#6552 ··· storing X coordinate value
#6553 ··· storing Y coordinate value
#6554 ··· storing Z coordinate value
#6555 ··· storing 4th coordinate value

These data can be treated as coordinate data in user macros.

When a skip signal is not given in spite the execution of G31 by setting (#6004D0), the program moves on to the next block automatically.

2.9.18 THREADCUTTING (G33)†

Provided that the machine is equipped with a spindle pulse generator generating reference pulses, and a synchronous feed function (G95), screw threads can be cut with this function.

G33 Z··· F··· ;

With this command, threadcutting is made in the Z direction at a lead of F.
Lead range is shown below.

	Range of lead
Metric	0.01 - 99.99 mm
Inch	0.001 - 3.936 inches

However, lead is subject to the following restriction from the spindle speed.

F(mm/rev) x S(rpm) ≦ 24000(mm/min) or a clamp value

NOTES:

- G33 may be programmed with two or more axes simultaneously as for example in G33YyZzFf ,. In this case, the tool moves along the axis for which the longest distance is specified at the lead specified by F.
- G33 is a G code in the 01 group.
- During a threadcutting operation, feedhold and mode change are impossible. Feedrate override is also ineffective, and the feedrate is locked at 100%.
- G33 can only be commanded in the G95 (mm/rev) mode. If it is commanded in the G94 (mm/min) mode, it causes an alarm.
- G33 can not be programmed in the tool radius compensation C mode. If it is commanded, it causes an alarm.
- Threadcutting command can not include 4 axes. If 4 axes are included in a threadcutting command, an alarm is caused.

2.9.19 AUTOMATIC CENTERING FUNCTION (G36, G37)†

With this function, the spindle is aligned with the center line of the machined bore, with a touch sensor.

- G36 I···(or J···) F··· ;

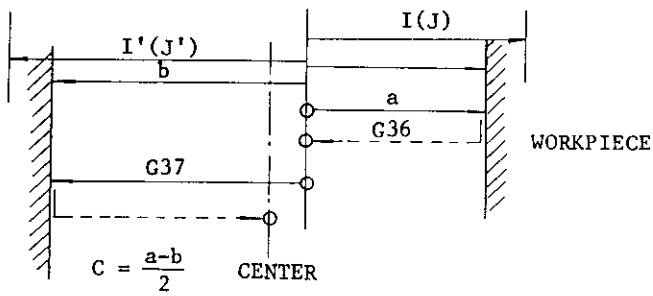
  With this command, the spindle moves in the X (or Y) direction at speed F until the touch sensor makes contact with the workpiece and gives a contact signal. The distance "a" from the start point to the contact point is stored with the function of G37. Then, the spindle returns to the start point in rapid traverse.

- G37 I'···(orJ'···) F··· ;

  With this command, the spindle moves in the X (or Y) direction at speed F until the touch sensor makes contact with the workpiece and gives a contact signal. Then, the spindle returns to the start point in rapid traverse.

$$D = \frac{a - b}{2}$$

where b = distance between G37 start point to contact point.



(I, J, I', J' = Incremental value)

Fig. 2.33

NOTES:

- When no contact signal is obtained during the movement through the specified incremental distances I (or J) and I' (or J'), this constitutes an error.
- When I and J are programmed together in a block of G36 or G37, this constitutes and error.
- When G36 or G37 is commanded in the tool radius compensation C mode or in a canned cycle, this command is treated as an error "034."