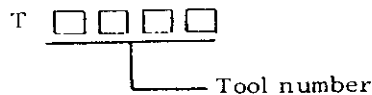


2.6.2 T 4-DIGIT⁺ PROGRAMMING

Four digits following the address T specifies the tool number.



- Leading zeros may be omitted.
- This tool code is the same as the T 2-digit codes, except for the increased number of digits.

2.7 TOOL COMPENSATION

2.7.1 OUTLINE OF TOOL COMPENSATION

The tool compensation function is in the following three modes.

- Tool length compensation
This function is for compensating for differences in tool length, and is effective in the Z axis direction. Specified length compensation becomes effective from the block in which G43 or G44 is programmed together with an H code. It is cancelled with H00 or G49.
- Tool position offset (for simple compensation for tool radius)
This function is for compensating for errors in machined dimensions to be introduced by the radius of tools. It is effective in the X, Y, and Z (4th⁺) axis directions. It is effective only for the block in which G45 - G48 is programmed together.
- Tool radius compensation C⁺ (for compensating for tool radius effects with complicated machining contours)
This function is for compensating for the tool radius effect with any given machining contours. It is effective in X-Y, Y-Z, and Z-X planes. It becomes effective from the moment G41, or G42 is commanded together with a D code, and is cancelled by G40.

NOTE: For details of these compensations functions, refer to 2.9 PREPARATORY FUNCTION (G-FUNCTION).

2.7.2 TOOL OFFSET MEMORY

For the three groups of offsets, all the necessary offset values must be stored in memory beforehand.

Up to 99 offset values can be stored in the tool offset memory.

Offset value storage: 99 max.

The setting range of offset values is as follows.

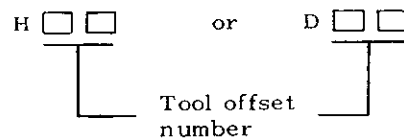
	Linear axis	Rotary axis [†]
Metric input	0 - ± 999.999 mm	0 - ± 999.999 deg
Inch input	0 - ± 99.9999 inch	0 - ± 999.999 deg

Listed input values do not depend on metric/ input output system.

For the procedures of storing values into memory, refer to 4.3.5 WRITING OF TOOL OFFSET VALUE.

2.7.3 H- AND D-FUNCTION (H, D CODES)

Two digits, following the address H or D, specify tool offset numbers.



The tool offset numbers 01 through 99 directly correspond to the 99 offset-value memory numbers. That is, when certain numbers are designated, the corresponding offset values stored in the offset memories will be used to offset the tools.

Tool offset numbers 00 (H00 or D00) have different meanings depending on the respective offset functions. For details, refer to the descriptions on the respective G functions.

H- and D-codes must be used properly according to their functions.

Code	Function
H code	Tool length offset
D code	Tool position offset, Tool radius compensation

The tool offset numbers 01 through 99 can be used freely in combination with the both H and D codes.