Volume Ⅱ OPERATION

VOLUME Ⅲ INSTALLATION

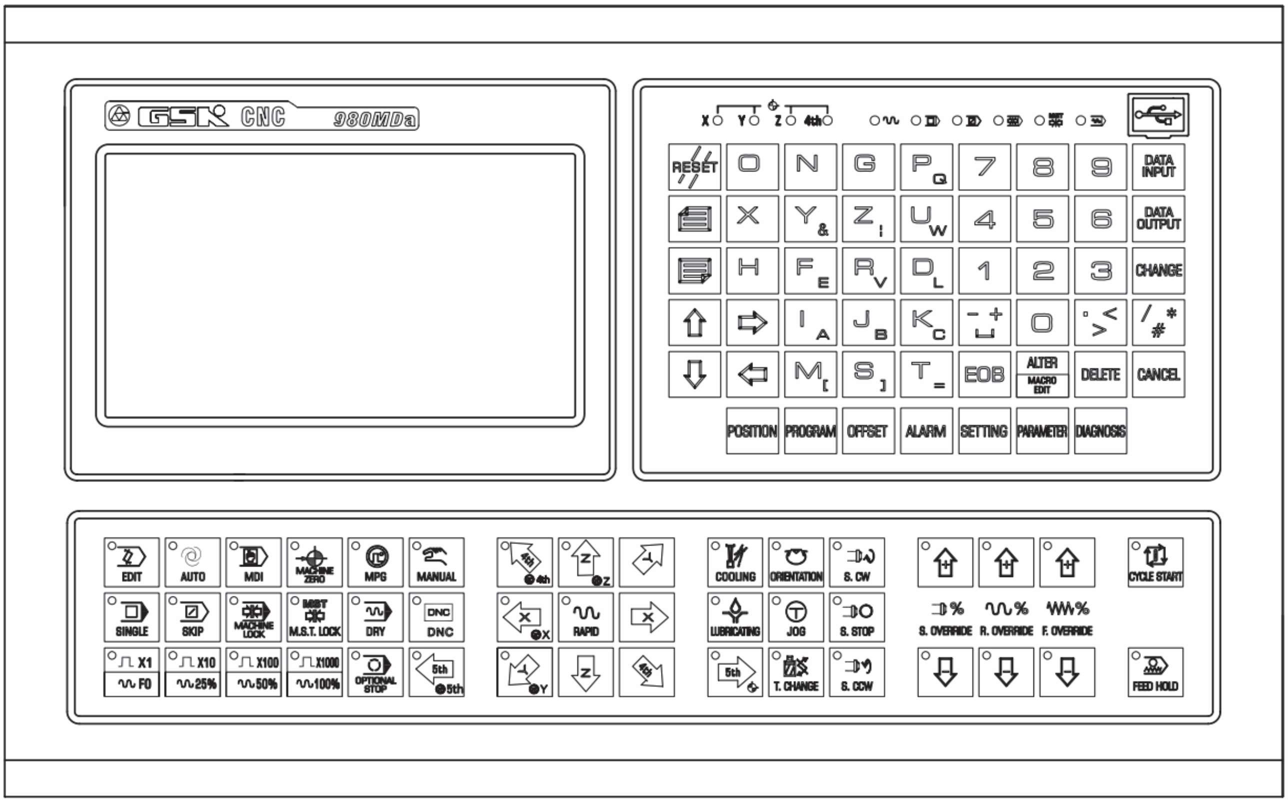
APPENDIX

VOLUME I PROGRAMMING

CHAPTER 1 PROGRAMMING FUNDMENTALS

1.1 Introduction

GSK980MDa Milling Machine is a new generation of CNC system developed by GSK Company. As the upgraded version of GSK980MD, it supports milling, boring and drilling cycle. It employs 32 bits high-capability CPU and very large scale programmable device FPGA, applies real-time multi-task control technology and hardware interpolation technology, and is able to perform μm level precision motion control and PLC logic control. GSK980MDa is the optimum choice for upgrading CNC milling machine.



Characteristics：

- ✓ Five axes control (X, Y, Z ,4th and 5th); 3 axes linkage; optional interpolation precision (1μm/0.1μm); maximum speed 60m/min; optional axis types (linear axis or revolving axis) for the 4th and 5th axes; CS axis control available for the 4th and 5th axes.
- ✓ Electronic gear ratio: (1～32767):(1～32767)
- ✓ Screw-pitch error compensation, backlash compensation, tool length compensation, tool abrasion compensation and tool nose radius compensation.
- ✓ Embedded with PLC can be downloaded to CNC from PC.
- ✓ DNC function supports for real-time program transmission for machining.
- ✓ Compatible with G commands in GSK980MC, GSK928MA and GSK980MD. 26 kinds of canned cycles, such as drilling/boring, circular/rectangular groove rough-milling, full circle/rectangular finish-milling, linear/rectangular/arc continuous drilling.
- ✓ Spindle encoder tapping and rigid tapping can be detected during tapping cycle, so that high precision machining can be performed.

