HNC-08M CNC System
Programming Manual

（V1.0）

April, 2009

Wuhan・China

Wuhan Huazhong Numerical Control Co., Ltd

Table of Contents

1 General

1.1 Program Introduction of Machining

1.1.1 The Format of Instruction Programming

An NC program consists of a sequence of NC blocks, each instruction constituted by a number of commands. Each command includes two parts: command word and command parameter. Function is identified by command keyword, such as the effect of command word ‘G’ in the G01 is preparatory function, the effect of command word ‘M’ in the M03 is miscellaneous function.

As it is shown in Figure 1-1.

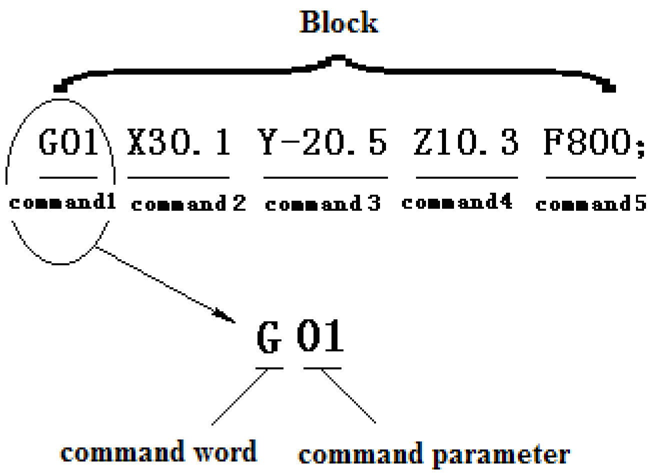

Figure 1-1 Structure of an NC Program

Note:

1) Many blocks can be written on the same line, separated by the ‘;’.

2) The block must be written on the same line, can not be written on different line.

1.1.2 Block skip and comments

1) block skip

The blocks can be executed optionally if there is a skip symbol '\' before them. If block skip is effective, the blocks which have a skip symbol '\' before them are not executed, else executed.
Warning, the skip symbol is '\', instead of '/', the latter for the division operator.
The blocks which need skip should be on one line, when block skip is effective, the blocks on the same line which have a skip symbol before them are skipped and not executed.

Example:

G0 X0 Y0 Z0;
\G00 X20 Y30; G01 Z50 F300;
M30;
When block skip is effective, all the code on the second line is not executed.

2) comment

All information after the "”" is regarded as comments which are ignored when the program is translated.
Comments can be done on whole line, or partly on the middle. It should be done on the tail when comment partly.

Example:

M03 S1000;
"Rapid positioning (comment whole line)
G90 G00 X100 Y200;
G17 G02 X150 I25 F300;　　" circular interpolation（comment partly on the tail）

1.1.3 Structure of an NC Program and execution

An NC program consists of main program and subprogram. It must have a main program, but unnecessary to subprogram. Main program should be placed at the beginning of the program file and end by M30 or M99. Subprogram can be ended by M99 only, and its name should be digital only, and can not have the extension.

The storage location of subprogram can be divided into two situations:

1) Subprogram and main program can exist in the same file, this time, subprogram must appear after the main program. The sequence between subprograms has non-requirements;
2) Subprogram can also be stored as a separate file, and the file name is the subprogram name.

Program executes as shown in Figure 1-3

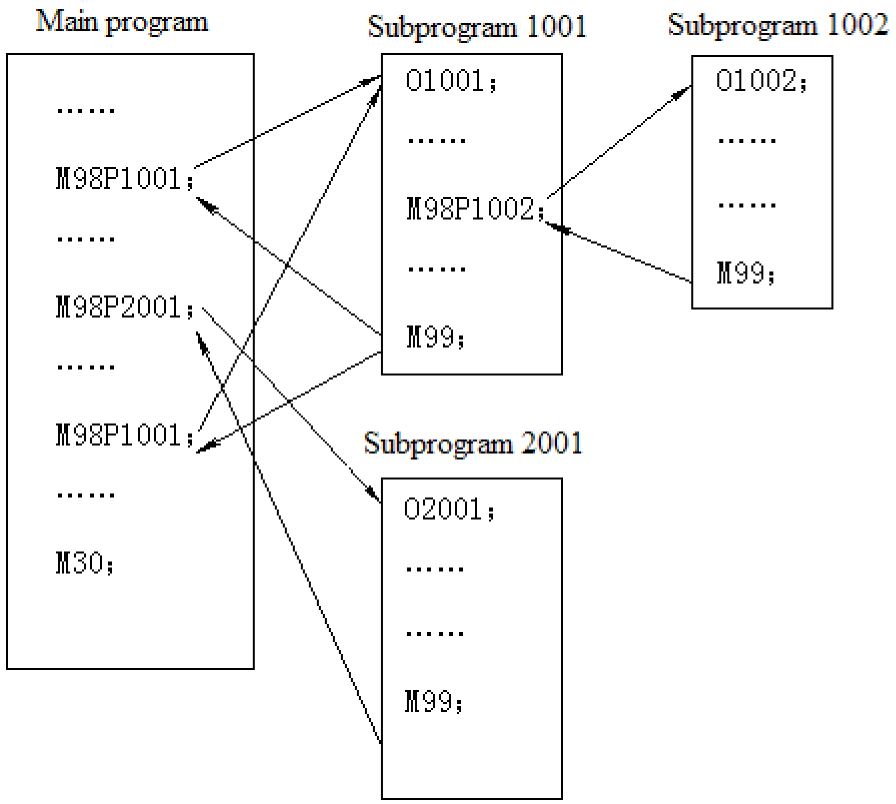

Figure 1-3 program execution

1.2 G Code

1.2.1 Modal G Code

G codes are divided into the following two types according to their validity:

1) One-shot G code: The G code is effective only in the block in which it is specified.
2) Modal G codes: This G codes are stored after the execution by the CNC system. The G code is effective until another G code of the same group is specified.

Example:

G53 X0 Y0 Z0； “G53 is One-shot G code, and needs to be specified each time
G01 X-200 F300；
Y-100； “G01 is Modal G code, and it is stored after the execution, and needs to be specified no longer.
Z-250；
M30；

1.2.2 G-code group

G codes are divided into several groups according to their function, G codes in 00 group are one-shot G code, while the other groups are modal G code. Multiple G codes can be specified in the same block if each G code belongs to a different group. If multiple G codes that belong to the same group are specified in the same block, only the last G code specified is effective.
The following table is the group of G code in HNC-08M system.
The system supports the G codes that already listed on the following table, else does not support them yet, please do not use.

Number	G code	Group	Function
01	G00	01	Rapid positioning
02	G01		Linear interpolation
03	G02		Circular interpolation/Helical interpolation CW
04	G03		Circular interpolation/Helical interpolation CCW
05	G02.4		3D Circular interpolation
06	G03.4		3D Circular interpolation
07	G04	00	Dwell，Exact stop
08	G28		Return to reference point
09	G29		Return from reference point
10	G30		Return to the second、third、fourth reference point
11	G52		Setting for local coordinate system
12	G53		Machine coordinate system selection
13	G53.1		Cutter arbor direction Control

14	G60		Single direction positioning
15	G65		Calling Macro program
16	G92		Setting for work coordinate system
17	G15	17	polar coordinate cancel
18	G16		polar coordinate
19	G17	02	XY plane selection
20	G18		ZX plane selection
21	G19		YZ plane selection
22	G24	12	Programmable mirror image (AXIS)
23	G25		Programmable mirror image (POINT)
24	G26		Programmable mirror image cancel
25	G40	07	Cutter compensation cancel
26	G41		Cutter compensation left
27	G42		Cutter compensation right
28	G43	08	Tool length compensation +direction
29	G44		Tool length compensation - direction
30	G43.4		3D Tool length compensation +direction
31	G44.4		3D Tool length compensation - direction
32	G49		Tool length compensation cancel
33	G50	11	Scaling cancel
34	G51		Scaling
35	G54	14	Workpiece coordinate system 1 selection
36	G55		Workpiece coordinate system 2 selection
37	G56		Workpiece coordinate system 3 selection
38	G57		Workpiece coordinate system 4 selection
39	G58		Workpiece coordinate system 5 selection
40	G59		Workpiece coordinate system 6 selection
41	G61	15	Exact stop mode
42	G05.1		High-speed and high-quality mode
43	G68	16	Coordinate rotation
44	G68.1		Special coordinate system selection
45	G69		Coordinate rotation cancel

46	G73	09	Peck drilling cycle
47	G74		Counter tapping cycle
48	G76		Fine boring cycle
49	G80		Canned cycle cancel
50	G81		Drilling cycle
51	G82		Drilling cycle (Dwell)
52	G83		Peck drilling cycle
53	G84		Tapping cycle
54	G85		Boring cycle
55	G86		Boring cycle
56	G87		Back boring cycle
57	G89		Boring cycle
58	G90	03	Absolute command
59	G91		Increment command
60	G94	05	Feed per minute
61	G95		Feed per rotation (not support yet)
62	G98	10	Return to initial point in canned cycle
63	G99		Return to R point in canned cycle

When the power is turned on , Its default setting as following:

01 Group: specified by the following parameter.

P1008 = 0: G00 when power on

P1008 = 1: G01 when power on

02 Group: specified by the following parameter.

P1011 = 0: G17 when power on

P1011 = 1: G18 when power on

P1011 = 2: G19 when power on

03 Group: specified by the parameter.

P1009 = 0: G90 when power on

P1009 = 1: G91 when power on

05 Group: G94 when power on

07 Group: G40 when power on

08 Group: G49 when power on

09 Group: G80 when power on

10 Group: G98 when power on

11 Group: G50 when power on
12 Group: G26 when power on
14 Group: G54 when power on
15 Group: G61 when power on
16 Group: G69 when power on
17 Group: G15 when power on

1.2.3 Interpolation Commands and State Commands

G codes can also be divided into interpolation commands and state commands according to their function.

1) Interpolation Command

The so-called interpolation command is the command to control the movement of axis, such as the G00, G01.
Coordinate commands which control the motor position are output after Interpolation command is executed.
It divides into the movement of the tool and the movement of the workpiece according to different machine, the movement of tool is assumed for Interpolation command in this manual.

2) State Command

The state command is that can affect the state of executing interpolation command .It is stored by the system only. It outputs no coordinates command and does not control the movement of axis, but will affect the execution and results of the subsequent interpolation command.
For example: G90 and G91 are state command. The following interpolation commands control the movement of axis by absolute value mode after G90 , similarly the following interpolation commands control the movement of axis by increment value mode. after G91

2 Interpolation Command

2.1 Basic Instructions

2.1.1 Rapid positioning (G00)

Description

The command G00 moves a tool to the position in the workpiece system specified with an absolute or an incremental command at a rapid traverse rate.
G00 is a straight line positioning methods, when executing G00, the system adjusts the speed of each axis automatically to make sure that a tool moves a straight line from the start point to the end point.

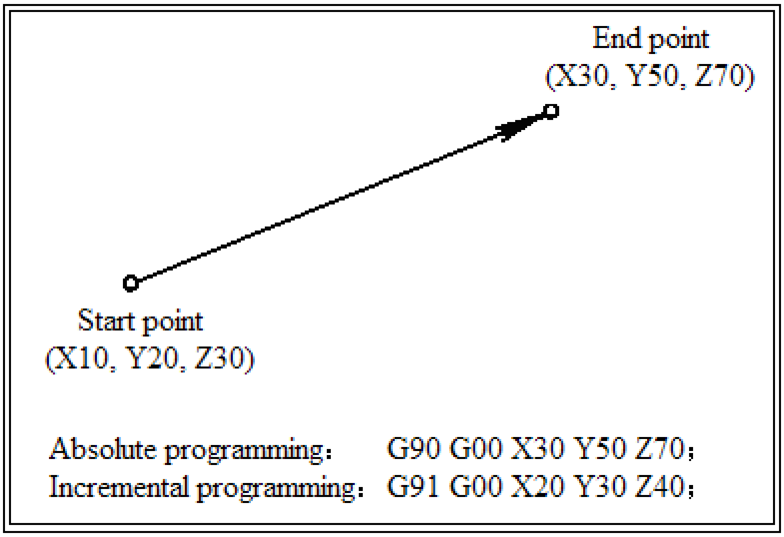

Figure 2-1 G00 Rapid positioning

The rapid traverse rate of G00 can not be specified in the address F, but can be overrided based on the parameter setting by the switch of rapid traverse magnification.

P2021: the acceleration of the linear axis G00.
P2022: the acceleration of the rotation axis G00.
P2025: the speed of linear axis G00.
P2026: the speed of rotation axis G00.

Format

G00 IP_;

IP_: In the absolute command, coordinate value of the end point is programmed. In the incremental command the distance the tool moves is programmed.

2.1.2 Linear Interpolation（G01）

Description

The command G01 moves a tool to the position in the workpiece system with an absolute or an incremental command at the speed which is specified in F.

The feedrate which is specified in F is effective until a new value is specified. It need not be specified for each block. The feedrate should be within the range which is determined by the system. The system will take the critical value automatically if it is beyond the range.

1) If the speed which is specified in F is more than its maximum speed, the system will take the maximum speed.

2) If the speed which is specified in F is less than its minimum speed, the system will take the minimum speed.

The maximum speed and minimum speed are limited by the following parameter settings:

P2035: the maximum speed in G01

P2041: the minimum speed in G01

In addition, its acceleration may also be limited the following parameter settings:

P2031: the acceleration of linear axis in G01.

P2032: the acceleration of rotation axis in G01.

Format

G01 IP_ F_;

IP_: For an absolute command, the coordinates of an end position, and for an incremental command, the distance the tool moves.

