CHAPTER 3
GENERAL INPUT

The Fanuc 11 machining center control is designed to operate from data supplied from perforated tape.

The control uses a word address and variable block length format for tape input. A block of data is comprised of a number of letter address words that command a desired machine action. A word (command) is one letter address and the designated number of numeral digits which establishes the value of the word. Thus, N00210 is one word of information.

3.1 FORMAT DETAIL

Tape input should be programmed to the following conventions. The G word must follow the N word.

INCH: N5 G3 X + 3.4 Y + 3.4 I + 3.4 J + 3.4 K + 3.4 R + 3.4 Z + 3.4 U + 3 W + 3.4 F3.2 S4 T2 D2* H2* M3 A5.3 B5.3 C5.3, O4, #3

METRIC: N5 G3 X + 4.3 Y + 4.3 I + 4.3 J + 4.3 K + 4.3 R + 4.3 Z + 4.3 U + 4 W + 4.3 F4.1 S4 T2 D2* H2* M3 A5.3 B5.3 C5.3, O4, #3

As an example of the above, consider the X + 3.4 in the inch system. The 3 represents the number of programmable digits to the left of the decimal point, the 4 represents the number of programmable digits to the right of the decimal point. S4 is an undimensioned word representing spindle RPM and is simply an S followed by four digits.

*D2 and H2 (100 total)

3.1.1 Decimal Point Programming

Decimal point programming applies to all dimension letters (X, Y, Z, R, I, J and K) and to the feedrate (F). Up to seven digits can be programmed for the dimensional commands and up to five digits for feedrate.

With decimal point programming, leading and trailing zeros need not be used for the command words. Only the significant digits and decimal point of a dimension command are programmed.

CAUTION: If a decimal point is left out of the dimension command, the control assumes the decimal is four places to the left of the last programmed digit.

WRONG	X10 equals X.0010
RIGHT	X10. equals X10

3.1.2 Inch to Metric/Metric to Inch Conversion

Inch to metric/metric to inch conversion differs depending on which side of the control, NC or CAP, the programmer/operator is working in.

A. NC Side of Control (for both 11MA and 11MF)

The G21 code converts control format from inch to metric. G20 converts the format from metric to inch. These codes must be entered in MDI and then the machine re-referenced in order to take effect.

B. CAP Side of Control (only applicable to 11MF)

When working on the CAP side of the control, the G21 or G20 code is still necessary to convert the control from inch to metric or metric to inch, but several additional steps are needed. Parameter write enable must be turned on and parameters 1 and 58 must be changed. This procedure is explained step by step in the Operator Instruction Charts at the back of Monarch's Operator's Manual.

3.2 BASIC PROGRAM CODES

3.2.1 Sequence Number (N5)

The sequence number is a five digit number preceded by the letter N that identifies each block of the program. We recommend that blocks be numbered in ascending order in increments of 10 to allow for future editing and to simplify sequence number search.

EXAMPLE

```
N50 G0 G90 G54
N60 G49 Z0 H0
N70 G28 Z0
N80 G0 X.5 Y11. T2 M6
N90 M3 S1000
```

With the program written as above, up to nine blocks of new information could be added between each block.

3.2.2 Preparatory Commands (G3)

The preparatory command is a three digit number preceded by the letter G to establish the type of operation. Two types of codes exist:

- one shot or non-modal
- modal or program modal (in effect until another G code of the same group is specified)

G90 or G91 - absolute or incremental notation for X and Y coordinates.

G80/G0 - cancels canned cycles

G0 - cancels contouring and canned cycle codes; forces 600 IPM automatic traverse in X and Y

G60 - single direction positioning; automatic backlash takeup, one shot

G81 thru G89 - canned cycles that generate complete Z axis motions such as drilling, tapping, boring, etc.

G1 thru G3 - contour milling codes.

G4 - programmed dwell.

SEE CHAPTER 5 for complete listing of preparatory commands.

3.2.3 Axis Commands and Directions (see Figures 3-1 and 3-2)

1. X axis command (X + 3.4 inches or X + 4.3 metric) The X letter address and up to seven digits command the X axis movement.

2. Y axis commands (Y + 3.4 inches or Y + 4.3 metric) The Y letter address and up to seven digits command the Y axis movement.

3. W axis command (W + 3.4 inches or W + 4.3 metric) The W axis is the vertical line in which the arm travels. The W letter address and up to seven digits are used to command the W axis movement.