- ✓ Metric/inch programming; automatic chamfering function and tool life management function.
- ✓ Chinese, English, Russian and Spanish display selected by the parameters.
- ✓ Full screen program editing; 40MB program capacity for storing up to 40000 of part programs.
- ✓ USB data communication; CNC system upgrading, machining programs reading through U disk and bidirectional transfer between CNC and U disk.
- ✓ Alarm log; multi-level passwords for equipment maintenance and management.
- ✓ Bidirectional transfer between CNC and CNC, CNC and PC; upgrade of CNC software and PLC programs;
- ✓ The installation dimensions and the electric ports are compatible with GSK980MD, GSK980MC.

Specifications

Motion control	Controlled axes: five axes (X,Y,Z,4th and 5th); (for the 4th and 5th axes) optional axis types (linear axis or revolving axis) and CS contouring control available;
	Interpolation functions: linear interpolation (for X, Y, Z, 4th and 5th axes); helical interpolation (for X, Y and Z axes); circular interpolation (for arbitrary 2 axes).
	Position command range: -99999999～99999999; least command increment: 1μm/0.1μm; (selected via parameters)
	Electronic gear ratio: command multiplier 1～32767, command frequency divisor 1～32767
	Rapid traverse speed: maximum 60000mm/min Rapid traverse override: F0, 25%, 50%, 100% four levels real-time tuning
	Cutting feedrate: maximum 15000mm/min (feed per min.) or 500mm/r. (feed per rotation) Feedrate override: 0～150% sixteen-level real-time tuning
	Manual feedrate: 0～1260mm/min sixteen-level real-time tuning
	MPG feed: 0.001, 0.010, 0.100,1.000mm four gears.
	Acceleration/deceleration type: S-type for rapid traverse; exponential-type for cutting feed.
	Automatic chamfering
G Code	65 kinds of G codes：G00, G01, G02, G03, G04, G10, G11, G17, G18, G19, G20, G21, G28, G29, G30, G31, G40, G41, G42, G43, G44, G49, G54, G55, G56, G57, G58, G59, G65, G66, G67, G73, G74, G80, G81, G82, G83, G84, G85, G86, G88, G89, G90, G91, G92, G94, G95, G98, G99, G110, G111, G112, G113, G114, G115, G134, G135, G136, G137, G138, G139, G140, G141, G142, G143
Macro command	31 kinds of arithmetic, logical operations and skip can be achieved by macro command G65
	Macro statement command. eg:IF,WHILE,GOTO
Operation mode	Seven operation modes: EDIT, AUTO, MDI, DNC, MACHINE ZERO, MPG/STEP and MANUAL.
Tapping	Tapping function: lead 0.001～500mm or 0.06～25400 pitch/inch

	Encoder tapping: settable line number of encoder（0 or100p/r～5000p/r）; no detect for spindle encoder (when the line number is set to 0)
	Rigid tapping: by rotary axis
	Drive ratio between encoder and spindle:（1～255）:（1～255）
Precision compensation	Backlash compensation: 0～2.000mm
	Pitch error compensation: 255 compensation points per axis; compensation amount of each point: ±0.255mm.
	Tool compensation: 32 groups tool length compensation, tool wear compensation, cutter compensation C
M command	Special M commands (redefinition unallowed): M02,M29, M30, M98, M99,M9000～M9999. Other M □□ commands are defined or disposed by PLC program.
	M commands defined by standard PLC program: M00, M03, M04, M05 M08, M09, M10, M11, M32, M33
T command	tool number T01～T32 (32 numbers at most); manual tool change or auto-tool change selected by the parameters; auto tool change sequence set by PLC program.
	Tool life management; 32 groups, 8 kinds/groups of tool life management data
Spindle speed control	Speed switching value control: S □□ command is defined or disposed by PLC program; the standard PLC programs S1, S2, S3 and S4 directly output; The output of S1,S2, S3, and S4 are closed by S0.
	Speed analog voltage control: the spindle speed per minute commanded by S codes; output 0～10V voltage to spindle converter; spindle stepless speed changing supports 4 spindle mechanical gears
PLC function	9 kinds of basic commands; 23 kinds of function commands; 2-level PLC program involving up to 5000 steps (2μs processing time for each step). 8ms refresh cycle for the first level program; Ladder diagram edit software and communication software downloadable
	Integrated machine panel: 44 points input (key), 44 points output (LED) Basic I/O: 41 points input/ 36 points output
Display interface	Displayer: 480×234 lattice, 7'' wide-screen multi-color LCD,
	Display modes: Chinese, English, Russian, Spanish display selected by parameters; machining path displayable
Program edit	Capacity: 40MB for up to 40000 part programs; custom macro program call; 4 nesting-levels of subprogram
	Edit modes: full-screen editing; absolute/incremental programming
USB	CNC system upgrade
	Part programs reading in USB
	Bidirectional files transfer between CNC and USB (including programs, parameters, PLC backup and recovery)
Clock display	Clock, date and week display.
Serial Communication	bidirectional transfer between CNC and PC, CNC and CNC (involving programs, parameters, tool compensation data); download and upgrade of system software and PLC program serial ports