F_: Speed of tool feed

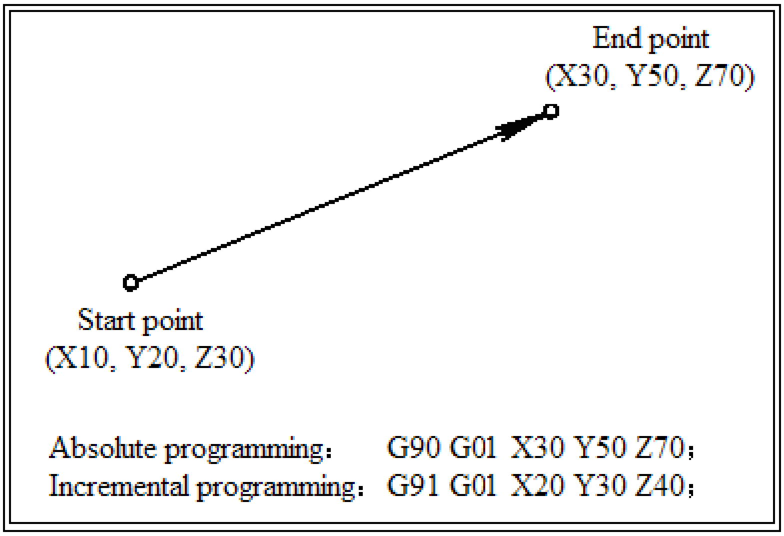

Figure 2-2 G01 linear interpolation

mm /min is used as the unit of feed rate for the linear axis, while degrees/ min for the rotation axis.

When using the linear axis (such as X、Y or Z) and rotation axis (such as A、B or C) for linear interpolation, the speed specified in F is feedrate along the arc in the Cartesian coordinates which is composed of the linear axis and the rotation axis .

To obtain the feed speed of rotation axis, the time for movement should be determined firstly. The feed speed of rotation axis (degrees / min) is equal to the ratio of the distance that rotation axis moves and the time.

Example:

G91 G01 X30 A50 F500;

The A axis regards as a linear axis to calculate the feed time.

Feed time $$t=\frac{\sqrt{30^2+50^2}}{500}=0.116\min$$

Rotation axis feed rate $$v=\frac{50}{t}=\frac{50}{0.116}=431.03degree/\min$$

Example:

1) Interpolation of Linear Axis

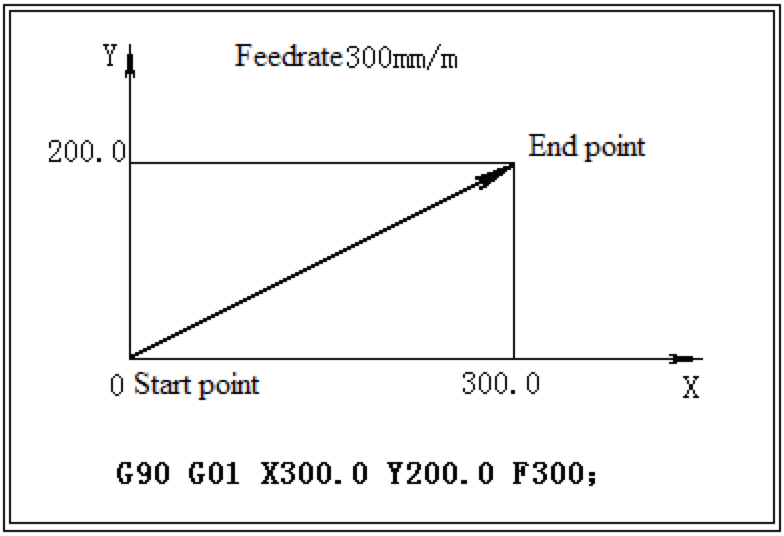

Figure 2-3 G01 Interpolation of linear axis

2) Interpolation of Rotation Axis

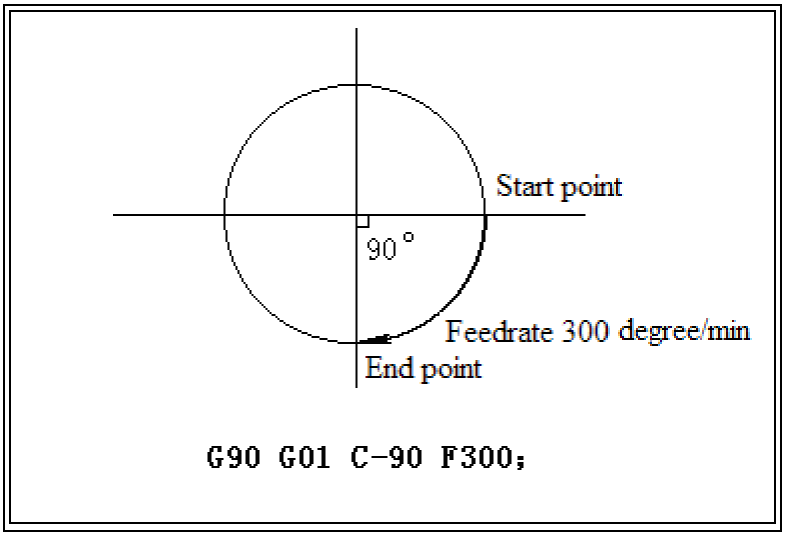

Figure 2-3 G01 Interpolation of rotation axis

2.1.3 Circular Interpolation (G02, G03)

Description

Circular interpolation commands move a tool along a circular arc clockwise (G02) or counterclockwise (G03). Circular interpolation includes several elements as following, the user can program with a few of them.

1) The direction of circular interpolation

Clockwise(G02) and counterclockwise(G03) on the XY plane are defined when the XY plane is viewed in the positive–to–negative direction of the Z axis in the Cartesian coordinate system. Similarly, the ZX plane is viewed in the positive–to–negative direction of the Y axis in the Cartesian coordinate system, the YZ plane is viewed in the positive–to–negative direction of the X axis in the Cartesian coordinate system
See the figure 2-5 below.

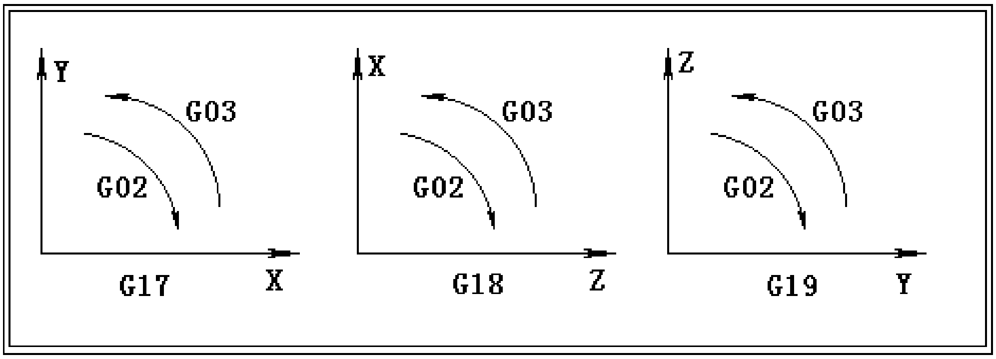

Fig 2-5 Direction of the circular interpolation

2) Expression of the arc end point

The circular arc end point is specified with the command (X, Y, Z) . For an absolute command, the workpiece coordinates of an arc end point, and for an incremental command, the distance the tool moves. See the figure2-6 below.

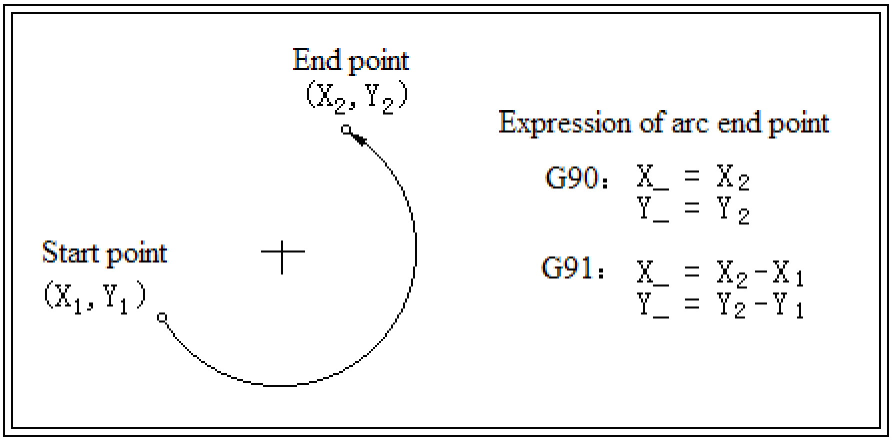

Fig 2-6 Expression of a circular arc end point

3) Distance from the start point to the center of the arc

The arc center is specified by addresses I, J and K. The numerical value following I, J or K, however, is a vector component in which the arc center is seen from the start point, and is always specified as an incremental value irrespective of G90 and G91, as shown below.

I, J and K must be signed according to the direction.

If X, Y, and Z are omitted , the end point is the same as the start point, and a full circle which the center is specified with I, J, and K is specified.

As shown in Figure 2-7.

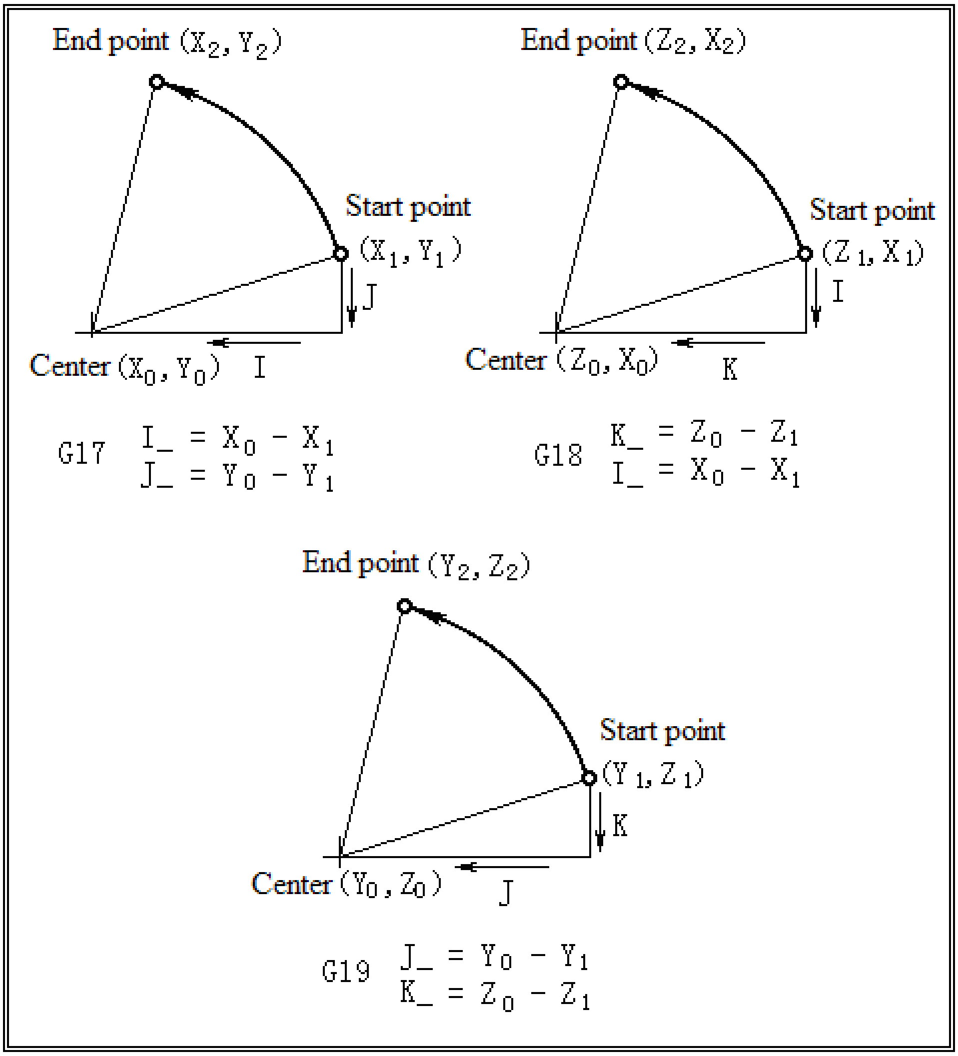

Fig 2-7 Expression of arc center

4) Radius

The arc center also can be specified with the radius in addition to the method above. There are two situations:

a) central angle is less than 180°;

b) central angle is more than 180°;

The arc should be clearly specified in programming. It is determined by the sign of the arc radius R. When the central angle is less than 180°, R is specified with a positive value. If the central angle is more than 180°, R must be specified with a negative value. As shown in Fig2-8.

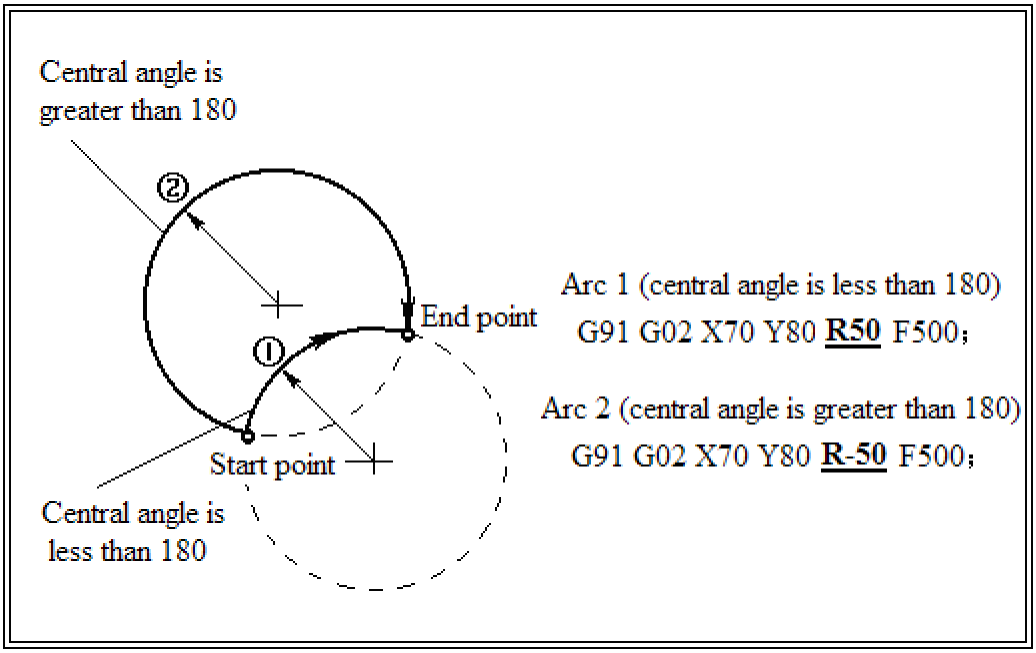

Fig 2-8 Sign of radius

5) The Feedrate

The feedrate in circular interpolation is equal to the feed rate specified by the F code, and the feedrate along the arc (the tangential feedrate of the arc) is controlled to be the specified feedrate.