4. Z axis command (Z + 3.4 inches or Z + 4.3 metric) The Z axis is the vertical line in which the spindle travels. The Z letter address and up to seven digits command the Z axis movement. The maximum Z axis travel is 10 inches.

5. R plane command (R + 3.4 inches or R + 4.3 metric) The R value is programmed to terminate the traverse movement of the Z axis. R is that plane at which the programmed feedrate is to be actuated. The R letter address and up to seven digits are used for this command. This code is ony used with the canned cycle commands.

6. I arc center offsets (I + 3.4 inches or I + 4.3 metric) The I letter address and up to seven digits are used. This code is only used with the contour milling preparatory commands, G2 and G3. The I value is the incremental distance as measured parallel to the X axis from the arc starting point being generated to the center of the arc.

7. J arc center offset (J + 3.4 inches or J + 4.3 metric) The J letter address and up to seven digits are used. This code is only used with the contour milling preparatory commands, G2 and G3. The J value is the incremental distance as measured parallel to the Y axis from the arc starting point being generated to the center of the arc.

8. K arc center offset (K + 3.4 inches or K + 4.3 metric) The K letter address and up to seven digits are used. This code is only used with the contour milling preparatory commands, G2 and G3. The K value is the incremental distance as measured parallel to the Z axis from the arc starting point being generated to the center of the arc.

