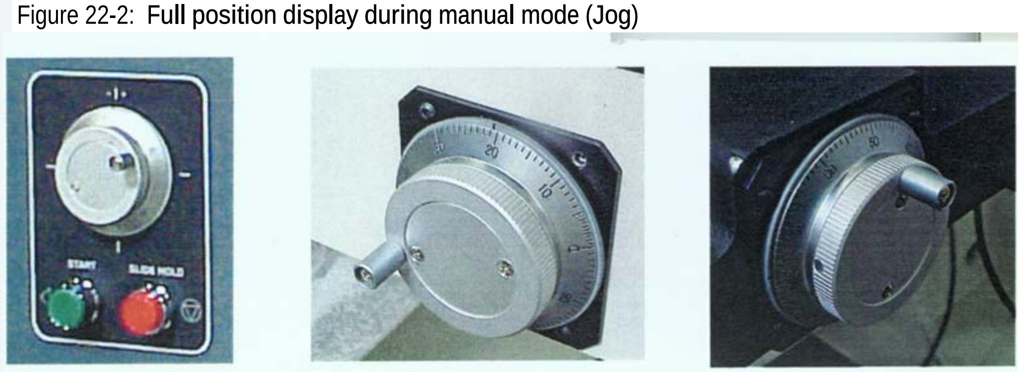


Figure 22-3: Manual MPG hand wheels for the X, Y, and Z axes

22.1.4 KEYBOARD

The TRM uses a notebook style keyboard with added functionality. Two of the keys are operated by holding the Shift key down and pressing the desired function key. They are identified by common color of the key text. The first of these two keys is the TOOL IN/OUT key, holding the SHIFT key down and then pressing the TOOL IN/OUT will operate the tool release or locking mechanism. The other key is the SPINDLE ON/OFF key, holding the SHIFT key down and then pressing the SPINDLE ON/OFF key will start the spindle at the last programmed spindle speed. Pressing the SPINDLE ON/OFF key by itself will turn off the spindle.

Figure 22-4: Keyboard

22.1.5 LEARN MODE

The Learn mode on the Fadal control has always been available, but when used on the TRM this feature can be very handy. As the machine is jogged to positions, the program can be built automatically. See the full description of this feature in the appropriate section of the Fadal User Manual.

22.1.6 SPINDLE AND AXIS MOTOR POWER LOAD PERCENTAGE

The current spindle and axis motor power load percentage is show on the control display while operating the machine in Manual mode or Jog mode. 100% is the continuous power output of the machine. When the power percentage is above 100%, the machine will only be able to maintain it for short periods of time. This time varies on the duty cycle above and below the 100% limit. When the machine is cutting material and the load goes above 100%, begin to back off on the load applied. This will prevent the machine from faulting from and overload.

Figure 22-5: Percentage

22.1.7 TOOL ROOM USER MACRO

For added capabilities, the TRM custom macro can be used to machine simple features. Using this custom macro will alleviate the need to move the machine manually or writing a CNC program. To use the Tool Room Mill Custom Macro program, the program will have to be loaded into memory and then accessed from the Quick Key menu. To get to this menu from any of the three Command Menu Editor menus, go to the Quick Key Menu by pressing Q. This will display the following menu at the bottom of the screen:

1-DRY RUN	4-SET FIXTURE	7-AXIS ZERO	0-CUSTOM MACRO
2-NEXT TOOL	5-SET LENGTH	8-READ	AUTO
3-ZERO RETURN	6-OFFSETS	9-PUNCH	

Press the 0 key to start the TRM Custom Macro loaded into program O9999. The custom macro program will then automatically be switched to and run in Auto Mode. Acknowledge the start of the program by pressing the START key. Note, there are minimum SETP (set parameters) requirements to use this special program which are as follows in SETP set to Format 1, tool table must be set to Diameter and be in Inch programming mode. In some cases, the Jog Mode may need to be entered first before running the TRM custom macro for the first time.

After pressing START to begin the program, the machine will move to the last SET home position. Once the TRM Custom Macro is started, the menu displayed will show the simple features that can be accomplished with the TRM Custom macro:

1 - DRILL	X - ZERO X AXIS
2 - TAP	Y - ZERO Y AXIS
3 - CIRCLE	Z - ZERO Z AXIS
4 - RECTANGLE	A - ZERO A AXIS
5 - MOVE TO XYZ	
0 - PARAMETERS	

PRESS KEY SELECTION?