Matching drive unit	AC servo or step drive device by using the pulse+direction signal input. (DA98 or DY3 series)

G Code Table

Code	Function	Code	Function	Code	Function
G00	Positioning (rapid traverse)	*G54	Workpiece coordinate system 1	G92	Coordinate system setting
*G01	Linear interpolation	G55	Workpiece coordinate system 2	*G94	Feed per min.
G02	Circular/helical interpolation (CW)	G56	Workpiece coordinate system 3	G95	Feed per rotation
G03	Circular/helical interpolation (CCW)	G57	Workpiece coordinate system 4	*G98	Return to initial plane in canned cycle
G04	Dwell, exact stop	G58	Workpiece coordinate system 5	G99	Return to R point in canned cycle
G10	Tool life management	G59	Workpiece coordinate system 6	G110	Inner circle groove roughing (CCW)
G11	Tool life management end	G65	Macro program/ macro code	G111	Inner circle groove roughing (CW)
*G17	XY plane selection	G66	Macro program modal call	G112	Inner circle finishing (CCW)
G18	ZX plane selection	*G67	Macro program modal call cancel	G113	Inner circle finishing (CW)
G19	YZ plane selection	G73	High-speed peck drilling	G114	Circular outer finish milling (CW)
G20	Inch input	G74	Counter tapping cycle	G115	Outer circle finishing (CCW)
G21	Metric input	*G80	Canned cycle cancel	G134	Rectangular groove roughing (CCW)
G28	Reference position return	G81	Drilling cycle (spot drilling cycle)	G135	Rectangular groove roughing (CW)
G29	Return from reference position	G82	Drilling cycle (stepped hole boring cycle)	G136	Rectangular groove inner finishing (CCW)
G30	2nd, 3rd, 4th, reference position return	G83	Peck drilling cycle	G137	Rectangular groove inner finishing (CW)
G31	Skip function	G84	Tapping cycle	G138	Rectangular outer finishing (CCW)
*G40	Cutter compensation cancel	G85	Boring cycle	G139	Rectangular outer finishing (CW)
G41	Cutter compensation left	G86	Drilling cycle	G140	Rectangular continuous drilling (CW)
G42	Cutter compensation right	G88	Boring cycle	G141	Rectangular continuous drilling (CCW)

G43	Tool length compensation □ direction	G89	Boring cycle	G142	□rc continuous drilling (CW)
G44	Tool length compensation □ direction	*G90	□bsolute programming	G143	□rc continuous drilling (CCW)
*G49	Tool length compensation cancel	G91	Incremental programming		

□ote□□ a□□ □□□□ □ean□initial □tate□

□□C Code□ □i□t

Code	Function	Code	Function	Code	Function
LD	□ormal open contact read	S□T	Setting	SP□	Subprogram end
LDI	□ormal closed contact read	RST	Resetting	□DDB	Binary addition
O□T	Output coil	CMP	Comparison setting	S□BB	Binary subtraction
□□D	□ormal open contact in series	CTRC	Counter	□LT	□lternative output
□□I	□ormal closed contact in series	TMRB	Timer	DIF□	Differential up
OR	□ormal open contact in parallel	CODB	Binary code transformation	DIFD	Differential down
ORI	□ormal closed contact in parallel	ROTB	Binary rotational control	MO□□	Logical □□D
ORB	Serial block in parallel	MO□□	Data copy	P□RI	Parity check
□□B	Parallel block in series	D□CB	Binary decode	LBL	Program skip numbering
□□D1	first level program end	□MPB	□ump	C□LL	Subprogram call
□□D2	Second level program end	SP	Subprogram numbering		

1.2 Program □xecution

□□□□ □□o□□a□ □□ecution □e□uence

The current program can only be run in automatic mode. GS□980MDa cannot run more than 1 program at the same time, so only one program can be performed at a time. The cursor is ahead of the first block when a program is opened, and can be moved in □DIT mode. In automatic mode, when the machine is in stop state, the cycle start signal (

key on the panel or external cycle start signal) enables the program to be run from the block where the cursor is located. □sually, blocks are executed in se□uence programmed in advanced. Program stops running till M02 or M30 is executed. The cursor moves along with program execution. The program execution sequence or state will be changed in following conditions:

- Program running stops when RESET key or the Emergency Stop button is pressed;

  Program running stops when the CNC alarm or PLC alarm occurs;

- When the system is switched in EDIT or MDI mode, program stops running after the current block is executed. After switching to automatic mode again, when CYCLE START key on the panel is pressed or external cycle start signal is ON, the program runs from the block where the cursor is located.