Format

$$\text{XY}: \quad \text{G17} \begin{Bmatrix} \text{G02} \\ \text{G03} \end{Bmatrix} \text{X\_ Y\_} \begin{Bmatrix} \text{I\_ J\_} \\ \text{R\_} \end{Bmatrix} \text{F\_;}$$

$$\text{ZX}: \quad \text{G18} \begin{Bmatrix} \text{G02} \\ \text{G03} \end{Bmatrix} \text{X\_ Z\_} \begin{Bmatrix} \text{I\_ K\_} \\ \text{R\_} \end{Bmatrix} \text{F\_;}$$

$$\text{YZ}: \quad \text{G19} \begin{Bmatrix} \text{G02} \\ \text{G03} \end{Bmatrix} \text{Y\_ Z\_} \begin{Bmatrix} \text{J\_ K\_} \\ \text{R\_} \end{Bmatrix} \text{F\_;}$$

Fig 2-9 Format of the arc command

G17: The working plane is XY.
G18: The working plane is XZ.
G19: The working plane is YZ.

G02: Circular interpolation clockwise direction (CW).
G03: Circular interpolation counterclockwise direction (CCW).
X_: Command values of X axis.
Y_: Command values of Y axis.
Z_: Command values of Z axis.
I_: X axis distance from the start point to the center of an arc with sign.
J_: Y axis distance from the start point to the center of an arc with sign.
K_: Z axis distance from the start point to the center of an arc with sign.
R_: Arc radius.
F_: Feedrate along the arc.

> **Note:**
>
> 1) If I, J, K, and R are all specified in the program, R takes precedence and the other are ignored.
> 2) If the axis is not on the plane which is specified, an alarm is output.

Example

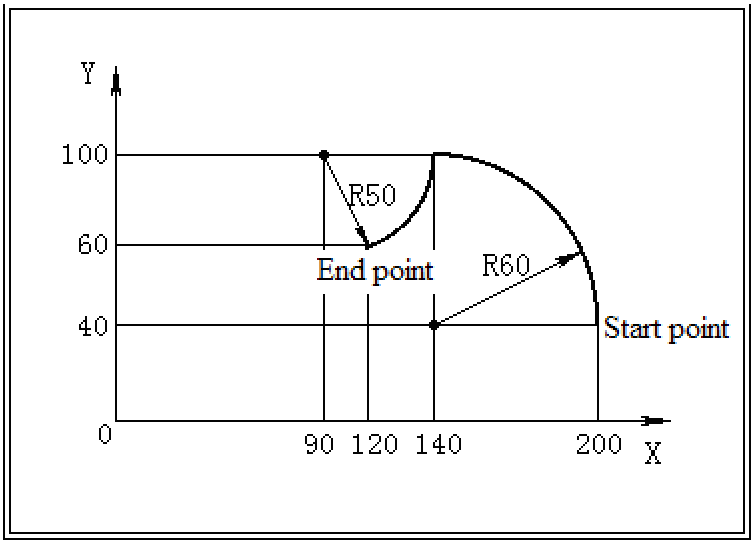

Fig 2-10 Circular Interpolation – Example

The tool path programming of the above Fig 2-10 is as follows:

1) Absolute programming

G92 X200.0 Y40.0 Z0;
G90 G03 X140.0 Y100.0 R60.0 F300.;
G02 X120.0 Y60.0 R50.0;
or
G92 X200.0 Y40.0Z0;
G90 G03 X140.0 Y100.0 I-60.0 F300.;
G02 X120.0 Y60.0 I-50.0;

2) Incremental programming

G91 G03 X-60.0 Y60.0 R60.0 F3000.;
G02 X-20.0 Y-40.0 R50.0;
or
G91 G03 X-60.0 Y60.0 I-60.0 F300.;
G02 X-20.0 Y-40.0 I-50.0;

2.2 Canned cycle programming

The canned cycle command can execute the specific operation using one G code, instead of several separated G commands in the program.
Canned cycles make it easier for the programmer to create programs. With a canned cycle, a frequently–used machining operation can be specified in a single block with a G function; without canned cycles, normally more than one block is required. In addition, the use of canned cycles can shorten the program to save memory.

2.2.1 High-speed Peck Drilling cycle (G73)

Description

The high–speed peck drilling cycle performs intermittent feeding along the Z–axis. When this cycle is used, chips can be removed from the hole easily. This allows, drilling to be performed efficiently.
Before specifying G73, rotate the spindle using a miscellaneous function (M code).When the G73 code and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. The system then proceeds to the next drilling operation.

When K is used to specify the number of repeats, the M code is executed for the first hole only; for the second and subsequent holes, the M code is not executed.
The action sequences as shown in Figure 2-11. Rapid positioning represents by dashed in the figure.

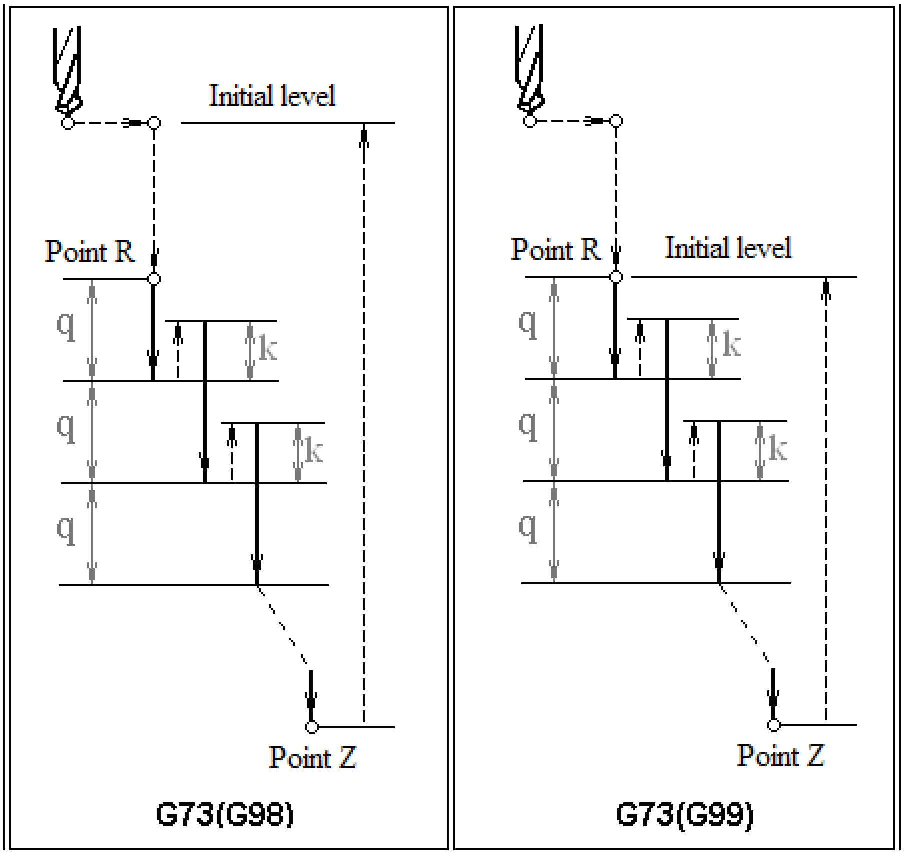

Fig 2-11 G73 high–speed peck drilling cycle

Format

(G98/G99) G73 X_ Y_ Z_ R_ Q_ P_ K_ F_ L_;

X_ Y_: Coordinate value of the hole position on XY plane in the absolute command, or the coordinate value of the hole position with reference to the initial point on XY plane in the incremental command

Z_: Coordinate value of the hole position on Z-axis in the absolute command, or the coordinate value of the hole position with reference to the point R on Z-axis in the incremental command

R_: Coordinate value of the point R in the absolute command, or the coordinate value of the point R with reference to the initial point in the incremental command

Q_: Depth of cutting for each cutting feed in the incremental command. The last depth of cutting near the hole may be less than this value, and Q should be negative.

P_: Dwell time at the bottom of a hole, in milliseconds

K_: Retraction amount at each time in the incremental command

F_: Cutting feedrate

L_: Number of repeats (when L = 1,it can be omitted).

> **Note:**
>
> 1) Drilling axis must be Z-axis;
> 2) Z-point must be less than R-point, else an alarm is output;
> 3) the parameters of the instruction G73 are stored as modal data, the same data can be omitted.

Example

M3 S2000; “Cause the spindle to start rotating.

G90 G99 G73 X300 Y-250 Z-150 R-100 Q15 K5 P2000 L5 F120;

“Position, drill hole 1, then return to point R.

Y-550; “Position, drill hole 2, then return to point R.

Y-750; “Position, drill hole 3, then return to point R.

X1000; “Position, drill hole 4, then return to point R.

Y-550; “Position, drill hole 5, then return to point R.

G98 Y-750; “Position, drill hole 6, then return to the initial level.

G80 G28 G91 X0 Y0 Z0 ; “drilling cancel, then return to the reference point

M5; “Cause the spindle to stop rotating

2.2.2 Left-hand Tapping Cycle (G74)

G74 is similar to G84, the difference is that G74 command can create a reverse thread. (Tapping is performed by turning the spindle counterclockwise. Then, the spindle turns clockwise for retraction when the tool reaches the bottom of the hole). In rigid tapping mode, using spindle motor to controlling the tapping process.

Spindle motor works as the servo motor. Performing tapping by the interpolation between the tapping axis and spindle. When tapping is performed in rigid mode, the spindle rotates one turn every time a thread lead which takes place along the tapping axis. This operation does not vary even during acceleration or deceleration.
The action sequence of G74 is shown in Figure 2-12. Rapid traverse to the R-point after positioning Along the X and Y axis, then the spindle turns counterclockwise, and performs tapping from the R-point to the Z-point. stop and pause the spindle when the tapping is completed, and then the spindle turns clockwise for retracting to the R point, then the spindle stops. Rapid traverse is performed to return to the initial level if it is in the G98 mode.

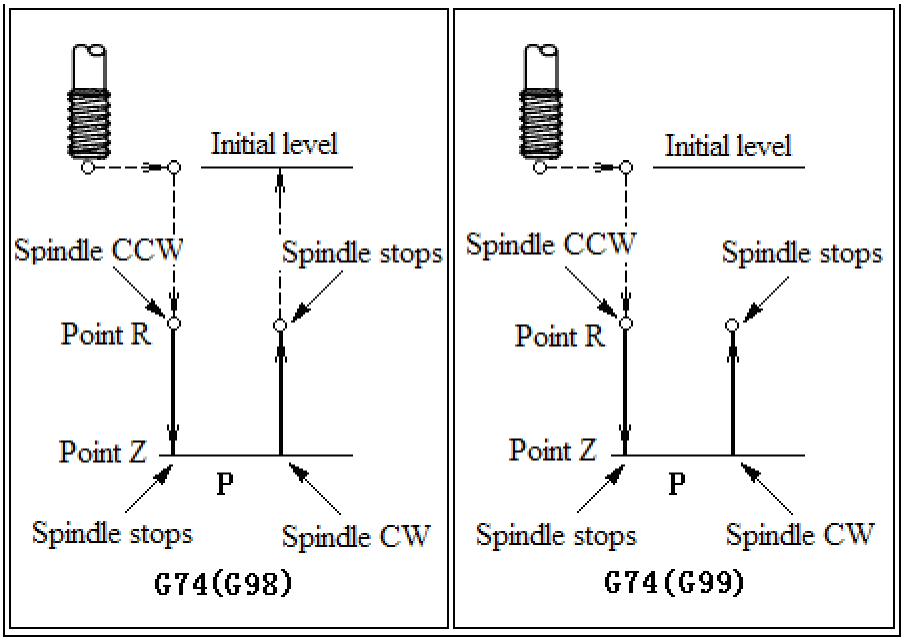

Fig 2-12 G74 left-hand tapping cycle

Format

G74 X_ Y_ Z_ R_ P_ F_ S_ L_;

X_ Y_: Coordinate value of the hole position on XY plane in the absolute command, or the coordinate value of the hole position with reference to the initial point in the incremental command

Z_: Coordinate value of the hole position on Z axis in the absolute command, or the coordinate value of the hole position with reference to the point R in the incremental command

R_: Coordinate value of the point R in the absolute command, or the coordinate value of the point R with reference to the initial point in the incremental command

P_: Dwell time at the bottom of a hole, in milliseconds.

F_: Thread lead

S_: The spindle speed of Tapping

L_: Number of repeats (when L = 1, it can be omitted).

The feedrate specified in the F is invalid during rigid tapping, the feedrate along the tapping axis is calculated by the following formula:

$$\text{Feedrate} = \text{spindle speed} \times \text{screw lead}$$

Note:

1) Tapping axis must be Z-axis;
2) Z-point must be less than R-point, else an alarm is output;
3) the parameters of the command G74 are stored as modal data, the same data can be omitted.

