Section 21: Formats

Overview From the SETP Command the control has the option to use either FORMAT 1 or FORMAT 2.

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.

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.

**Format 1
Programming**
**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.

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

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

Format 2
Programming
Notes on Format 2 1) The safe block is mandatory at the beginning of the program.
Style Programming

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

2) The G28, with the axis designated to move, will bring the machine to the zero position established by the cold start command (CS) or to the zero position established by the SETX, SETY, SETZ, or SETH commands and/or fixture offsets, whichever was used last. The SET commands can be used to establish a zero position at any desired location. The program does not need to start from the machine home position. See also the section on establishing home position.

3) Fixed subroutines such as bolt hole pattern subroutines or pocket cutting routines from other machines cannot be used on this control. See the users manual for information on using fixed subroutines from the CNC 88 control.

4) The SETP command allows certain changes to default codes, diameter or radius input, and M7 M8 preferences.

5) When typing any variable (R0,R1 - R9) always type the positive or negative symbol before the value, for example, R1+.5 R2+2.34 R3+5.7124 R4-6.765. See *Section Six, Subroutines* for proper usage.

6) The H word will only pick up the tool length offset. The D word must be used to pick up the diameter or radius from the tool table.

7) Tool changes can be made from any XY position. The M6 code change will move the head to the tool position and make the exchange. The T word and the M6 can appear together, M6T# or T# M6, and usually do with this type of tool changer; however they may appear on separate lines.

8) Each line requires a sequence number. Sequence numbers use an N####.### word. Decimal points can be used to input lines between existing line numbers. The sequence numbers can be added when the program is typed at the computer, but the numbers are added automatically when the program is transferred to the control with the TA (tape input) command. So it is possible to type the program without line numbers and have the control add the line numbers automatically. The NU command is used to renumber the lines.

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

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

If decimals 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.

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

Default Parameters,
SETP Command
FORMAT 1 & Changing from FORMAT 1 (FADAL mode), to FORMAT 2 (6M, 10M, 11M
FORMAT 2 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.

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.

```
                                                     PAGE 1 OF
*FORMAT: 1              BAUD RATE: 2400      SPINDLE AFTER M6: N
 AXES: X,Y,Z           TRAVEL: X40 Y20 Z20  PENDANT: MOUNTED
 DEFAULT: G0
 DEFAULT: G90
 DEFAULT: G17          TOOL CHANGER CAP: 21 IMM. FIXED CYCLE: N
 RPM FACTOR: 5         SPINDLE TYPE: 10,000 ORIENTATION FACTR:



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



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

Figure 21-1 Format 1 Parameter Settings

```
                                                     PAGE 2 OF
*DEFAULT: INCH         M8=FLOOD   M7=MIST   3 PHASE > 5% LOW: N
 PU FORMAT: FILE
 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 21-2 Format 1 Parameter Settings (Continued)

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




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


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

Figure 21-3 Format 1 Parameter Settings (Continued)

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)

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




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


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

Figure 21-6 Format 2 Parameter Settings (Continued)

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.

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.

DEFAULT: G0 ENTER THE DEFAULT VALUE

1) G0
2) G1

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

DEFAULT: G90 ENTER THE DEFAULT VALUE

1) G90
2) G91

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

DEFAULT: G17 ENTER THE DEFAULT VALUE

1) G17
2) G18
3) G19

This parameter is used to select the default machine plane.

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.

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.

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.

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.

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.

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.

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.

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

Section 21: Formats

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.**

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.

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.

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.

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.

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.

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.

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

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.

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.

PALLET   DO YOU HAVE A PALLET CHANGER?

1) NO
2) YES

Select the option appropriate for the machine.

ASPECT   ENTER THE ASPECT RATIO OF Y TO X

THE RATIO MUST BE BETWEEN 25 AND 150

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.

BINARY BUFFERS: *SELECT THE NUMBER OF BINARY BUFFERS FOR CNC LOOK-AHEAD.*
255

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.

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.

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.

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.

3 PHASE 5% LOW: *IS YOUR 3 PHASE POWER MORE THAN 5% LOW?*
NO

1) YES
2) NO

The selection chosen is based upon the building power supply.

HIGH TORQUE DO YOU HAVE THE HIGH TORQUE OPTION OR RIGID TAP OPTION?

1) NO
2) YES

This parameter is set at the factory.

CMD MENU *TURN COMMAND MENUS:*

1) OFF
2) ON
3) TOGGLE ON WITH SPACE BAR

The operator may select the command menu structure.

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.

OVERLOAD *ENTER THE MOTOR OVERLOAD FACTOR (2 IS STANDARD)*

THE FACTOR MUST BE BETWEEN 1 AND 8

SCREW SELECT THE SCREW PITCH TYPE?

1) INCH PITCH
2) 8/10MILLIMETER PITCH

IPM SELECT THE MAX RAPID FEED RATE?

1) 400 IPM   4) 800 IPM
2) 500 IPM   5) 900 IPM
3) 700 IPM

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.

Z TAP AGAIN SELECT THE Z GAIN DURING RIGID TAPPING?

1) NORMAL
2) MEDIUM
3) HIGH

Only a factory trained representative should modify this parameter.

VECTOR DO YOU HAVE VECTOR DRIVE (10 RPM MIN)?

1) YES
2) NO

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.

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.

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.

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.

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.

GO 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.

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.

Parameters Applicable to FORMAT 2 Only

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.

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).