# 527F CNC Control

**User Manual**

Calmotion LLC
21720 Marilla St.
Chatsworth, CA 91311
Phone: (818) 357-5826
www.calmotion.com

Section 1: Summary

# Section 1: Summary

## NC Word Summary

**Table 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 |

| NC Word Summary | Definition |
|---|---|
| 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 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 |

## G Codes

**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.

**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 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 |

Section 1: Summary

**Table 3: G Code Summary Table (Continued)**

| Code | Group Designation | Modal | Non Modal | Description |
|------|-------------------|-------|-----------|-------------|
| 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 |
| 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 |

Section 1: Summary

| Code | Group Designation | Modal | Non Modal | Description |
|------|-------------------|-------|-----------|-------------|
| 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 |
| 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.

**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, and after M2.

**Reset**    Format 1 will default to this status automatically. Format 2 will use this default status after the HO 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) command will reset and use the default status from the SETP parameters in both format 1 and 2.

**Table 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 command.

## M Functions

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

**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.

Section 1: Summary

***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 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 |
| 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 |

Section 1: Summary

**Table 5: M Function Summary Table (Continued)**

| Code | Starts with Motion | Starts after Motion | Modal | Non Modal | Description |
|------|------|------|------|------|-------------|
| 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 |
| M64 | - | - | Yes | - | Activate MP8 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) |

Section 1: Summary

**Table 5: M Function Summary Table (Continued)**

| Code | Starts with Motion | Starts after Motion | Modal | Non Modal | Description |
|---|---|---|---|---|---|
| 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 |

**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 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 command at the VMC.


## Program Numbers, Protection & Storage

**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).

**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.*

Section 1: Summary

*N7 X1.F60.*
*N8 M5 M9*
*N9 G90 G0 H0 Z0*
*N10 M2*

**EXAMPLE:**  Format 2ONLY

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*

## Program Protection

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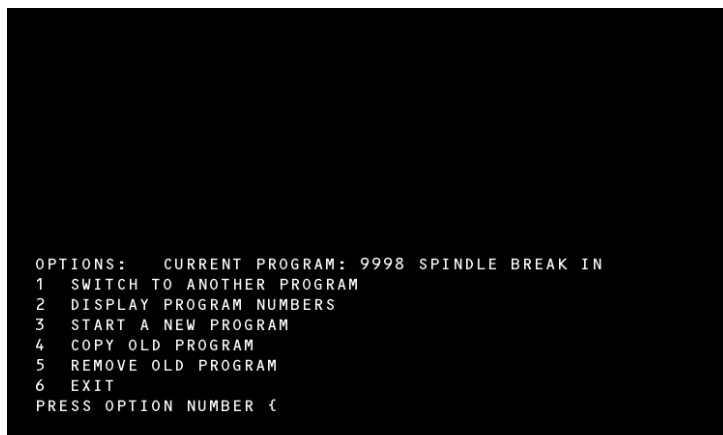
A NOEDIT program will not allow commands CH, DE, IN, NU, NE, CO, LE, PU or from PA: C, I, O, N, and R (see the PA command).

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.

**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.

**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.

**Program Storage**  Programs stored in memory can be managed by using the PR command (see the COMMAND SECTION). The PR menu allows the operator to switch, display, start, copy and remove programs. Enter PR command to see the following menu:

```
OPTIONS:    CURRENT PROGRAM: 9998 SPINDLE BREAK IN
1   SWITCH TO ANOTHER PROGRAM
2   DISPLAY PROGRAM NUMBERS
3   START A NEW PROGRAM
4   COPY OLD PROGRAM
5   REMOVE OLD PROGRAM
6   EXIT
PRESS OPTION NUMBER {
```

*Figure 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

Section 1: Summary

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.

**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 command to begin keyboard input after the line containing the program number. Alternatively, use the PA 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.

**Format Classification Sheet**

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

**Machine**

Vertical Machining Center (VMC).

**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

**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*

Section 1: Summary

*A+4.3*
*C+5.1*
*M2.1 H2 T2 D2*

*F4.2 S5.1 L4 P4 O4*

**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

**M Functions 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.

**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)

**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 interger. 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

Section 1: Summary

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

**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.

**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.

**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).

**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"

**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"

Section 1: Summary

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"

**Geometric Relationship**  X, Y, Z, C per RS-267-A

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

Section 1: Summary