2.2.3 Fine Boring Cycle (G76)

Description

G76 command would bore a hole precisely. When the tool reaches the bottom of the hole, the spindle stops, and the tool is retracted to the direction opposite to the tool tip.

The shift amount which is opposite to the tool tip along X-axis at the bottom of a hole is specified by I and J. I and J is modal value retained within canned cycles.

The shift direction is determined when loading a tool.

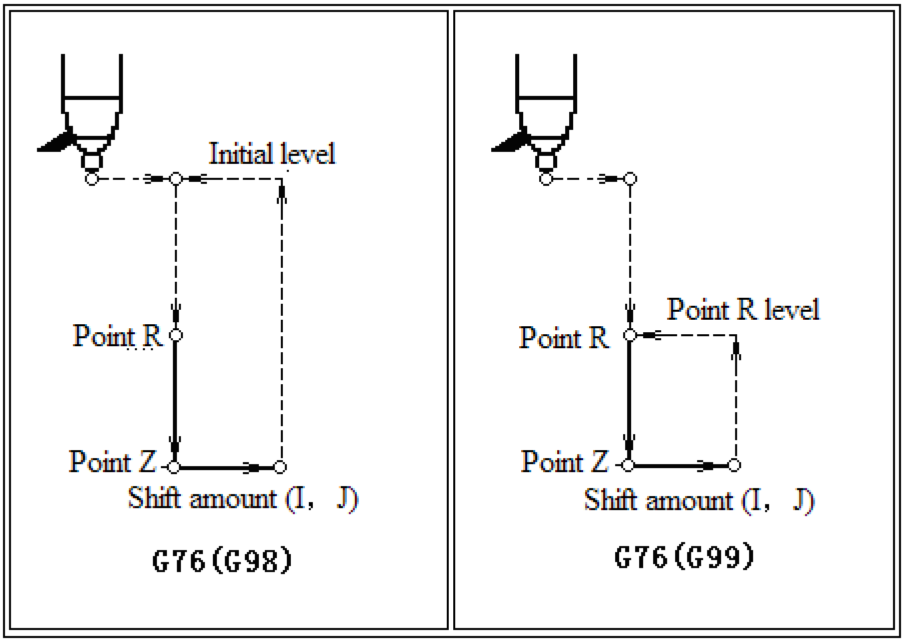

Fig 2-13 G76 fine boring cycle

Format

(G98/G99) G76 X_ Y_ Z_ R_ I_ J_ P_ F_ L_;

X_ Y_: Coordinate value of the hole position in the absolute command, or the coordinate value of the hole position with reference to the initial point in the incremental command.

Z_: Coordinate value of the hole position in the absolute command, or the coordinate value of the hole position with reference to the point R in the incremental command.

R_: Coordinate value of the point R in the absolute command, or the coordinate value of the point R with reference to the initial point in the incremental command.

I_: Shift amount along X-axis at the bottom of a hole.

J_: Shift amount along Y-axis at the bottom of a hole.

P_: Dwell time at the bottom of a hole.

F_: Cutting feedrate.

L_: Number of repeats (when L = 1, it can be omitted).

Note:

1) Drilling axis must be Z-axis;
2) Z-point must be less than R-point, else an alarm is output;
3) The parameters of the command G76 are stored as modal data, the same data can be omitted.

2.2.4 Drilling Cycle, Spot Drilling (G81)

Description

This cycle is used for normal drilling. Cutting feed is performed to the bottom of the hole. The tool is then retracted from the bottom of the hole in rapid traverse.
After positioning along the X-and Y-axes, rapid traverse is performed to point R. Drilling is performed from point R to point Z. The tool is then retracted in rapid traverse. Before specifying G81, use a miscellaneous function (M code) to rotate the spindle. When the G81 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. The system then proceeds to the next drilling operation. When K is used to specify the number of repeats, the M code is performed for the first hole only; for the second and subsequent holes, the M code is not executed.
The action sequence of G81 is shown in Figure 2-14. Rapid positioning represents by dashed in the figure.

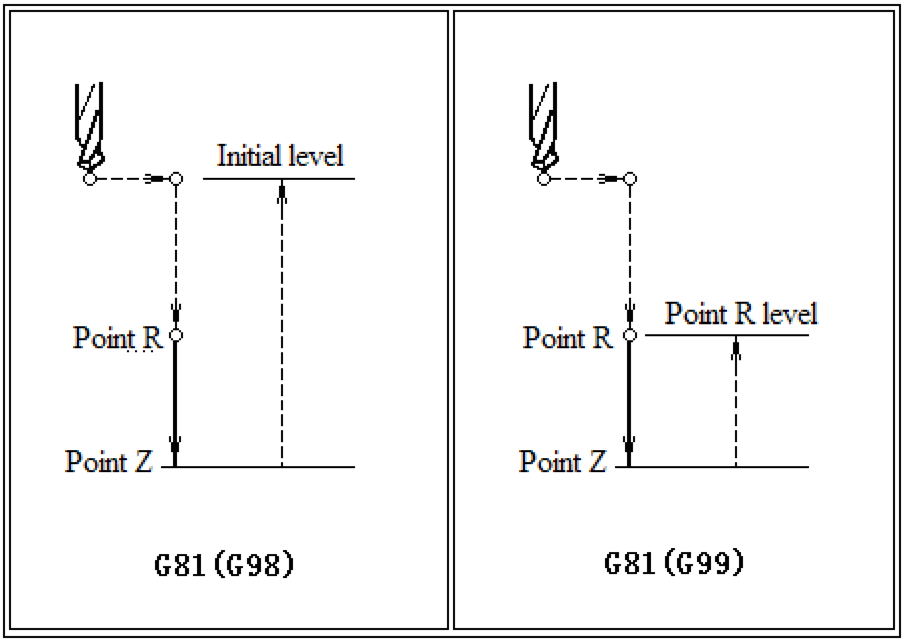

Fig 2-14 G81 drilling cycle, spot drilling

Format

(G98/G99) G81 X_ Y_ Z_ R_ F_ L_;

X_ Y_: Coordinate value of the hole position on XY plane in the absolute command, or the coordinate value of the hole position with reference to the initial point on XY plane in the incremental command.

Z_: Coordinate value of the hole position on Z axis in the absolute command, or the coordinate value of the hole position with reference to the point R on Z axis in the incremental command.

R_: Coordinate value of the point R in the absolute command, or the coordinate value of the point R with reference to the initial point in the incremental command.

F_: Cutting feedrate.

L_: Number of repeats (when L = 1, it may be omitted).

Note:

1) Drilling axis must be Z-axis;
2) Z-point must be less than R-point, else an alarm is output;
3) the parameters of the command G81 are stored as modal data, the same data can be omitted.

Example

M3 S2000; “Cause the spindle to start rotating.
G90 G99 G81 X300 Y-250 Z-150 R-100 F120;
“Position, drill hole 1, then return to point R.
Y-550; “Position, drill hole 2, then return to point R.
Y-750; “Position, drill hole 3, then return to point R.
X1000; “Position, drill hole 4, then return to point R.
Y-550; “Position, drill hole 5, then return to point R.
G98 Y-750; “Position, drill hole 6, then return to the initial level.
G80 G28 G91 X0 Y0 Z0 ; “drilling cancel, then return to the reference point
M5; “Cause the spindle to stop rotating

2.2.5 Drilling Cycle Counter Boring Cycle (G82)

Description

This cycle is used for normal drilling. After positioning along the X and Y axes, rapid traverse is performed to point R. Drilling is then performed from point R to point Z. When the bottom of the hole has been reached, a dwell is performed. The tool is then retracted in rapid traverse.
Before specifying G82, use a miscellaneous function (M code) to rotate the spindle.
When the G82 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. The system then proceeds to the next drilling operation.
When K is used to specify the number of repeats, the M code is executed for the first hole only; for the second and subsequent holes, the M code is not executed.
When a tool length offset (G43, G44, or G49) is specified in the canned cycle, the

offset is applied at the time of positioning to point R.
The action sequence of G82 is shown in Figure 2-15.

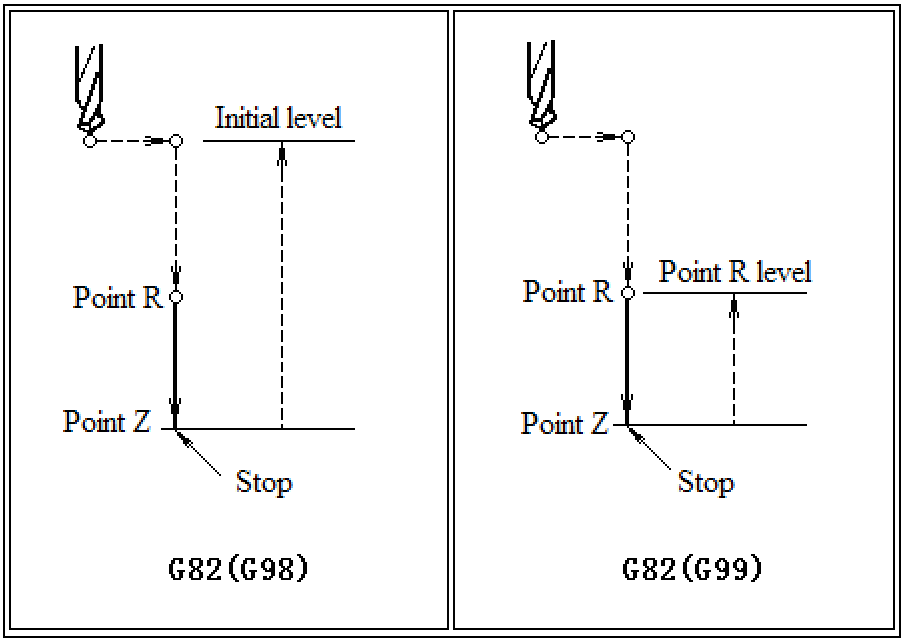

Fig 2-15 G82 drilling cycle counter boring cycle

Format

(G98/G99) G82 X_ Y_ Z_ R_ P_ F_ L_;

X_ Y_: Coordinate value of the hole position on XY plane in the absolute command, or the coordinate value of the hole position with reference to the initial point on XY plane in the incremental command

Z_: Coordinate value of the hole position on Z axis in the absolute command, or the coordinate value of the hole position with reference to the point R on Z axis in the incremental command

R_: Coordinate value of the point R in the absolute command, or the coordinate value of the point R with reference to the initial point in the incremental command

P_: Dwell time at the bottom of the hole

F_: Cutting feedrate

L_: Number of repeats (when L = 1, it can be omitted)

Note:

1) Drilling axis must be Z-axis;
2) Z-point must be less than R-point, else an alarm is output;
3) the parameters of the command G82 are stored as modal data, the same data can be omitted.

Example

M3 S2000;	“Cause the spindle to start rotating.
G90 G99 G82 X300 Y-250 Z-150 R-100 P1000 F120;	
	“Position, drill hole 1, then return to point R.
Y-550;	“Position, drill hole 2, then return to point R.
Y-750;	“Position, drill hole 3, then return to point R.
X1000;	“Position, drill hole 4, then return to point R.
Y-550;	“Position, drill hole 5, then return to point R.
G98 Y-750;	“Position, drill hole 6, then return to the initial level.
G80 G28 G91 X0 Y0 Z0 ;	“drilling cancel, then return to the reference point
M5;	“Cause the spindle to stop rotating

2.2.6 Peck Drilling Cycle (G83)

Description

This cycle performs peck drilling. It performs intermittent cutting feed to the bottom of a hole while removing shavings from the hole.
Q represents the depth of cut for each cutting feed. It must always be specified as an incremental value. The tool is retracted to the point R level to remove shavings after performing the depth Q of each cutting feed. In the second and subsequent cutting feeds, rapid traverse is performed up to a k point just before where the last drilling ended, and cutting feed is performed again, Q is the depth of cutting.
K represents the retraction amount at each time in the incremental command, the k point is the position that the retraction amount is k above the point the last drilling ended.
Before specifying G83, use a miscellaneous function (M code) to rotate the spindle.
When the G83 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. The system then

proceeds to the next drilling operation.
When K is used to specify the number of repeats, the M code is executed for the first hole only; for the second and subsequent holes, the M code is not executed.
When a tool length offset (G43, G44, or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.
The action sequence of G83 is shown in Figure 2-16.

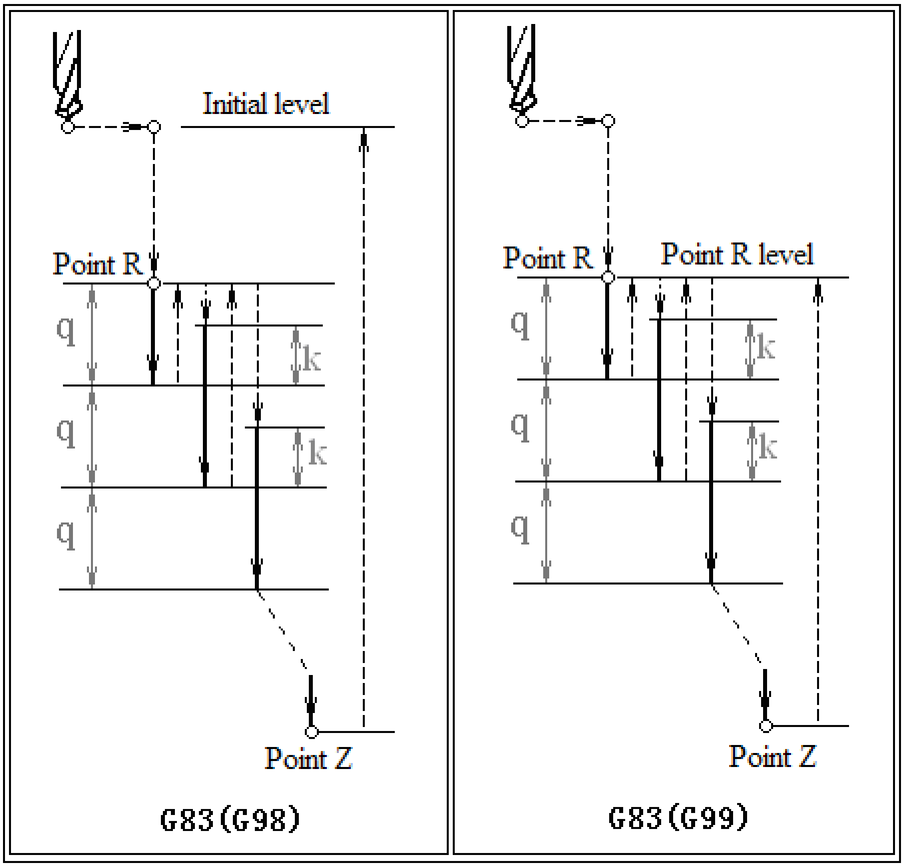

Fig 2-16 G83 peck drilling cycle

Format

(G98/G99) G83 X_ Y_ Z_ R_ Q_ K_ F_ L_;

X_ Y_: Coordinate value of the hole position on XY plane in the absolute command, or the coordinate value of the hole position with reference to the initial point on XY plane in the incremental command.

