2.9.7 EXACT STOP (G09, G61, G64) (CONT'D)

· Exact stop mode (G61)

When once G61 is commanded, all the following blocks will be completed in the Error Detect On mode before proceeding to the next block.

· Exact stop mode cancel (G64)

This G command is for cancelling the effect of G61.

NOTES:

· In the Error Detect On mode, the program proceeds to the next block only after the number of servo delay pulses is found to have decreased below a permissible limit following the complete distribution of circular interpolation command pulses.

· In the G09 and G61 off modes, the program proceeds to the next block immediately after the complete distribution of the pulses of ordinary linear and circular interpolations, and therefore, because of the servo delay, tool pathes are rounded at the corner. This mode is called "Error Detect Off" mode.

· For rapid traverse, the Error Detect On and Off modes are controlled only by G00 and G06, and not by the above G codes.

2.9.8 TOOL OFFSET VALUE DESIGNATION (G10)

With a G10 command, correction of tool offset values and work coordinate system can be made as follows.

· Designation of tool offset value (G10)

Normally, tool offset values are written in by MDI. On the other hand, with a program G10 P··· R··· ;, (where P = tool offset number and R = tool offset value), any programmed offset values can be replaced by a designated value.
When G10 is commanded in the G90 mode, R is regarded as an absolute value.
When G10 is commanded in the G91 mode, R is regarded as an incremental value.

· Changing work coordinate system†

Corresponding to G54 through G59, separate work coordinate systems are set up as setting data in advance.

G10 Q2 Pn X··· Y··· Z··· α··· ;

(where Q2 is used to discriminate from tool offset value designation and a means to set up a work coordinate system. Pn (n = 1 to 6) corresponds to the work coordinate system n to be set up.)

For P1··· G54

For P2··· G55

.
.
.

For P6··· G59 corresponds.

With the above command, data of any desired work coordinate system can be changed.

2.9.9 CIRCLE CUTTING (G12, G13)†

This is a canned cycle includes a complete series of movements for machining a circle in a single block. It includes the following functions.

· Format

G12(G13) I··· D··· F··· ;

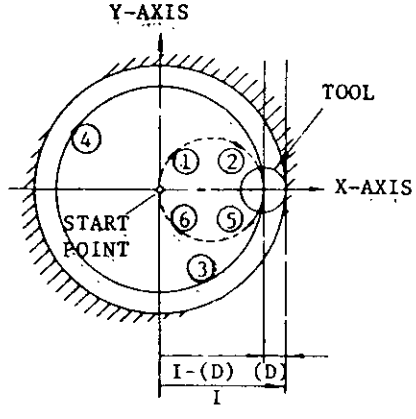

Fig. 2.19

· Designation of rapid traverse section R

G12 (G13) I··· R··· D··· F··· ;

With this command, a circular bore is machined as shown below. Numerals following an address character R specifies rapid traverse sections.