- If the operation mode is switched to MANUAL/MPG/STEP/MACHINE ZERO RETURN mode when the program is running, the execution dwells; after switching to automatic mode again, when CYCLE START key on the panel is pressed or external cycle start signal is ON, the program runs from where it stops.

- The execution dwells when FEED HOLD key is pressed or external pause signal is cut off; program starts running from where it stops when CYCLE START key on the panel is pressed or external cycle start signal is ON;
- The program dwells at the end of each block when the single block switch is on; after pressing CYCLE START key or switching on external cycle signal, program continuously runs from the next block;
- Blocks with mark "/" is skipped when the skip switch is ON.
- The object block is executed when command G65 or macro program skip (GOTO) is specified.
- When M98 or M9000～M9999 command is performed, the corresponding subprogram or macro program is called; M99 is executed at the end of the subprogram or macro program, after returning to the main program, the subsequent block (the one after the block in which the subprogram is called) is executed. (return to a specified block, if it is commanded by M99);
- When M99 command is specified in the middle of a main program which is not called by other programs, the current program is repeatly executed after returning to the head of the program.

□□□□ Word Execution Sequence within Block

When multiple words (such as G, X, Y, Z, F, R, M, S, T,) are in one block, most of M, S, and T words are interpreted by NC and sent to PLC for processing. Other words are processed by NC directly. M98, M99, M9000～M9999 and S word (which specify the spindle speed in r/min, m/min) are directly processed by NC as well.

When G words share the same block with M00, M01, M02 and M30, M words are executed after G words, and NC sends corresponding signals to PLC for processing.

When the G words share the same block with the M98, M99, M9000～M9999, these M words are performed by NC after G words (the M signal not sent to PLC).

When G words and M, S, T words share the same block, PLC program (ladder diagram) determines the execution conse□uence (executed at the same time or G words before M, S, T words). Refer to the manual from tool builder for relevant words execution se□uence.

1.3 Basic □xes Increment System

The increment system consists of the least input increment (for input) and least command increment (for output). The least input increment is the minimum unit for programming moving distance. The least command increment is the minimum unit for moving the tool on the machine. Both increments are represented in mm,inches.or deg.

The basic axes herein means X, Y, Z axes. The basic increment system includes IS-B and IS-C types which can be selected by bit ISC of parameter □O.038.

□□□	□□C							

ISC □1：The increment system is IS-C(0.1□)；

□0：The increment system is IS-B(1□)

In different increment system, different pulse output type enables different output speed. (Selected by bit □BPx of parameter □O.039)

□□□				□□□□	□□□□	□□□□	□□□□	□□□□

□BPx □1：The impulse mode of axis is □B phases□

□0：The impulse mode of axis is impulse and direction.

□□□□ □□eed o□□nc□e□ent □□□te□ □

□ut□ut □ode	□□eed			
	□u（□□□）		□□□u（□□C）	
	Metric machine system (mm/min)	Inch machine system (inch/min)	Metric machine system (mm/min)	Inch machine system (inch/min)
□ul□e □di□ection	60,000	6,000	6,000	600
□□ □uad□atu□e □□a□e	240,000	24,000	24,000	2,400

□□□□ □nit o□□nc□e□ent □□□te□ □

In different increment system, the least input/output increment varies with metric/inch system. The specific data is shown as follows□

□u（□□□）		□ea□t in□ut inc□e□ent □o□in□ut□	□ea□t co□□and inc□e□ent □o□out□ut□
Metric machine system	Metric input (G21)	0.001 (mm)	0.001 (mm)
		0.001 (deg)	0.001 (deg)
	Inch input (G20)	0.0001 (inch)	0.001 (mm)
		0.001 (deg)	0.001 (deg)
Inch machine	Metric input (G21)	0.001 (mm)	0.0001 (inch)
		0.001 (deg)	0.001 (deg)

system	Inch input (G20)	0.0001 (inch)	0.0001 (inch)
		0.001 (deg)	0.001 (deg)

0.1u (IS-C)		Least input increment (for input)	Least command increment (for output)
Metric machine system	Metric input (G21)	0.0001 (mm)	Metric machine system
		0.0001 (deg)	
	Inch input (G20)	0.00001 (inch)	
		0.0001 (deg)	
Inch machine system	Metric input (G21)	0.0001 (mm)	Inch machine system
		0.0001 (deg)	
	Inch input (G20)	0.00001 (inch)	
		0.0001 (deg)	

Least input increment (for input) is metric or inch can be set by G20 or G21.