Z_: Coordinate value of the hole position on Z axis in the absolute command, or the coordinate value of the hole position with reference to the point R on Z axis in the incremental command.

R_: Coordinate value of the point R in the absolute command, or the coordinate value of the point R with reference to the initial point in the incremental command.

Q_: Depth of cutting for each cutting feed in the incremental command.

K_: Retraction amount at each time in the incremental command.

F_: Cutting feedrate.

L_: Number of repeats (when L = 1, it can be omitted).

Note:

1) Drilling axis must be Z-axis;
2) Z-point must be less than R-point, else an alarm is output;
3) the parameters of the command G83 are stored as modal data, the same data can be omitted.

Example

```
M3 S2000;                         "Cause the spindle to start rotating.
G90 G99 G83 X300 Y-250 Z-150 R-100 Q15 K5 L3 F120;
                                  "Position, drill hole 1, then return to point R.
Y-550;                            "Position, drill hole 2, then return to point R.
Y-750;                            "Position, drill hole 3, then return to point R.
X1000;                            "Position, drill hole 4, then return to point R.
Y-550;                            "Position, drill hole 5, then return to point R.
G98 Y-750;                        "Position, drill hole 6, then return to the initial level.
G80 G28 G91 X0 Y0 Z0 ;            "drilling cancel, then return to the reference point
M5;                               "Cause the spindle to stop rotating
```

2.2.7 Tapping Cycle (G84)

In rigid tapping mode, using spindle motor to controlling the tapping process. Spindle motor works as the servo motor. Performing tapping by the interpolation between the tapping axis and spindle. When tapping is performed in rigid mode, the spindle rotates one turn every time a thread lead which takes place along the tapping axis. This operation does not vary even during acceleration or

deceleration.
The action sequence of G84 is shown in Figure 2-17. Rapid traverse to the R-point after positioning Along the X and Y axis, then the spindle turns counterclockwise, and performs tapping from the R-point to the Z-point. stop and pause the spindle when the tapping is completed, and then the spindle turns clockwise for retracting to the R point, then the spindle stops. Rapid traverse is performed to return to the initial level if it is in the G98 mode.

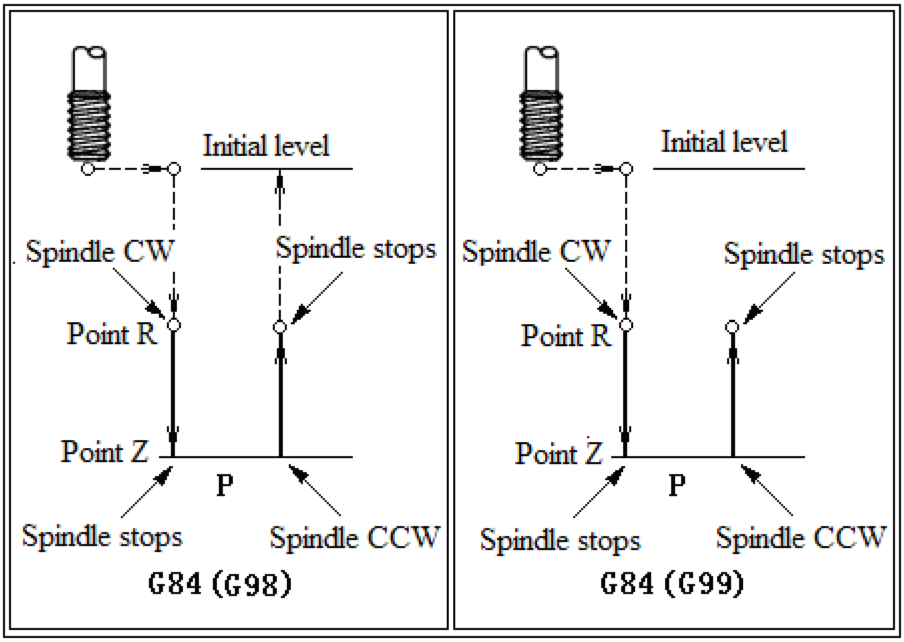

Fig 2-17 G84 tapping cycle

Format

G84 X_ Y_ Z_ R_ P_ F_ S_ L_;

X_ Y_: Coordinate value of the hole position on XY plane in the absolute command, or the coordinate value of the hole position with reference to the initial point in the incremental command

Z_: Coordinate value of the hole position on Z axis in the absolute command, or the coordinate value of the hole position with reference to the point R in the incremental command

R_: Coordinate value of the point R in the absolute command, or the coordinate value of the point R with reference to the initial point in the incremental command

P_: Dwell time at the bottom of a hole, in milliseconds.

F_: Thread lead
S_: The spindle speed of Tapping
L_: Number of repeats (when L = 1, it can be omitted).

The feedrate specified in the F is invalid during rigid tapping, the feedrate along the tapping axis is calculated by the following formula:

$$\text{Feedrate} = \text{spindle speed} \times \text{screw lead}$$

> **Note**
>
> 1) Tapping axis must be Z-axis;
> 2) Z-point must be less than R-point, else an alarm is output;
> 3) the parameters of the command G84 are stored as modal data, the same data can be omitted.

2.2.8 Boring Cycle (G85)

This cycle is used to bore a hole.
After positioning along the X and Y axes, rapid traverse is performed to point R. Drilling is performed from point R to point Z. When point Z has been reached, cutting feed is performed to return to point R., rapid traverse is performed to return to initial level if it is in G98 mode.
Before specifying G85, use a miscellaneous function (M code) to rotate the spindle.
When the G85 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. The system then proceeds to the next drilling operation.
When L is used to specify the number of repeats, the M code is executed for the first hole only; for the second and subsequent holes, the M code is not executed.
When a tool length offset (G43, G44, or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.

The action sequence of G85 is shown in Figure 2-17.

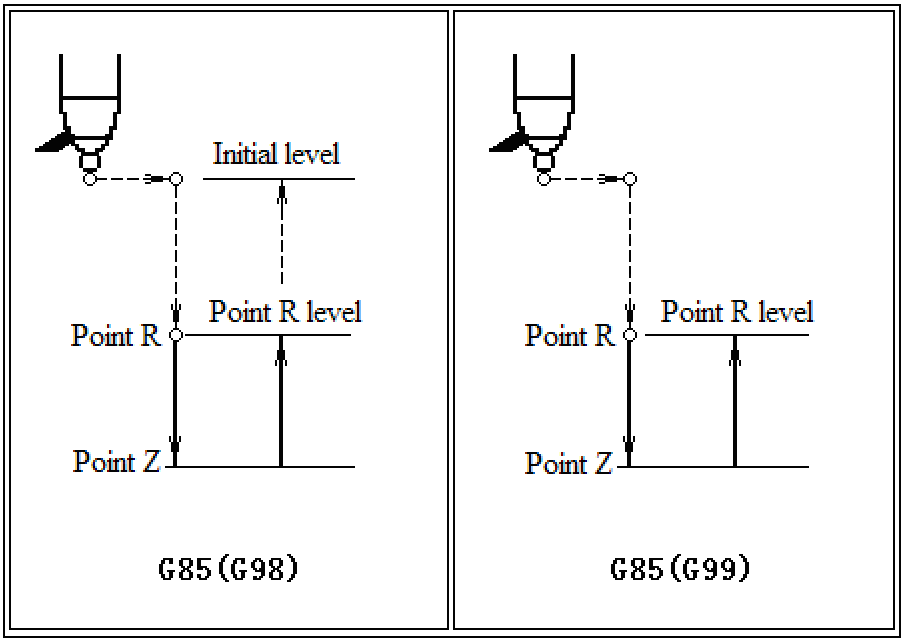

Fig 2-18 G85 boring cycle

Format

(G98/G99) G85 X_ Y_ Z_ R_ P_ F_ L_;

X_ Y_: Coordinate value of the hole position on XY plane in the absolute command, or the coordinate value of the hole position with reference to the initial point on XY plane in the incremental command

Z_: Coordinate value of the hole position on Z axis in the absolute command, or the coordinate value of the hole position with reference to the point R on Z axis in the incremental command

P_: Dwell time at the bottom of the hole

K_: Retraction amount at each time in the incremental command

F_: Cutting feedrate

L_: Number of repeats (when L = 1, it can be omitted)

Note

1) Drilling axis must be Z-axis;
2) Z-point must be less than R-point, else an alarm is output;
3) The parameters of the command G85 are stored as modal data, the same data can be omitted.

Example

M3 S2000;	“Cause the spindle to start rotating.
G90 G99 G85 X300 Y-250 Z-150 R-120 P3000 F120;	
	“Position, drill hole 1, then return to point R.
Y-550;	“Position, drill hole 2, then return to point R.
Y-750;	“Position, drill hole 3, then return to point R.
X1000;	“Position, drill hole 4, then return to point R.
Y-550;	“Position, drill hole 5, then return to point R.
G98 Y-750;	“Position, drill hole 6, then return to the initial level.
G80 G28 G91 X0 Y0 Z0 ;	“drilling cancel, then return to the reference point
M5;	“Cause the spindle to stop rotating

2.2.9 Boring Cycle (G86)

The action of G86 is the same as G81, when the bottom of the hole has been reached, the spindle stops, and then the tool is retracted in rapid traverse. It is used to bore a hole, which is not required the precise boring.

Format

(G98/G99) G86 X_ Y_ Z_ R_ F_ L_;

X_ Y_: Coordinate value of the hole position on XY plane in the absolute command, or the coordinate value of the hole position with reference to the initial point on XY plane in the incremental command

Z_: Coordinate value of the hole position on Z axis in the absolute command, or the coordinate value of the hole position with reference to the point R on Z axis in the incremental command

R_: Coordinate value of the point R in the absolute command, or the coordinate value of the point R with reference to the initial point in the incremental command

F_: Cutting feedrate

L_: Number of repeats (when L = 1, it can be omitted)

2.2.10 Back Boring Cycle (G87)

The command G87 generally is used to bore a hole which the top is smaller than the bottom. The Z-point on the bottom of the hole is generally above the reference point R. The command is different from others.

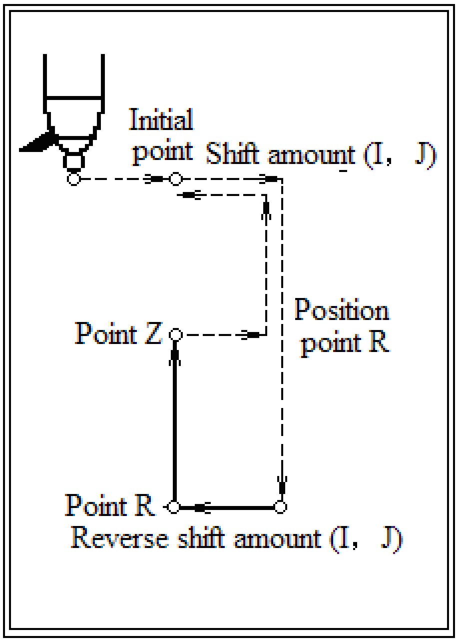

Fig 2-19 G87 back boring cycle

Format

(G98) G97 X_ Y_ Z_ R_ I_ J_ P_ F_ L_;

X_ Y_: Coordinate value of the hole position on XY plane in the absolute command, or the coordinate value of the hole position with reference to the initial point on XY plane in the incremental command

Z_: Coordinate value of the hole position on Z axis in the absolute command, or the coordinate value of the hole position with reference to the point R on Z axis in the incremental command

R_: Coordinate value of the point R in the absolute command, or the coordinate value of the point R with reference to the initial point in the incremental command

I_: Shift amount along X-axis at the bottom of a hole

J_: Shift amount along Y-axis at the bottom of a hole

P_: Dwell time at the bottom of the hole

F_: Cutting feedrate

L_: Number of repeats (when L = 1, it can be omitted)

Note:

1) Drilling axis must be Z-axis;
2) Z-point must be less than R-point, else an alarm is output;
3) the parameters of the command G87 are stored as modal data, the same data can be omitted.
4) G87 can be used with G98 only, invalid with G99.

2.2.11 Boring Cycle (G89)

This cycle is used to bore a hole.

This cycle is almost the same as G86. The difference is that this cycle performs a dwell at the bottom of the hole.

Before specifying G89, use a miscellaneous function (M code) to rotate the spindle. When the G89 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. The system then proceeds to the next drilling operation. When L is used to specify the number of repeats, the M code is executed for the first hole only; for the second and subsequent holes, the M code is not executed.

When a tool length offset (G43, G44, or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.

The action sequence of G89 is shown in Figure 2-20.

