2.9.24 SCALING FUNCTION (G50, G51)+

With this function, workpiece contours programmed by part programs can be enlarged or reduced at any desired scale.

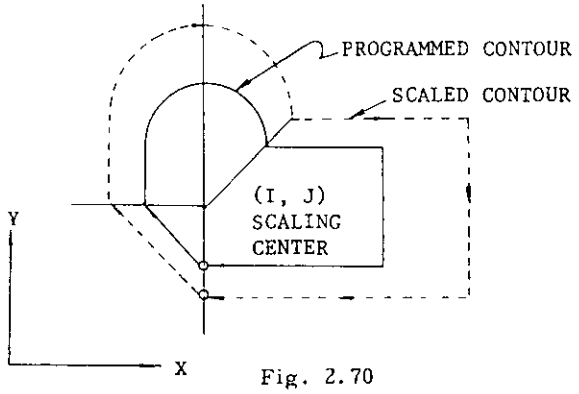


Fig. 2.70

The following G codes are used for this function.

G code	Group	Meaning
G50	15	Scaling OFF
G51	15	Scaling ON

Note: When power is applied or the control is reset, the control is in the state of G code marked with ◥.

- G51 I··· J··· K··· P··· ;

  With this command, the program is executed on an enalrged or reduced scale with the scale ratio specified by P, and the center of scaling specified by I, J, and K.

- G50; command cancels the scaling mode.
- The enlarging and reducing scales can be selected within the following range.

Enlarging and reducing range	0.000001-99.999999

When no scale ratio is specified in the program, the ratio set by parameter #6500 becomes effective as the scale ratio.

Command unit for P is: 1 = 0.000001.
When P command includes decimal number, numbers after decimal point are regarded as six-digit numbers.

Example

P0.999999	0.999999 time
P2.0	2 times
P2	0.000002 time

When P (designating multiplication) is omitted, multiplication is determined by setting #6500 and #6501.

$$\text{Multiplication} = \frac{\#6500}{\#6501}$$

Example

Where setting #6500 = 3, #6501 = 100

$$\text{Multiplication} = \frac{3}{100} = 0.03 \text{ times}$$

Multiplication should not exceed the enlarging and reducing range.

When I, J, or K is programmed in the G51 command, scaling functions on the axis designated: I··· X-axis, J··· Y-axis, K··· Z-axis. Scaling will work only on the axis selected by I, J, or K.

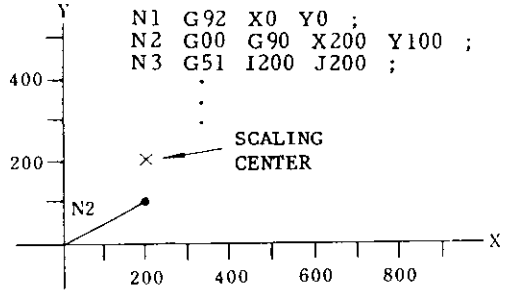
Example

G51 I100 J0 P08

With this command, scaling will work on X- and Y-axis and not on Z-axis.

Where the work coordinate system is specified, I, J, and K in the G51 block designates the distance between coordinate system zero point and scaling center.

Example



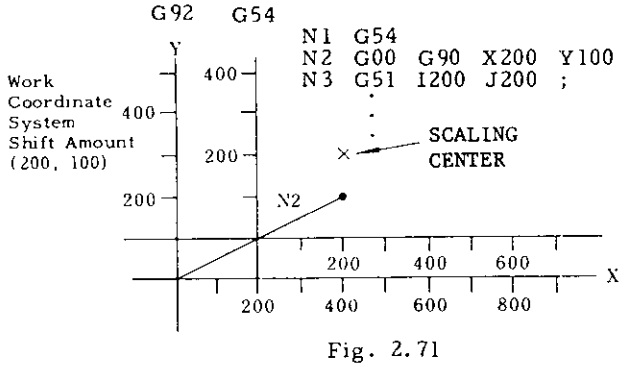


Fig. 2.71

NOTES:

- Scaling is turned on when approaching for usual machining and off after retraction on completion of approaching. Turning off and on scaling during machining will not form the correct contour.
- Scaling is executed on the two axes on machining plane. If scaling is executed on a single axis, an alarm occurs at circular command because scaling cannot work according to circular command.