Least command increment (for output) is metric or inch is determined by machine tool and set by bit SCW of parameter NO.004.

Data Range of Increment System

Limited by pulse output frequency, the data ranges may vary due to different increment system.

Increment system		Command data input range	Data format
1u (IS-B)	Metric input (G21)	-99999.999～99999.999 (mm) -99999.999～99999.999 (deg)	5.3 5.3
	Inch input (G20)	-9999.9999～9999.9999 (inch) -9999.999～9999.999 (deg)	4.4 4.3
0.1u (IS-C)	Metric input (G21)	-9999.9999～9999.9999 (mm) -9999.9999～9999.9999 (deg)	4.4 4.4
	Inch input (G20)	-999.99999～999.99999 (inch) -999.9999～999.9999 (deg)	3.5 3.4

Note: 5.3 in the table above indicates 5 integers and 3 decimals, the other data are alike.

Data Range and Unit of Increment System

- **Speed Parameter**

Machine tool types decide the units of linear axes speed, i.e. mm/min for metric machine system is 0.1inch/min for inch machine system.

The range of linear axis speed parameter is codetermined by machine tool type and increment system.

For example, data parameter NO.070: upper limit of cutting feedrate.

Machine tool type	Increment system	Linear axis speed unit	Parameter range	Rotary axis speed unit
Metric machine system	1 u（IS-B）	mm/min	10～60000	deg/min
	0.1u （IS-C）		10～6000	
Inch machine system	1 u（IS-B）	0.1inch/min	5～60000	
	0.1u（IS-C）		5～6000	

As rotary axes are not involved in metric-inch interconversion, the rotation speed unit is always deg/min.

The switch between different increment systems may cause the excess of permitted running speed set by data parameter. Therefore, at the first power-on after switching, the system automatically modifies relevant speed parameters and gives an alarm.

- **Increment parameter**

The unit and range of linear axis speed parameter are codetermined by machine tool type and increment system.

For example, parameter O135, X axis software limit.

Machine tool type	Increment system	Linear axis increment unit	Linear axis parameter range
Metric machine system	1 u（IS-B）	0.001mm	-99,999.999～99,999.999
	0.1u（IS-C）	0.0001 mm	-9,999.9999～9,999.9999
Inch machine system	1 u（IS-B）	0.0001inch	-9,999.9999～9,999.9999
	0.1u（IS-C）	0.00001 inch	-999.99999～999.99999

As rotary axes are not involved in metric-inch interconversion, the rotary axis increment parameter unit is determined by increment system types. The ranges of rotary axis increment parameters are the same as that of metric machine tool.

Machine tool type	Increment system	Rotation axis speed unit	Rotation axis parameter range
Metric, inch machine tool system	1 u（IS-B）	0.001deg	0～99999.999
	0.1u（IS-C）	0.0001 deg	0～9999.9999

- **Coordinate data（G[illegible]～G[illegible]）**

The unit of linear axis coordinate data is determined by metric/inch input system, namely, mm for metric system, inch for inch system.

The ranges of linear axis coordinate data are codetermined by metric/inch input system and increment system. It is the same as command data input ranges. Shown as follows:

Increment system		Linear axis coordinate data range
1 u（IS-B）	Metric input (G21)	-99999.999 ～99999.999(mm)
	Inch input (G20)	-9999.9999 ～9999.9999(inch)
0.1u（IS-C）	Metric input (G21)	-9999.9999 ～9999.9999(mm)
	Inch input (G20)	-999.99999 ～999.99999(inch)

As rotary axis is not involve in metric-inch interconversion, the unit of rotary axis coordinate data is deg. The ranges of rotary axis coordinate data is the same as linear axis coordinate data ranges in metric system.

Input type	Increment system	Rotary axis coordinate data range
Metric, inch input	1 u（IS-B）	-99999.999 ～99999.999（deg）
	0.1u（IS-C）	-9999.9999 ～9999.9999(deg)

- **Tool compensation data**

The unit of tool compensation data is determined by metric/inch input system, namely, mm for metric input, inch for inch input.

The range of tool compensation data is limited as 9999999, determined by inch input system and increment system. It is smaller than command data. Shown as follows：

Input type	Increment system	Tool compensation data unit	Tool compensation data range
Metric input (G21)	1 u（IS-B）	mm	±9999.999
	0.1u（IS-C）		±999.9999
Metric input (G21)	1 u（IS-B）	inch	±999.9999
	0.1u（IS-C）		±99.99999

- **Screw pitch error compensation data**

The unit and range of linear axis screw-pitch error compensation data is codetermined by machine tool type and increment system.