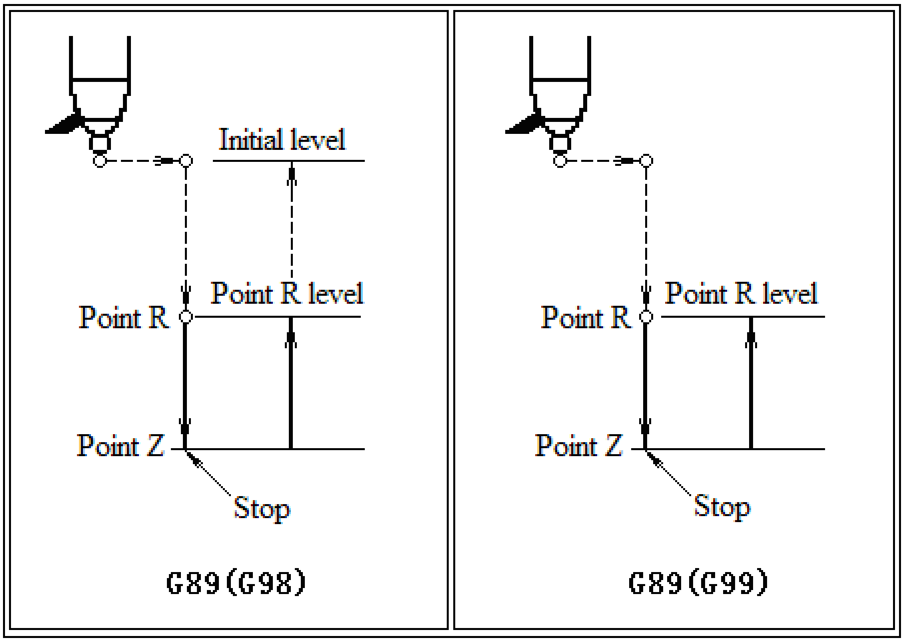

Fig 2-20 G89 boring cycle

Format

(G98/G99) G89 X_ Y_ Z_ R_ P_ F_ L;

X_ Y_: Coordinate value of the hole position on XY plane in the absolute command, or the coordinate value of the hole position with reference to the initial point on XY plane in the incremental command.

Z_: Coordinate value of the hole position on Z axis in the absolute command, or the coordinate value of the hole position with reference to the point R on Z axis in the incremental command.

R_: Coordinate value of the point R in the absolute command, or the coordinate value of the point R with reference to the initial point in the incremental command.

P_: Dwell time at the bottom of the hole.

F_: Cutting feedrate.

L_: Number of repeats (when L = 1, it can be omitted).

Example

M3 S2000; “Cause the spindle to start rotating.

G90 G99 G89 X300 Y-250 Z-150 R-120 P1000 F120;

“Position, drill hole 1, then return to point R then stop at the bottom of the hole for 1s

Y-550; “Position, drill hole 2, then return to point R.
Y-750; “Position, drill hole 3, then return to point R.
X1000; “Position, drill hole 4, then return to point R.
Y-550; “Position, drill hole 5, then return to point R.
G98 Y-750; “Position, drill hole 6, then return to the initial level.
G80 G28 G91 X0 Y0 Z0 ; “drilling cancel, then return to the reference point
M5; “Cause the spindle to stop rotating

Note:

1) Drilling axis must be Z-axis;
2) Z-point must be less than R-point, else an alarm is output;
3) The parameters of the command G85 are stored as modal data, the same data can be omitted.

2.2.12 Canned Cycle Cancel (G80)

Description

All canned cycles are canceled to perform normal operation. Point R and point Z are cleared. This means that R=0 and Z=0 in incremental mode. Other drilling data is also canceled (cleared).

Format

G80

Example

M3 S2000; “Cause the spindle to start rotating.
G90 G99 G81 X300 Y-250 Z-150 R-100 F120;
“Position, drill hole 1, then return to point R.
Y-550; “Position, drill hole 2, then return to point R.
Y-750; “Position, drill hole 3, then return to point R.
X1000; “Position, drill hole 4, then return to point R.
Y-550; “Position, drill hole 5, then return to point R.
G98 Y-750; “Position, drill hole 6, then return to the initial level.

G80 G28 G91 X0 Y0 Z0 ;	“drilling cancel, then return to the reference point
M5;	“Cause the spindle to stop rotating

3 State Command

3.1 Absolute and Incremental Programming (G90/G91)

Description

There are two ways to command travels of the tool; the absolute command, and the incremental command. In the absolute command, coordinate value of the end position is programmed; in the incremental command, move distance of the position itself is programmed. G90 and G91 are used to command absolute or incremental command, respectively.

Format

G90/G91 （……）；

（……）： the command to move the tool, such as G00、G01.

Example

The tool path is shown in the Fig 3-1, absolute and incremental programming as follows:

1) G90 programming

G90 G00 X100 Y200;

2) G91 programming

G91 G00 X-50 Y100;

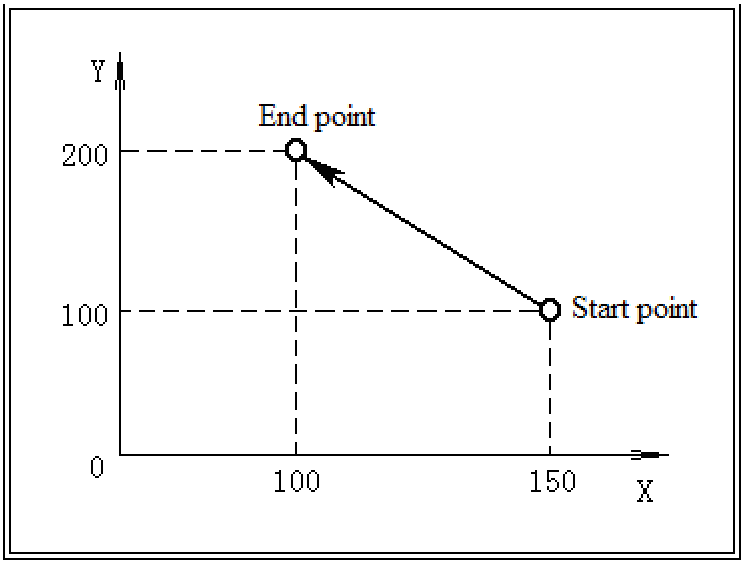

Fig 3-1　G90/G91 example

3.2 Dwell G04

Description

Stop is performed when the execution of G04 during the automatic running of a NC program, the dwell time is specified by the P or X following the command G04, then the following blocks are performed automatically.
It regards 4ms as a counting unit, 4ms is delayed if the time is less than 4ms.

Format

G04 P_/X_;

P_ / X_: Specified a time. The dwell time is specified in seconds by the P, and in millisecond by the X following the command G04.
Only one between the command P and X can be specified, and one must be specified, else system alarm.

3.3 Coordinate System

3.3.1 Setting a Workpiece Coordinate System (G92)

A workpiece coordinate system is set so that a point on the tool, such as the tool tip, is at specified coordinates.

As shown in Figure 32, if the tool tip location is (X = 50, Y = 40), the workpiece coordinate system $X_1O_1Y_1$ is established, if the tool tip location is (X = 70, Y = 50), the workpiece coordinate system $X_2O_2Y_2$ is established.

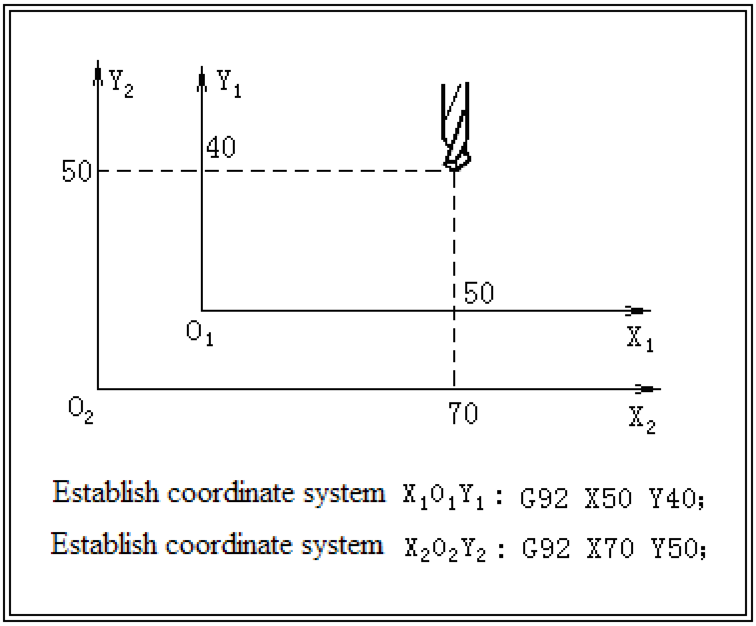

Fig 3-2 Setting a Workpiece Coordinate System

If a coordinate system is set using G92 during tool length offset, a coordinate system in which the position before offset matches the position specified in G92 is set. As shown in Figure 3.3

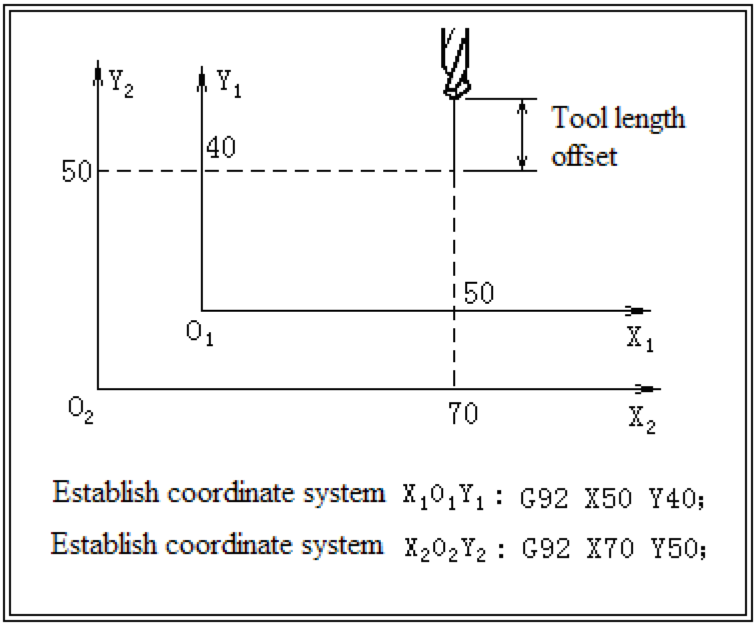

Fig 3-3 Setting a Workpiece Coordinate System by G92

Format

G92 IP_;

IP_: The coordinates of the tool tip at the set workpiece coordinate system. the zero position of the axes which are not specified in the workpiece coordinate remains the same in the command IP_, so the command G92 can set the zero position of all axes in the coordinate system respectively.

> **Note:**
>
> The values of the workpiece coordinate system which is set by G92 are losable, it is cleared when power off.

3.3.2 Selecting a Workpiece Coordinate System（G54~G59）

The user can also choose the coordinate system which has been set in CNC system as the workpiece coordinate system for machining. It can set 6 workpiece coordinate systems (refer to system operation manual) as follows:

G54: Workpiece coordinate system 1 selection

G55: Workpiece coordinate system 1 selection
G56: Workpiece coordinate system 1 selection
G57: Workpiece coordinate system 1 selection
G58: Workpiece coordinate system 1 selection
G59: Workpiece coordinate system 1 selection

The zero position of all axes in the workpiece coordinate system is reset by the workpiece coordinate system which has been chosen.

When the power is turned on, G54 coordinate system is selected.

3.3.3 Plane Selection (G17/G18/G19)

The user needs to select the planes for the functions of circular interpolation、coordinate system rotation、 mirror image and so on. There are three planes: XY plane、ZX plane、YZ plane. Select the planes by the following command:

G17: XY plane, X-axis for the first axis, Y-axis for the second axis;
G18: ZX plane, Z-axis for the first axis, X axis for the second axis;
G19: YZ plane, Y-axis for the first axis, Z axis for the second axis;

The default planes are setting by the following parameters when the power is turned on:

P1011 = 0: XY plane when power on (G17).
P1011 = 1: ZX plane when power on (G18).
P1011 = 2: YZ plane when power on (G19).

3.4 Tool compensation

3.4.1 Tool Length Compensation (G43/G44/G49)

Description

Generally, the tool length assumed during programming is not always equal to the actual tool length. There is a offset value between them. To simplify the operation and program, we can store it into the offset memory, then compensate the offset value by the code of tool length compensation. In this way, normally machining is performed without changing the program as long as we know the tool offset value, even using the tools which have different length.

Specify the direction of offset with G43 or G44.

G43: Positive offset
G44: Negative offset

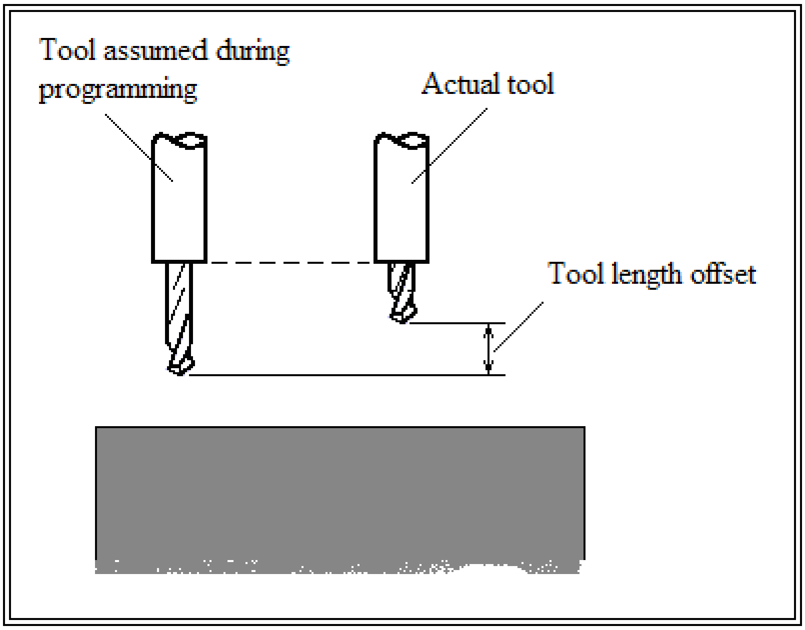

Fig 3-4 Tool length compensation

Format

G43/G44 H_;
…… ;
G49;

G43/G44: Tool length compensation effective;
H_: Address for specifying the tool length offset value;
……: The command to move a tool;
G49: Tool length offset cancel;

Note:

1）Compensation is performed along the Z-axis only by G43/G44 (positive or negative);
2）The command H in the tool length compensation must be specified with the G43 or G44, else system alarm.