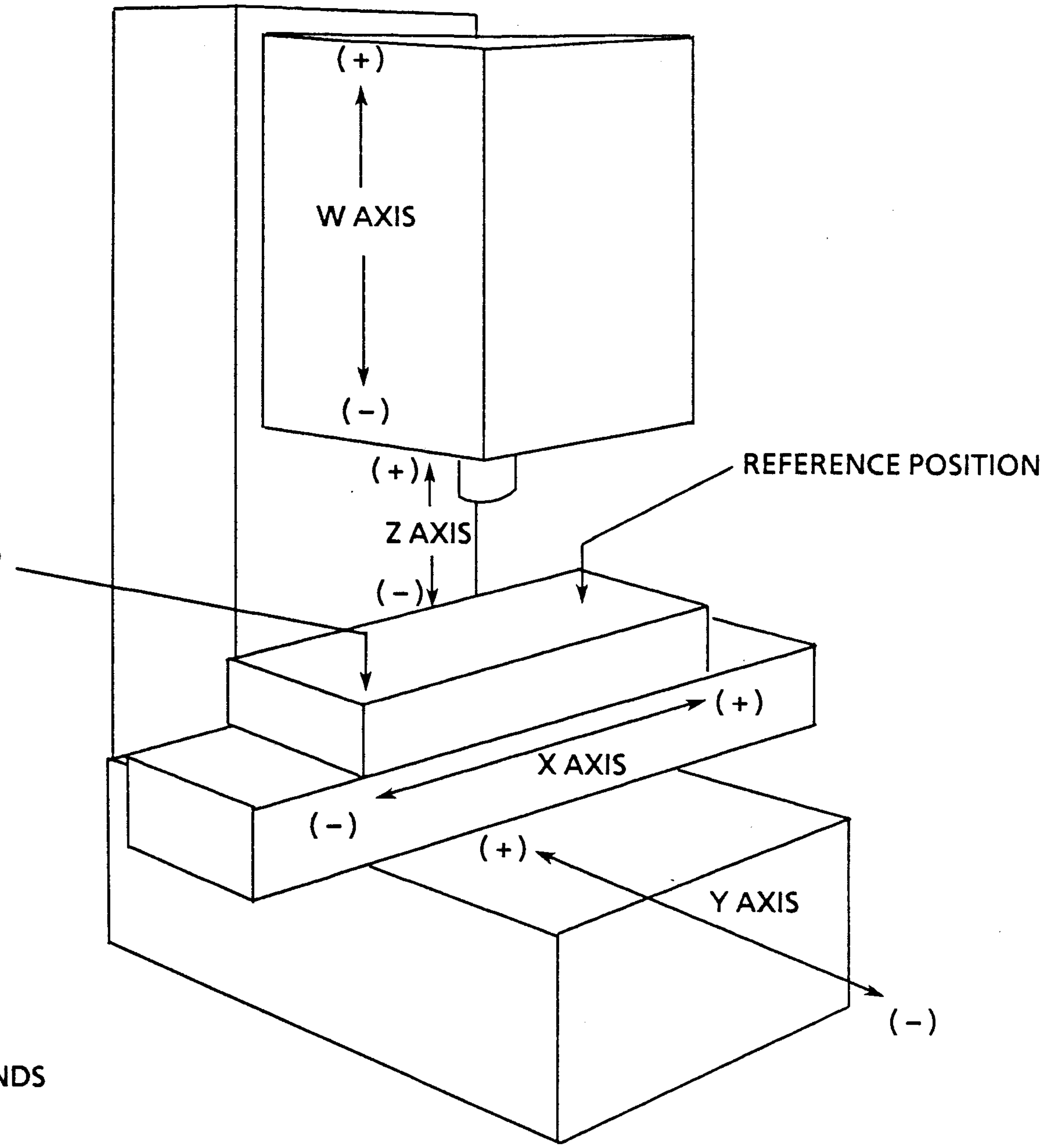

FIGURE 3-1
AXES COMMANDS

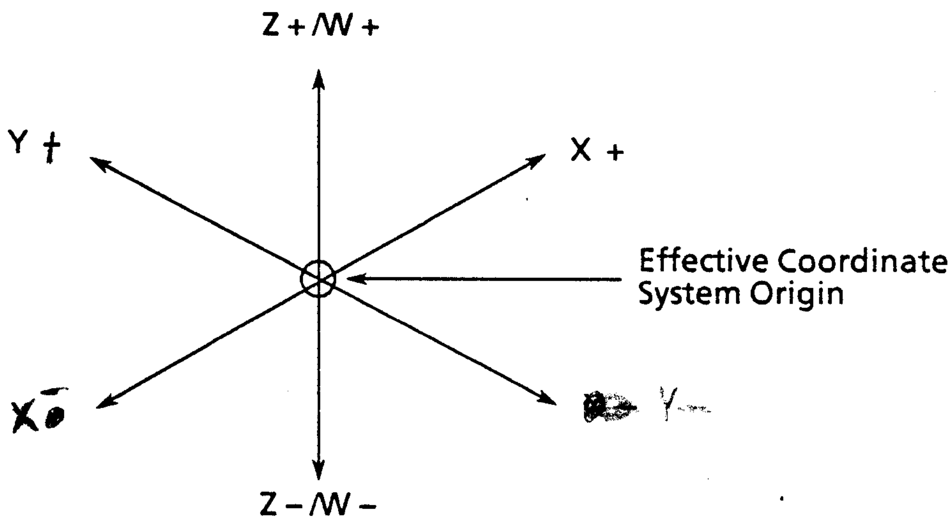

FIGURE 3-2
AXES DIRECTIONS
(X-Y-Z/W Quadrant System)

3.2.8 Tool Compensation Memory

Offset Number	LENGTH (Hxx)		RADIUS (Dxx)	
	Geometry (TLG) Comp	Wear (TLW) Comp	Geometry (TRG) Comp	Wear (TRW) Comp
1	- 4.0000	- .0000	0.0	0.0
2				
3				
•				
•				
99				

Memory for geometry compensation as well as tool wear compensation is stored separately in tool offset (compensation) memory.

The net offset activated is the sum of both geometry and wear; i.e. for H1 effective tool length compensation would equal -4.0000.

The same holds true for tool radius compensation. For more information on the use of tool radius compensation values, see Chapter 8.

100 total radius (D) and length (H) offsets are provided with a range of ± 99.9999 inches (G70) or + 999.999 mm (G71).

The tool compensation memory may be set via MDI/CRT or with the Auto Offset pushbutton.

3.2.9 Miscellaneous Functions (M3)

A miscellaneous function command is the letter address M and up to three digits which command such functions as program stop, spindle start, coolant on, etc. See Chapter 6.

3.2.10 Spindle Speeds (S4)

The spindle speed command is an S letter address followed by up to four digits from 20 RPM to 3500 RPM.

Examples

RPM	Program Command
45	S45
280	S280
1400	S1400

The spindle speed command should be programmed before the first cutting block; spindle speed remains active until a new speed is programmed.

For normal machine operation, speeds from 20 to 923 RPM are in low gear and speeds 924 through 3500 are in high gear.

When tapping, 20 RPM through 199 RPM are in low gear and speeds from 200 RPM through 3500 RPM are in high gear.