Shown as following table：

Machine tool type	Increment system	Linear axis screw pitch error compensation data unit	Linear axis screw pitch error compensation data range
Metric tool machine system	1 u（IS-B）	0.001mm	-255～255
	0.1u（IS-C）	0.0001mm	-2550～2550
Inch tool machine system	1 u（IS-B）	0.0001inch	-255～255
	0.1u（IS-C）	0.00001inch	-2550～2550

Rotary axes are not involved in metric-inch conversion. The unit of rotary axes screw-pitch error compensation is determined by increment system. The range is the same as that of the metric machine tool.

Machine tool system	Increment system	Rotary axis screw pitch error compensation unit	Rotary axis screw pitch error compensation range
Metric, inch machine system	1 u（IS-B）	0.001deg	0～255
	0.1u（IS-C）	0.0001 deg	0～2550

- **Graphic setting data**

The maximum and minimum data ranges of X, Y, Z set by graph is in accordance with the command data ranges.

Increment system		Graphic setting data ranges
1 u（IS-B）	Metric input (G21)	-99999.999～99999.999 (mm)
	Inch input (G20)	-9999.9999～9999.9999 (inch)
0.1u（IS-C）	Metric input (G21)	-9999.9999～9999.9999 (mm)
	Inch input (G20)	-999.99999～999.99999 (inch)

The units and ranges of some address values

- **Definition and ranges of the pitch：**

	Code	1μ（IS-B）	0.1μ（IS-C）	Unit
Input in Metric（G21）	F	0.001～500.000	0.0001～500.00	mm/pitch（lead）
	I	0.06～25400	0.06～2540	Pitch（lead）/inch
Inch input（G20）	F	0.0001～50.00	0.00001～50.0	inch//pitch（lead）
	I	0.06～2540	0.06～254	Pitch（lead）/inch

- **Speed F definition**

G94：feed per minute, F unit：mm/min

G95：feed per rotation, F definition and ranges are as follows：

	□μ（□□□）	□□μ（□□C）	Unit
Metric input（G□□）	0.001～500.000	0.0001～500.0000	mm/revolution
Inch input（G□□）	0.0001～50.0	0.00001～50.0	inch/revolution

1.4 Additional Axes Increment System

In the least increment system (IS-B or IS-C), under the condition that the additional axes are not involved in simultaneous control and just used for separate motion (such as feeding), and the requirement for precision is not high, when the least increment is 0.01, the feedrate will be much faster, greatly increasing the efficiency. Therefore, the additional axes least increment system is not necessary to be in accordance with the current least increment system. To meet various requirements of users, the system adds optional function to least increment system.

Additional axes increment system is set by state parameter No.026, No.028. Shown as follows：

□□□	□□□□	□□□□		RCS4			ROS4	ROT4

□4IS1, □4IS0：Select increment system of 4th.

□□□□	□□□□	Increment system of □□T□	Least input/output
0	0	Same to the X, Y, Z	
0	1	IS-□	0.01
1	0	IS-B	0.001
1	1	IS-C	0.0001

□□□	□□□□	□□□□		RCS5			ROS5	ROT5

□5IS1, □5IS0：Selecte increment system of 5th.

□□□□	□□□□	Increment system of □□T□	Least input/output
0	0	Same to the X, Y, Z	
0	1	IS-□	0.01
1	0	IS-B	0.001
1	1	IS-C	0.0001

Note: the least input/output in the table above are described without considering the metric/inch system and rotation axes.

□.□.□ Additional Axes in Current Increment System

When IS-B or IS-C is selected, the speed and range of additional axes are the same as described in 1.3.

□.□.□ Additonal Axes in IS-□ Increment System

When IS-□ is selected, the maximum speed of additional axes can reach 100 times of that of IS-B and IS-C. The relevant data and parameters ranges are the same as that of the current basic axes increment system. (Refer to section 1.3)

CHAPTER 2 MSTF CODES

2.1 M Codes (Miscellaneous Function)

The M codes are composed by code address M and 1、2 or 4 digits after the codes M is used for controlling the program execution or outputting M code to PLC.

M □□□□

Codes value (00～99, 9000～9999，leading zero can be omitted)

Address

M98, M99 and M9000～M9999 are independently processed by CNC, and the M codes are not output to PLC.

The function of M29 is fixed, namely, to output M codes to PLC.

The M02 and M03 are defined as program END codes by NC, meanwhile it also outputs M codes to PLC for the I/O control (spindle OFF, cooling OFF control etc.).