3.4.2 Cutter Compensation (G40/G41/G42)

Description

Generally, during programming the user just programs for the center of the tool path, but during the actual machining, the tool path should be shifted by the radius of the tool which is not zero, so the function of cutter compensation is needed

Specify the direction of tool radius compensation with G41 or G42:

G41: Activate cutter compensation, tool operates in machining operation to the left of the contour (Fig 3-5a).

G42: Activate cutter compensation, tool operates in machining operation to the right of the contour (Fig 3-5b).

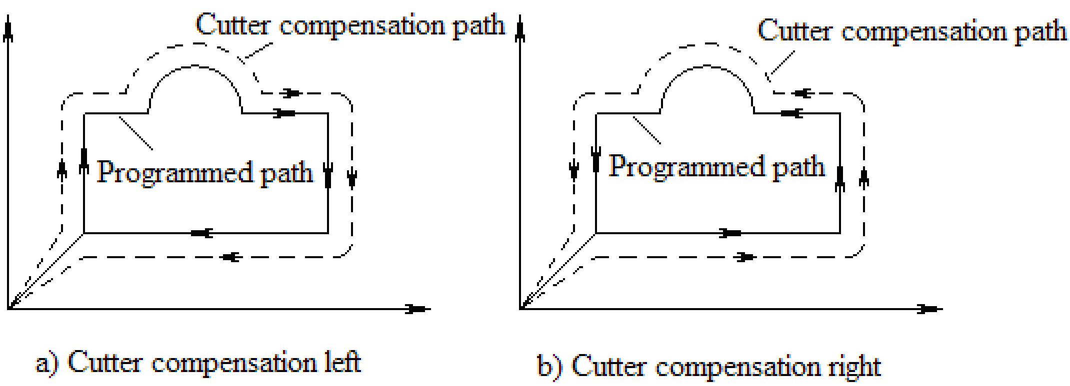

Fig 3-5 cutter compensation

Format

G41/G42 D_;
…… ;
G40;

G41/G42: Cutter compensation effective; G41-Cutter compensation left; G42-Cutter compensation right.

D_: Code for specifying as the cutter compensation value. Assign a cutter compensation values to the D codes. The command D must be specified with the G41 or G42, else system alarm.

……: The command to move a tool.

G40: Cutter compensation cancel.

3.5 Subprogram Control (M98)

Description

M98 is used to call a subprogram. The subprogram can be in the same file with the main program, also can be in a file individually. If it is in the same file with the main program, the letter O must be specified at the beginning of a program, else the file name of the subprogram must be specified by a four-digit number, and has no extension. It must be end with M99.
The subprogram in the same file with the main program has the priority to be called.

Format

M98 P_;

P_: In the sequence number following the P, the lower four digits are registered as a subprogram name, the others are the times to be called which are as many as you can.

Example: M98 P121001
the subprogram name is 1001, and number of times the subprogram be called repeatedly is 12.

Example

a) The subprogram in the same file with the main program

```
G00 X0 Y0 Z0
G01 X25 Y40 F1000
M98 P52001          "the subprogram 2001 is called, it is called 5 times.
G00 X0 Y0
M30

O2001
M3 S2000
G00 Z-10
G01 Z-50 F100
G00 Z0
M05
```

```
M99
```

b) The subprogram in a file individually

Main program：main.nc

```
G00 X0 Y0 Z0
G01 X25 Y40 F1000
M98 P52001          "the subprogram 2001 is called，it is called 5 times.
G00 X0 Y0
M30
```

Subprogram (no extension)：2001

```
M3 S2000
G00 Z-10
G01 Z-50 F100
G00 Z0
M05
M99
```

4 High-speed And High-precision Mode (G05.1)

Generally, it approaches to the complex curving surface by minute line segments during programming for moldings machining. In general machining mode, it is insufficient to deal with minute line segments , which leads to low work efficiency and crude surface. To realize high-speed machining, High-speed and High-precision Mode which improves the function to deal with minute line segments can enhance the machining speed.

High-speed High-precision Mode is specified by G05.1：

G05.1 Q1： High-speed and High-precision Mode Ⅰ on

G05.1 Q2： High-speed and High-precision Mode Ⅱ on

G05.1 Q0： High-speed and High-precision Mode off

It is in the G61 mode after High-speed High-precision Mode off

	High-speed and High-precision Mode Ⅰ	High-speed and High-precision Mode Ⅱ
Segments look ahead	50	50

Speed look ahead	YES	YES
High-speed interpolation for short line segments	YES	YES
Optimization for uniting minute line segments	YES	YES
Optimization for length of short line segments	NO	YES
Splines interpolation	NO	YES

4.1 High-speed and High-precision Mode Ⅰ(G05.1Q1)

Under High-speed and high-precision mode Ⅰ, the value of transition speed in the point where the nearby line segments is connected is calculated automatically. To realize high speed machining, it makes the value of transition speed to the maximum under the precondition of no excessive acceleration appears.

Under High-speed and high-precision mode Ⅰ, the interpolation path is the same as the programmed path.

4.2 High-speed and High-precision Mode Ⅱ(G05.1Q2)

High-speed and high-precision mode Ⅱ is the spline curve interpolation mode . Under this mode, the tool path specified by G01 in the program is connected by spline to interpolation. In the figure as follows if it satisfies the spline condition, the dashed represents the programmed path and the real line represents the spline path which is the actual movement of a tool. On the knee point of the programmed path (such as the point B、C、D), a tool passes by with a high speed to realize the high-speed machining.

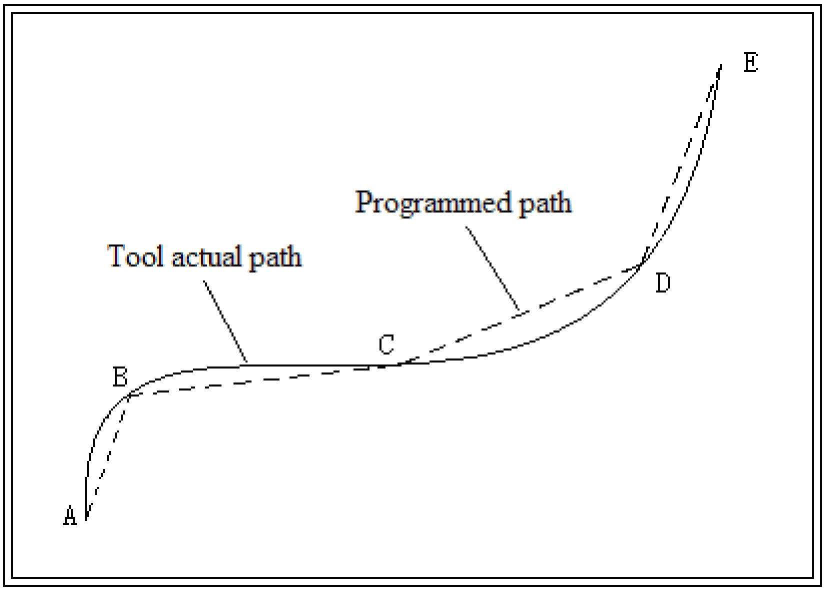

Fig 4-1 spline interpolation

The line path specified by G01 can be made to spline under definite condition. It includes the condition as follows:

1) Angle between the vectors of two nearby line segments

The angle is defined in the following figure. If it is less than the limited value, it satisfies the spline condition and the interpolation controller makes the nearby line segments to spline for interpolation, else it does not satisfy the spline condition. If the angle of the line from nearby line exceed the limited value, linear interpolation is performed for the line.

The limited value of the angle shaped by two nearby line segments is set by the parameter P2050 for spline interpolation.

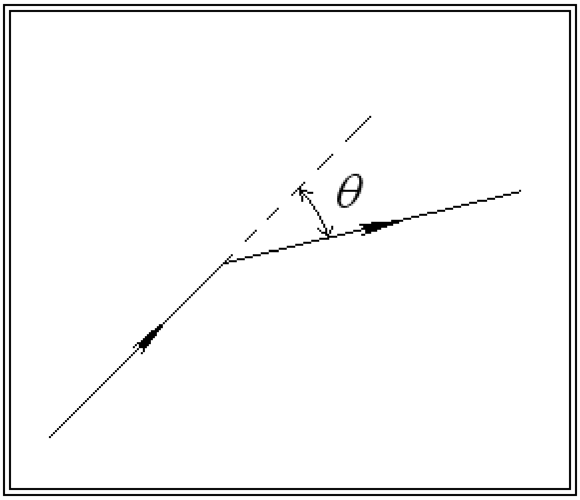

Fig 4-2 defining angle of the vectors

2) Ratio of the length of the two nearby line segments

If the ratio between L1 and L2 exceeds the limited value, it does not satisfy the spline condition. We assume that the limited value is ε (ε>1), the spline condition is defined as follows:

$$\frac{1}{\varepsilon} < \frac{|L1|}{|L2|} < \varepsilon$$

If the ratio exceeds the limited value, linear interpolation is performed for the line. The limited value of the ratio is set by the parameter P2051.

5 Appendix: Program Alarm and Description

Alarm NO.: **No.1000** Wrong Reason: This alarm is output when system can not open the program file. Solution: it usually happens because: 1) program file is corrupted, please rebuild the program file; 2) memory defective, please contact with the system manufacturer.
Alarm NO.: **No.1003** Wrong Reason: Wrong Format of five axes machining program Description: During five axes machining programming, the user can use the angle programming (ABC format) as well as the tool vector programming (IJK format), but it can not be mixed in the same program segment, otherwise an alarm is output.. Solution: Modify the program format
Alarm NO.: **No.1005** Wrong Reason: System does not support the instruction that has been used in the program. Solution: Refer to System Programming Manual to delete the instruction that the system does not support.
Alarm NO.: **No.1007** Wrong Reason: System does not support the command word that has been used in the program. Solution: Refer to the format of the correlative command in the wrong line to delete the redundant command word
Alarm NO.: **No.1008** Wrong Reason: There are redundant characters in the program segment. Solution: Refer to the format of the correlative command in programming manual to delete the redundant characters.

Alarm NO.：**No.1010** Wrong Reason：There is no command word corresponding to the command parameter which has already existed. Example: there is no command word before the “00” in “00 X20 Y30;”. Solution：Input the command word again.
Alarm NO.：**No.1011** Wrong Reason：There is no command parameter corresponding to the command word which has already existed. Example: there is no command parameter after the “X” in “G00 X;” Solution：Input the command parameter again.
Alarm NO.：**No.1012** Wrong Reason：Wrong position of the skip symbol. Description：The skip symbol ‘\’ can be placed at the beginning of a block only, and can not be placed in other positions. Solution：Modify the position of the skip symbol.
Alarm NO.：**No.1015** Wrong Reason：Wrong programming type has been set. Description：Special programming type is set for special machining (rigid tapping). The programming type is set to be not-RTCP for rigid tapping. Solution：Check and set the parameter of the programming type correctly.
Alarm NO.：**No.1020** Wrong Reason：Wrong tool number specified by the command T. Description：If the capacity of the tool library is N, the tool number should be in [1,N], else error. Solution：Input the right tool number.
Alarm NO.：**No.1050** Wrong Reason：Wrong parameter specified by the command N. Description：There are two types: 1) No parameter after the command N; 2) The parameter following the command N is less than 0. Solution：Input the right parameter N.

Alarm NO.: **No.1051**
Wrong Reason: Wrong parameter specified by the command M.
Description: There are two types: 1) No parameter after the command M; 2) The parameter following the command M is less than 0.
Solution: Input the right parameter M.

Alarm NO.: **No.1052**
Wrong Reason: Wrong parameter specified by the command S.
Description: There are two types: 1) No parameter after the command S; 2) The parameter following the command S is less than 0.
Solution: Input the right parameter S.

Alarm NO.: **No.1053**
Wrong Reason: Wrong parameter specified by the command T.
Description: There are two types: 1) No parameter after the command T; 2) The parameter following the command T is less than 0.
Solution: Input the right parameter T.

Alarm NO.: **No.1054**
Wrong Reason: Wrong parameter specified by the command F.
Description: There are two types: 1) No parameter after the command F; 2) The parameter following the command F is less than 0.
Solution: Input the right parameter F.

Alarm NO.: **No.1055**
Wrong Reason: Wrong parameter specified by the command O.
Description: There are two types: 1) No parameter after the command O; 2) The parameter following the command O is less than 0.
Solution: Input the right parameter O.

