

### 2.9.33.2 Restrictions on High-Speed Cutting Feature

(1) When the high-speed cutting feature is made valid by parameter setting, the part program memory capacity is decreased to about 1/2 (see Table 7.1.1 and

(2) The program between G101 (or G102); and G100 ; should follow the restrictions shown below:

a. The addresses that can be specified are as follows:

O, /(slash), N, G, X, Y, Z, I, J, K, R, D, H, M, L, F

The 4th-axis address cannot be specified.

b. The G codes that can be specified are as follows:

G00, G01, G02, G03, G09, G40 through G42, G43, G44, G45 through G48, G49, G90, G91

If any G code other than above is specified, program error "142" will be caused.

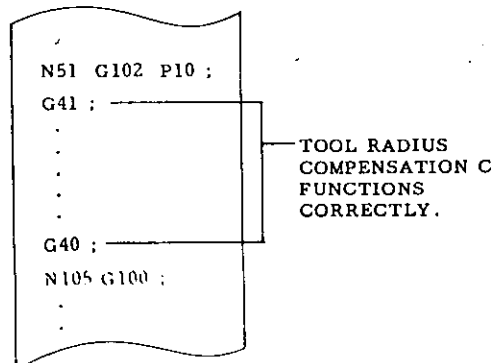
c. The M code that can be specified are as follows:

M90 through M99

If any M code other than above is specified, program error "142" will be caused.

#### Considerations and Remarks

(1) Do not enter the high-speed cutting mode (G101 or G102) with the tool radius compensation C mode being on. Cancel (by G40) the compensation C mode. Otherwise, program error "024" will be caused. With the high-speed cutting mode being on, the compensation C mode can be entered. However, before canceling the high-speed cutting by G100 ; command, cancel the compensation C by G40 command. Otherwise, program error "024" will be caused.



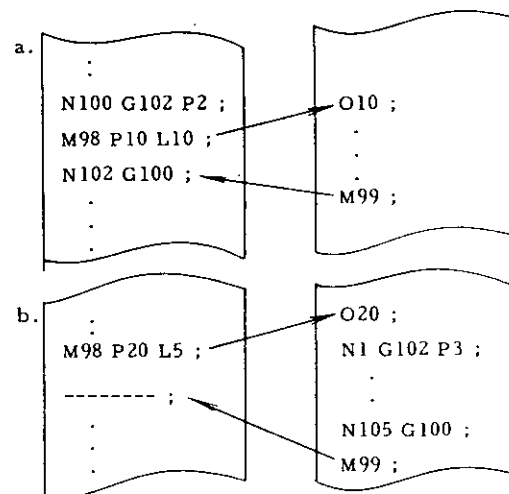
(2) The tool length compensation and tool position offset features are not affected by G100 through G102 commands and therefore always valid.

(3) The execution of G102 P... ; command, thereafter, needs the use of the data processed and registered in the first execution, so that the following points should be taken into account:

a. The tool offset amount to be used should be the same as that used in the first execution. To change the tool offset amount, reset the registered data and execute the command again for the first time (see 7.4, (7)).

b. If there is a single coordinate command of absolute value in a program of memory processing mode, the absolute coordinate value immediately before G102 P... ; command must be in the same position as that used in the first execution. Otherwise, the correct position cannot be obtained as specified by the absolute command in the program.

(3) Specifying a subprogram (M98) is as follows: as follows:



(4) G101 ;  
G101 P... ;  
G100 ;

These commands should be specified independently.

(5) The blocks between G101 (or G102 P) ; and G100 ; are handled as a single block with regard to the single block stop feature. That is, the single block stop feature is disabled between these commands.

(6) During the data processing and registration of the high-speed cutting program, the display on the CRT screen is as shown below. The function keys are automatically put in the COM display mode, displaying the screen shown below. At this time, the NC operator's station keys except for RESET are disabled.