The PLC program can not change the meaning of the above-mentioned codes when the M98, M99 and M9000～M9999 are regarded as program CALL codes and the M02 and M30 are regarded as program END codes. The codes of other M codes are all output to PLC program for specifying the code function，please refer to the manual issued by machine tool manufacturer.

One block only has one M code. The CNC alarm occurs when two or more M codes are existed in one block.

Table 2-1 M code table for program execution

Code	Function
M02	End-of-Run
M29	Rigid tapping designation
M30	End-of-Run
M98	Subprogram call
M99	Return from the subprogram，the program will be repeatly executed If the code M99 is used for main program ending (namely, the curren program is not called by other programs).
M9000～M9999	Call macro program (Program No. is larger than 9000)

2.1.1 End of Program M02

Format：M02

Function：The M02 code is executed in the Auto mode. The automatic run is ended after the other codes of current block are executed，the cursor stops in the block in which the M02 is located and does not return to the head of the program. If the program is to be executed again, the cursor should return to the beginning of the program.

Besides the above-mentioned functions processed by CNC, the functions of code M02 also can be defined by the PLC ladder diagram. The function defined by standard ladder diagram can be：the current input state of CNC is not change after the code M02 is executed.

2.1.2 Rigid Tapping Designation M29

Format：M29

Function：In auto mode, after the execution of M29, the G74, G84 that followed is processed as在自

rigid tapping codes.

End of Run M30

Format：M30

Function：If M30 command is executed in the Auto mode, the automatic run is ended after the other commands of current block are executed；the system cancels the tool nose radius compensation and the cursor returns to the beginning of the program when the workpieces number is added by one (whether the cursor returns to the head of the program is determined by parameters).

The cursor does not return to the beginning of the program when the BIT4 of parameter No.005 is set to 0；when it is set to 1, the cursor returns to the beginning of the program as soon as the program execution is finished.

Besides the above-mentioned functions processed by CNC, the functions of code M30 also can be defined by the PLC ladder diagram. The function defined by standard ladder diagram can be：turn OFF the M03, M04 or M08 output signal after the M30 command is executed, and meanwhile output M05 signal.

Subprogram Call M98

Format：M98 P○○○○□□□□

The called subprogram No.（0000～9999）.The leading zero of subprogram can be omitted when the called times are not input; the subprogram No. should be 4 digits when the called times is input;

Called times（1-9999）calling for once, the input can be omitted

Function：In Auto mode, when the M98 is executed, the subprogram specified by P is called after the execution of other codes in the current block. The subprogram can be performed 9999 times at most. M98 cannot be performed in MDI, or an alarm will occur.

Return from Subprogram M99

Format： M99 P○○○○

The block No. (0000～9999) when return to main program is executed, the leading zero can be omitted.

Function：(in subprogram) as the other commands of current block are executed, the block specified by P is performed continuously when the main program is returned. The next block is performed continuously by calling current subprogram of M98 command when returning to the main program，because of the P is not given. If the main program is ended by using the M99 (namely, the current program is not called by other programs for execution), the current program will be run circularly. So, the M99 command is disabled in MDI.

Example：Fig. 2-1shows that the execution route of the subprogram is called (the P command within M99). Fig. 2-2 shows that the execution route of the subprogram is called (the P command is not in M99.