At this menu, the individual MPG hand wheels are enabled. The machine may manually be moved with the wheels to desired locations. Use the X, Y, Z or A keys to set zero references. All functions on the menu will work as incremental moves.

22.1.8 DEFAULT PARAMETERS FOR TRM CUSTOM MACRO

To aid in repetitive tasks, there are a set of default parameters that are stored by the program that are shown in the following Parameter Menu:

1. SPINDLE SPEED/DIRECTION, S[1000.] M[3]
2. TOOL DIAMETER, D[0.125]
3. CUTTING FEED, F[20.]
4. CONVENTION/CLIMB CUTTING, [CLIMB]
5. PECK AMOUNT, Q[0.1]
6. Z PLUNGE FEED RATE F[3.]

PRESS KEY SELECTION?

The values shown between the brackets in the menu are the current default values set. These values are stored even if the TRM Macro program is stopped and exited. Select the parameter from the menu to change it. To keep certain parameters at their default value, simply press the ENTER key to use defaults.

22.1.9 DRILL FUNCTION

From the main menu, move the machine with either the MPG hand wheels or the Incremental Move function over the desired hole location. The spindle speed, feeds and pecking amount can be set in the parameters section. Position the tool tip at a known distance above the hole. The incremental depth of the hole is taken from the Z position when the Drill function is started.

INCREMENTAL DEPTH FROM CURRENT Z [-1.]?

The incremental depth shown in the brackets is the default depth. If this is the depth of the hole desired, press ENTER to continue with default value. Otherwise, type in desire depth from the current Z position.

S 1000. M 3
PLUNGE AT F 5.
CUT AT F 20.
DEPTH Z- 1.
* PRESS START TO CONTINUE OR ANY OTHER KEY TO ABORT *

If the settings for the Drilling function are the correct, press the START button to begin the cycle.

22.1.10 TAPPING FUNCTION

The tapping function will use a compression tapping cycle. From the main menu, move the machine with either the MPG hand wheels or the Incremental Move function over the desired hole location. Position the tool tip at a known distance above the hole. The incremental depth of the hole is taken from the Z position when the Tapping function is started.

INCREMENTAL DEPTH FROM CURRENT Z [-1.]?
SPINDLE SPEED [500.] = S
THREAD LEAD [.05]?

The incremental depth shown in the brackets is the default depth. If this is the depth of the tapping desired, press ENTER to continue with default value. Otherwise, type in desired depth from the current Z position. Do the same for the spindle speed to tap with and the thread lead to follow.

S 1000. M 3
PLUNGE AT F 5.
CUT AT F 20.
DEPTH Z- 1.
* PRESS START TO CONTINUE OR ANY OTHER KEY TO ABORT *

If the settings for the Tapping function are the correct, press the START button to begin the cycle.

22.1.11 CIRCLE AND RECTANGLE FUNCTIONS

The circle and the rectangle function can make either inside or outside shapes. They both use the diameter that is set up in the parameters, 0 from main menu. The position of the shapes are always from the XY center of the shape, i.e. center of circle. From the main menu, move the machine with either the MPG hand wheels or the Incremental Move function over the desired circle/rectangle location. Position the tool tip at a known distance above the shape. The incremental depth of the circle/rectangle is taken from the Z position when the function is started.

1=INSIDE OR 2=OUTSIDE CIRCLE?
INCREMENTAL DEPTH FROM CURRENT Z [-1.]?
DIAMETER OF CIRCLE [1.125]?
1=POCKET OR 2=MILL BORE?

Choose whether the shape will be either an outside or inside feature. When inside is chosen, there will be an addition prompt at the end whether to make a pocket or to just mill bore the hole. The values shown in the brackets are the default values, if these are the desire values, press ENTER to continue with default value. Otherwise, type in desired values.

CIRCLE DIAMETER = 1.125
S 1000. M 3
PLUNGE AT F 5.

CUT AT F 20.
DEPTH Z- 1.

* PRESS START TO CONTINUE OR ANY OTHER KEY TO ABORT *

If the settings for the function are the correct, press the START button to begin the cycle. Note, that the diameter of the tool is automatically compensated using the value in the parameters. The direction of the cutting is determined by the spindle direction and the desired direction of climb or convention.

INDEX

A