The actual index positions corresponding to the respective B codes depend on the machine tool builder. For this, refer to the specifications of the machine tool builder. When a B-function is given together with a move command in one block,

- · the B command is executed simultaneously with the move command, or
- B command is executed after the execution of the move command,

depending on the design of the machine tool. For this refer to the specifications of the machine tool builder.

B codes are modal. When one B code is given, it remains effective until another B-command is given.

NOTES:

- B function standard interface is in 3-digit BCD output.
- With MDI operation on NC panel, "B" is used to specify address for B codes. Therefore, when the control has B-function, the 4th axis control cannot be added.

2.9 PREPARATORY FUNCTION (G-FUNCTION)

2.9.1 LIST OF G CODES AND GROUPS

An address character G and up to 3 digits following it specify the operation of the block.

Ordinary G codes are either non-modal G codes marked with * or modal G codes belonging to groups 01 through 15. Those G codes belonging to the division B are included in the basic specifications.

- The G codes belonging to groups 01 through 15 are modal, remaining effective when once commanded until other G codes in the same group will be commanded.
 - The G codes in the * group are non-modal, and are effective only for the block in which they are commanded.
- The G codes belonging to groups 01 through 15 may be programmed twice or more in the same block. However, when different G codes in the same group are programmed, the last appearing G code only is effective.

- The G codes belonging to the * group can not be programmed twice or more in a block. They must be programmed only once in a block of its own.
- When a G code belonging to the 01 group is commanded during a canned cycle (G73, G74, G76, G77 and G81 through G89), the canned cycle is cancelled and these codes in the group 09 becomes G80.
- When the RESET key is depressed during the execution of a tool compensation C (G41, or G42) or a canned cycle, they become respectively G40 or G80 which cancells the programmed commands.
- G43, G44, G49 and G45 through G48 belonging to the * group can be programmed together with the following G codes in the 01 group in the same block.

	Combination G code
G43, G44, G49	G00, G01, G60
G45 - G48	G00, G01, G02, G03, G60

- The transition state of the following G codes can be changed by setting of parameters.
- The G codes belonging to the following groups can be specified as to the state immediately after the application of supply power. (#6005D0 - D4)

Group	Initial state	Parameter
01	G00 or G01	#6005D2
03	G90 or G91	#6005D0
05	G94 or G95	#6005D1
08	G43, G44 or G49	#6005D3, D4

- G codes in the 01 group may selectively be changed to G00 or kept unchanged after resetting. (#6005D6)
- During the execution of G92, Display may selectively be made. (#6005D5).