. G00 Z-4000;

G13 I-5000 J700 D15 F300; D15 = -8.0 mm

G00 Z4000;

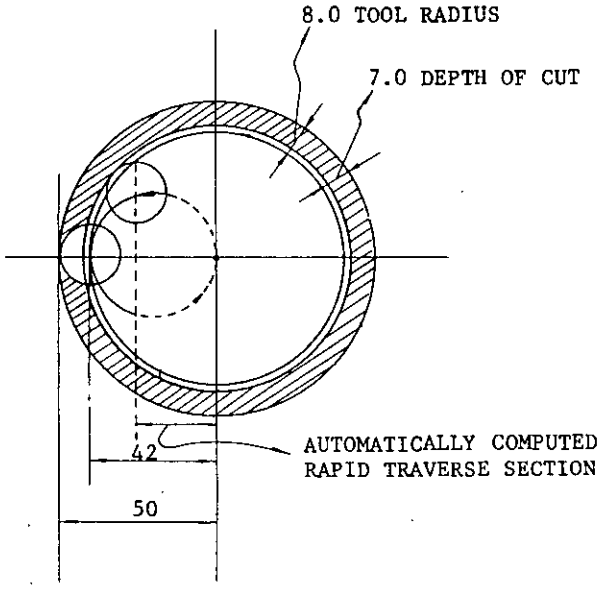


Fig. 2.25

2.9.10 PLANE DESIGNATION (G17, G18, G19)

The plane for making circular interpolation and tool radius compensation is designated by G codes G17/G18/G19.

G17: XY plane
G18: ZX plane
G19: YZ plane

When the 4th axis† is selected, the following planes are newly added.

G17: XY plane or Xα plane
G18: ZX plane or Zα plane
G19: YZ plane or Yα plane

α means U, V or W axis.

The move command in each axis can be programmed regardless of the plane designation by G17/G18/G19.

For example, if

G17 Z··· ;

is designated, motion is on Z axis.

The plane for making tool radius compensation by command G41 or G42 is univocally determined by G17, G18 or G19. It is not possible to designate compensation plane including the fourth axis.

The XY plane (G17) is selected when the power is turned on.

2.9.11 INCH/METRIC DESIGNATION BY G CODE (G20, G21)†

Unit of input data are selectively specified by the following G codes between metric and inch.

G code	Input unit
G20	Inch
G21	Metric

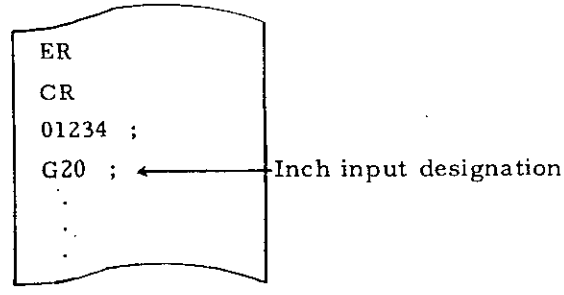
These G codes are programmed at the leading end of a block of its own. If one these G codes are commanded, the units of all the following motions are changed afterwards.

- subsequent programs
- tool offset values
- part of setting parameters
- part of manual movements
- displays

NOTES:

- When G20 or G21 is commanded, the setting of inch/metric selection is changed. Therefore, the state of G20/G21 at the time of power application depends on the setting by parameter #6001 D0.

EXAMPLE



- When G20/G21 selection is commanded in the program, take the following procedure beforehand.

A. When work coordinate system (G54 to G59) is used, return it to base coordinate system.

B. Cancel all tool compensation command. (G41 to G48)