<u>High Speed Spindle Direct RPM Programming (S4)</u>
(only for machines with high speed spindle)

Machines with the high speed spindle are programmed exactly as explained above except that the spindle speed range is from 20 RPM to 5200 RPM.

Low gear is 20 to 999 RPM and high gear is 1000 to 5200 RPM for normal machine operation. When tapping, low gear is 20 through 199 RPM and high gear is 200 through 5200 RPM.

3.2.11 Program Number (O4)

A four digit program number following address O (oh) is used to index and differentiate one program from another.

3.2.12 Variable (#3)

A variable index number is composed of a code sign (#) and numeric index number; for example, #110.

SEE Chapter 10 "Variables, Math Capability, Program Flow Word Commands and Report Generation".

CHAPTER 4
WRITING THE PROGRAM

A program is a series of part program blocks stored in the control's memory and called up for execution by the operator.

Programs are stored in memory automatically when the control recognizes the program identification block (Oxxxx). Programs must be defined (stored) before they are called for execution.

Every program must have a program identification number which must be the first block of the program. This program identification block would look like this:

O1234 (BOLT CIRCLE)

The identification number represented by 1234 in the above example can be up to four digits long; the description remark represented by (BOLT CIRCLE) is not required, but if included can be up to 16 alphanumeric characters long.

The program must end with an M30 or M2 block: N50 M30.

Programs are written in a modular form. Modules are one or more blocks used to perform independent operations. One module may be the necessary blocks for a safe start. Another module may contain the blocks required for some Z axis cycle such as drilling, tapping or boring. If at the end of the Z axis cycle module the Z axis is at final depth in preparation for milling, the next module would contain the necessary blocks to perform the X and Y milling movements (G1, G2, or G3); and so on.

4.1 SAFE START MODULES

Safe start modules are blocks of information which contain all the necessary data needed for starting on a particular series of operations. Normally, every tool change is in a safe start series. **NOTE:** All safe start blocks must be written in the absolute (G90).

The first safe start module for any program must include the W axis (arm)move in order to insure the arm is in the correct position prior to any axis moves. The following examples first show the safe start module including the W axis move, then the safe start module that can then be used throughout the remainder of the program.

4.1.1 The tool change and axis moves are to be performed at the same time. The only variables would be the N, X, Y, S and T values.

Example showing the safe start module with W axis move:

```
N10 G0 G90 G54
N20 G49 Z0 H0
N30 G28 Z0
N40 G60 W – 2.5
N50 G28 Z0
N60 G55 X15. Y5. T1 M6
N70 M3 S149
N80 G43 Z.1 H1
```

In block N10 (G0) caused traverse mode, (G90) selects absolute positioning, (G54) selects machine coordinate system.

N20 (G49 Z0 H0) cancels tool length compensation.

N30 (G28 Z0) selects reference point return. The only axis motion in this block will be a Z axis return to reference. The action must precede a tool change.

N40 (G60 W – 2.5) selects single direction positioning and moves the W axis to – 2.5".

N50 (G28 Z0) ensures Z axis is at its reference position.

N60 (G55) selects work coordinate system, (X15. Y5.) moves the X and Y axis to the given positions relative to G55. (T1) selects tool number; (M6) requests a tool change.

N70 (S149 M3) starts spindle rotating clockwise at 149 RPM's.

N80 (G43) selects tool length offset add which causes a tool offset value to be added to the Z position /command. (Z.1) initiates axis movement to R point. (H1) specifies offset index number. This move occurs at traverse.

4.1.2 In this example, the tool change is to be prior to the X and Y axis movements.

First safe start module in the program:

```
N10 G0 G90 G54
N20 G49 Z0 H0
N30 G28 Z0
N40 G60 W - 2.5
N50 G28 Z0
N60 T1 M6
N70 G55 X15. Y5.
N80 S149 M3
N90 G43 Z.1 H1
```

In block N10 (G0) caused traverse mode, (G90) selects absolute positioning, (G54) selects machine coordinate system.

N20 (G49 Z0 H0) cancels tool length compensation.

N30 (G28 Z0) selects reference point return. The only axis motion in this block will be a Z axis return to reference. The action must precede a tool change.

N40 (G60 W – 2.5) selects single direction positioning and moves the W axis to – 2.5".

N50 (G28 Z0) ensures Z axis is at its reference position.

N60 (T1) selects tool number; (M6) requests a tool change.