Alarm NO.: **No.1090**
Wrong Reason: The plane selection command is specified in the blocks which G68 is effective.
Description: We can not use G17、G18、G19 in the blocks which G68 is effective.
Solution: The plane selection command can be specified after G69 which

makes the G68 ineffective.
Alarm NO.: **No.1250** Wrong Reason: Wrong arc data in the command G02 or G03. Description: System needs to check the arc data during circular interpolation for the arc command. If the error of the arc data exceeds the limited value, an alarm of Wrong arc data is output. Solution: 1) If the actual arc error is in the reasonable range, modify the setting of the value limited by arc data in the system parameter (P1085) to make it greater than the actual arc error. 2) if the actual arc error is excessive, please calculate the start, end and radius of the arc again to make sure the arc data is right.
Alarm NO.: **No.1251** Wrong Reason: There are axis commands beyond the current plane in the arc command G02. Description: The moving axis command in the arc command must be in the current plane. Solution: 1) If the axis command in G02 is correct, please select the plane again. 2) If the current plane is correct, please modify the wrong axis command in G02.
Alarm NO.: **No.1252** Wrong Reason: The command G02 is half-baked. Solution: Refer to the format of the command G02 to complement it.
Alarm NO.: **No.1253** Wrong Reason: There are axis commands beyond the current plane in the arc command G03. Description: The moving axis command in the arc command must be in the current plane. Solution: 1) If the axis command in G03 is correct, please select the plane again. 2) If the current plane is correct, please modify the wrong axis command in G03.
Alarm NO.: **No.1254**

Wrong Reason：The command G03 is half-baked. Solution：Refer to the format of the command G03 to make it full.
Alarm NO.：**No.1260** Wrong Reason：The command G60 is half-baked. Description：Single direction positioning G60 can not be specified alone , there must be the moving axis command following it. Solution：Complement the moving axis command following it.
Alarm NO.：**No.1281** Wrong Reason：The command G92 is half-baked. Description：Single direction positioning G92 can not be specified alone , there must be the command of moving axis following it. Solution：Complement the moving axis command following it.
Alarm NO.：**No.1290** Wrong Reason：Redundant time command for G04 Description：The dwell time is specified by the P or X following G04, but can not be used together. Solution：Delete the redundant time command.
Alarm NO.：**No.1291** Wrong Reason：Lack of time command following G04 Description: The dwell time must be specified by the P or X following G04. Solution：Complement the dwell time.
Alarm NO.：**No.1295** Wrong Reason：Wrong format of G05 Description：Q must be specified following G05, and its value can be 0、1 or 2 only, else an alarm is output. Solution：Modify the program.
Alarm NO.：**No.1300** Wrong Reason：Redundant command for scaling magnification after G51. Description：Same magnifications which are specified by P are applied to each axes, also different magnifications which are specified by I、J and K

are applied to each axes, but it can not be specified together. Solution： Delete redundant command.
Alarm NO.： **No.1301** Wrong Reason： There are axis commands beyond the current plane following G68 Description：The moving axis command for the center of rotation following G68 must be in the current plane. Solution：1) If the axis command following G68 is correct, please select the current plane again. 2) If the current plane is correct, please modify the wrong axis command following G68.
Alarm NO.： **No.1305** Wrong Reason： Redundant command for axis of symmetry following G24 Description： Axis of symmetry must be specified by single axis only following G24. Solution： Delete redundant command for axis of symmetry
Alarm NO.： **No.1306** Wrong Reason： Axis of symmetry is not specified following G24. Description： There must be one axis command for axis of symmetry at least. Solution： Complement the command for axis of symmetry.
Alarm NO.： **No.1307** Wrong Reason： There is axis of symmetry beyond the current plane following G24. Description： The axis of symmetry following G24 must be in the current plane. Solution：1) If the axis command following G24 is correct, please select the current plane again. 2) If the current plane is correct, please modify the wrong axis of symmetry command following G24.
Alarm NO.： **No.1310** Wrong Reason： There is axis beyond the current plane following G25. Description： The axis command of point of symmetry following G25 must

be in the current plane.
Solution: 1) If the axis command following G25 is correct, please select the current plane again. 2) If the current plane is correct, please modify the wrong of symmetry command following G25.

Alarm NO.: **No.1320**
Wrong Reason: Target block which is specified by the skip command does not exist .
Solution: Add correct sequence number at the beginning of the target block.

Alarm NO.: **No.1321**
Wrong Reason: Subprogram which is called by M98 or G65 does not exit.
Solution: Check the program name, if not exist, rebuild it.

Alarm NO.: **No.1322**
Wrong Reason: The command M98 is half-baked.
Description: The program called must be specified by P following M98.
Solution: Complement the program name called by M98.

Alarm NO.: **No.1390**
Wrong Reason: R level is higher than initial level in canned cycle.
Description: R level must be lower than initial level in canned cycle.
Solution: heighten initial level or decline R level

Alarm NO.: **No.1391**
Wrong Reason: R level is higher than the bottom of the hole in canned cycle.
Description: R level must be lower than the bottom of the hole in canned cycle.
Solution: heighten R level or decline the bottom of the hole

Alarm NO.: **No.1392**
Wrong Reason: Wrong number of repeats is specified in the canned cycle.
Description: Number of repeats which is specified by K in the canned cycle must be greater than or equal to 0.
Solution: Specify right number of repeats.

Alarm NO.: **No.1400** Wrong Reason: Have not specified the coordinate value on the bottom of the hole in the canned cycle G73. Solution: Specify the correct coordinate value on the bottom of the hole by the command Z.
Alarm NO.: **No.1401** Wrong Reason: Have not specified R level in the canned cycle G73. Solution: Specify the correct R level by the command R.
Alarm NO.: **No.1402** Wrong Reason: Have not specified the depth of cutting for each cutting feed in the canned cycle G73. Solution: Specify the correct depth of cutting level by the command Q.
Alarm NO.: **No.1440** Wrong Reason: Have not specified the coordinate value on the bottom of the hole in the canned cycle G81. Solution: Specify the correct coordinate value on the bottom of the hole by the command Z.
Alarm NO.: **No.1441** Wrong Reason: Have not specified R level in the canned cycle G81. Solution: Specify the correct R level by the command R.
Alarm NO.: **No.1445** Wrong Reason: Have not specified the coordinate value on the bottom of the hole in the canned cycle G82. Solution: Specify the correct coordinate value on the bottom of the hole by the command Z.
Alarm NO.: **No.1446** Wrong Reason: Have not specified R level in the canned cycle G82. Solution: Specify the correct R level by the command R.
Alarm NO.: **No.1450**

Wrong Reason：Have not specified the coordinate value on the bottom of the hole in the canned cycle G83. Solution：Specify the correct coordinate value on the bottom of the hole by the command Z.
Alarm NO.：**No.1451** Wrong Reason：Have not specified R level in the canned cycle G83. Solution：Specify the correct R level by the command R.
Alarm NO.：**No.1452** Wrong Reason：Have not specified the depth of cutting for each cutting feed in the canned cycle G83. Solution：Specify the correct depth of cutting level by the command Q.
Alarm NO.：**No.1460** Wrong Reason：Have not specified the coordinate value on the bottom of the hole in the canned cycle G85. Solution：Specify the correct coordinate value on the bottom of the hole by the command Z.
Alarm NO.：**No.1461** Wrong Reason：Have not specified R level in the canned cycle G85. Solution：Specify the correct R level by the command R.
Alarm NO.：**No.1490** Wrong Reason：Have not specified the coordinate value on the bottom of the hole in the canned cycle G89. Solution：Specify the correct coordinate value on the bottom of the hole by the command Z.
Alarm NO.：**No.1491** Wrong Reason：Have not specified R level in the canned cycle G89. Solution：Specify the correct R level by the command R.
Alarm NO.：**No.1495** Wrong Reason：The setting value of the parameter P1081 is too small.

Description： The retraction value should be greater than 0.1 in the canned cycle, else an alarm is output. Solution： Modify the parameter P1081.
Alarm NO.： **No.1550** Wrong Reason： The command G43/G44 is half-baked. Description： The offset number for tool length must be specified following the G43/G44. Solution： Specify the correct offset number by the command H.
Alarm NO.： **No.1551** Wrong Reason： The offset number for tool length is ineffective following the G43/G44. Description: The offset number for tool length should be in the range which has been specified. Solution： Refer to the tool parameter list of CNC to specify the correct offset number.
Alarm NO.： **No.1580** Wrong Reason： The reference position specified by the command P following G30 is ineffective. Description: The 2nd、3rd and 4th reference position return functions can be used only after G30, so the parameter after the command P can be 2、 3 or 4 only. Return the 1st reference position using G28. Solution： Specify the correct reference number by the command P.
Alarm NO.： **No.1600** Wrong Reason： Have not specified G68.1 before G53.1. Description： The special coordinate system must be set by G68.1 before G53.1. Solution： Setting the special coordinate system before G53.1.
Alarm NO.： **No.1610** Wrong Reason： The 3D arc command can not be used when the polar coordinate mode is effective. Solution: Polar coordinate command should be cancelled by G15 before the

3D arc command.
Alarm NO.: **No.1611** Wrong Reason: The 3D arc command (G02.4/G03.4) is half-baked. Description: Middle point and end point of the arc should be specified by two blocks following G02.4/G03.4. Solution: Modify program to complement the command of G02.4/G03.4.
Alarm NO.: **No.3000** Wrong Reason: The line for rebuilding the cutter compensation is not straight line. Description: The program segment for rebuilding the cutter compensation must be straight line, an alarm is output if it is an arc. Solution: Modify it to be a straight line.
Alarm NO.: **No.3001** Wrong Reason: Tool interference. Description: The interference will occur if the distance between the tool compensation path and the programmed path is less than the tool radius. Example: During machining a internal circle, overcutting will occur if the tool radius is greater than circle radius. During machining a groove, overcutting will occur if the tool diameter is greater than the groove width. Solution: Modify program or replace the tool with one has a small radius.
Alarm NO.: **No.3015** Wrong Reason: The current plane (G17/G18/G19) can not be changed when the cutter compensation mode (G41/G42) is effective. Solution: The plane can be changed after cancelling the cutter compensation by G40.
Alarm NO.: **No.3020** Wrong Reason: Wrong plane for cutter compensation. Description: System supports cutter compensation on the plane of G17 only, an alarm is output if on the plane of G18/G19. Solution: Change the plane for cutter compensation.

Alarm NO.： **No.3022** Wrong Reason： The arc for compensation is not intersectant. Description： The tool path from arc to arc is not intersectant after cutter compensation. Solution： Check program and size of the tool.
Alarm NO.： **No.5000** Wrong Reason： Wrong format of G65 Solution： Refer to the right format of G65 in System programming manual.
Alarm NO.： **No.5001** Wrong Reason： Wrong format of IF Solution： Refer to the right format of IF in System programming manual.
Alarm NO.： **No.5002** Wrong Reason： Wrong format of WHILE Solution： Refer to the right format of WHILE in System programming manual.
Alarm NO.： **No.5003** Wrong Reason： Wrong format of mathematic expression
Alarm NO.： **No.5004** Wrong Reason： Wrong format of GOTO Solution： Refer to the right format of GOTO in System programming manual.
Alarm NO.： **No.5005** Wrong Reason： Have not specified the skip block after GOTO. Solution： Specify the correct skip block number by the command L
Alarm NO.： **No.5006** Wrong Reason： Wrong connection of the conditional expression after IF or WHILE. Description： The number of conditional expression to connect is two at most.

Solution： Modify program to satisfy the condition.
Alarm NO.： **No.5007** Wrong Reason： Wrong format of conditional expression after IF or WHILE. Solution： Modify program.
Alarm NO.： **No.5010** Wrong Reason： Wrong using start symbol '{' of block. Description： Start symbol '{' must be used with the command IF、WHILE and so on, and that can not be used alone. Solution： Refer to System programming manual to modify program.
Alarm NO.： **No.5011** Wrong Reason： Have not specified start symbol '{' of block after IF and WHILE. Description： Start symbol '{' must be specified after IF and WHILE. Solution： Refer to System programming manual to modify program.
Alarm NO.： **No.5012** Wrong Reason： No end symbol'}' of block. Description： Start symbol '{' must be used with end symbol '}'. There is a '{' corresponding to a '}', so the number of '{' is as same as the number of '}'. The alarm represents the number of '{' is more than the number of '}'. Solution： Modify program.
Alarm NO.： **No.5013** Wrong Reason： Redundant end symbol'}' of block. Description： End symbol '}' must be used with Start symbol '{'. There is a '}' corresponding to a'{', so the number of '{' is as same as the number of '}'. The alarm represents the number of '{' is less than the number of '}'. Solution： Modify program.
Alarm NO.： **No.5051** Wrong Reason：Have not specified variable after the variable symbol of '#'.

Alarm NO.：**No.5052**
Wrong Reason：Wrong variable has been specified after the variable symbol of '#'.
Description：Variable is composed of the number limited in a range, an alarm is output if it exceeds the range.
Solution：Refer to the definition of variable in System programming manual to use variable correctly.

Alarm NO.：**No.5080**
Wrong Reason：Have not specified the number of macro program after G65.
Solution：Specify the correct number of macro program by the command P.

Alarm NO.：**No.5081**
Wrong Reason： Wrong number of repeats has been specified after G65.
Description：The number of repeats should be greater than 0, else an alarm is output.
Solution：Specify the correct number of repeats by the command L.

Alarm NO.：**No.5090**
Wrong Reason：Have not specified the correct returning command after ending a subprogram.
Description：Subprogram must be ended by M99, an alarm is output if it ends by M99 or M30.
Solution：Subprogram must be ended by M99.

Alarm NO.：**No.6001**
Wrong Reason：Wrong parameter for the function TAN
Description：Parameter for the function TAN can not be the odd multiple of 90º, else an alarm is output.
Solution：Modify program.

Alarm NO.：**No.6002**
Wrong Reason：Wrong parameter for the function ASIN
Description：Parameter for the function ASIN should be in [-1,1], else an alarm is output.

Solution： Modify program.
Alarm NO.： **No.6003** Wrong Reason： Wrong parameter for the function ACOS Description： Parameter for the function ACOS should be in [-1,1], else an alarm is output. Solution： Modify program.
Alarm NO.： **No.6004** Wrong Reason： Wrong parameter for the function SQRT Description： Parameter for the function SQRT should be greater than or equal to 0, else an alarm is output. Solution： Modify program.
Alarm NO.： **No.6005** Wrong Reason： Wrong parameter for the function LN Description：Parameter for the function LN should be greater than 0, else an alarm is output. Solution： Modify program.