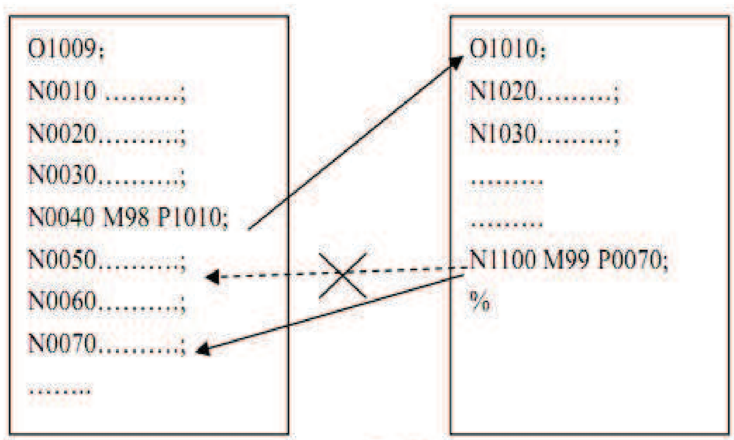


Fig. 2-1

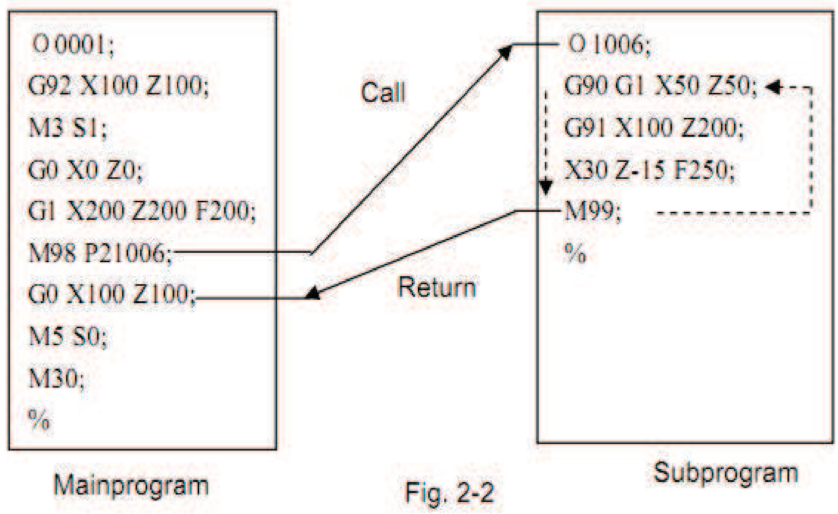


Fig. 2-2

This □□□9□0□□a can calls □uadruple subprogram, namel□, the other subprogram can be called from the subprogram. (□ee □ig. □□□)

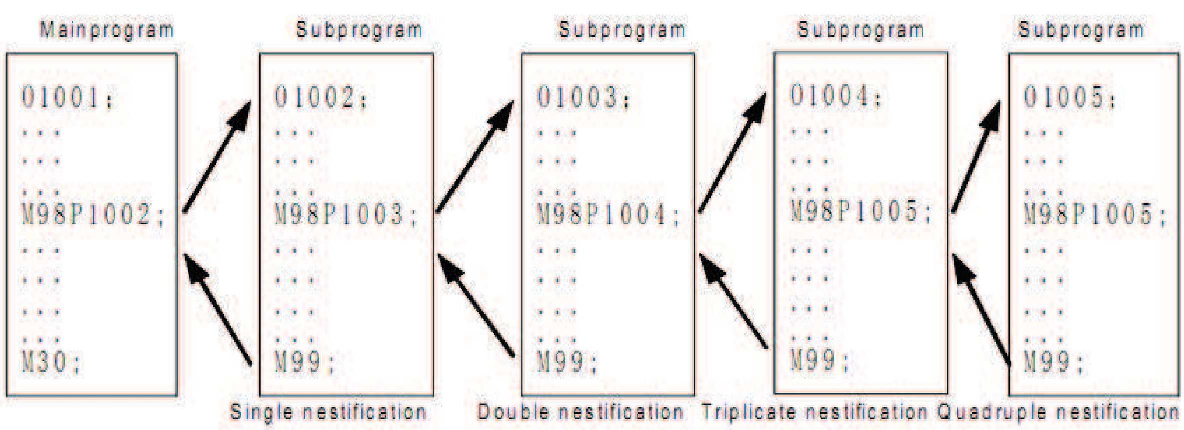


Fig. 2-3 Subprogram nestifications

2.1.6 Macro program call (M9000~M9999)

[illegible]

2.1.7 M command defined by standard PLC ladder diagram

[illegible]

Command	Function	Remark
[illegible]	[illegible]	
[illegible]	[illegible]	[illegible]
[illegible]	[illegible]	
[illegible]	[illegible]	
[illegible]	[illegible]	[illegible]
[illegible]	[illegible]	
[illegible]	[illegible]	[illegible]
[illegible]	[illegible]	

Note: The command with " * " specified by standard PLC is valid when the power is on.

2.1.8 Program stop M00

[illegible]

2.1.9 Spindle CCW, CW, stop control(M03, M04 and M05)

[illegible]

Note: The control time se□uence and logic of M03, M04 and M05 are specified by standard □□C □□o□□a□ □□e□e□to t□e □□□endi□ o□t□i□ □anual□

2.1.10 Cooling control (M08, M09)

[illegible]

Note: The control time se□uence and logic of M08 and M09 are specified by standard PLC □□o□□a□ □□e□e□to t□e □□□endi□ o□t□i□ □anual□

2.1.11 Lubricating control (M32,M33)

[illegible]

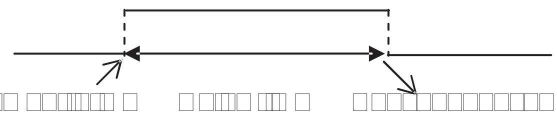
Note: The control time se□uence and logic of M32 and M33 are specified by standard PLC □□o□□a□ □□e□e□to t□e □□□endi□ o□t□i□ □anual□

[illegible]

[illegible]

2.2.1 Spindle Speed Switch Value Control

[illegible]



2.2.2 Spindle speed analog voltage control

2.2.3 Spindle override

2.4.1 Cutting feed (94 95, F command)

Volume I Programming

Note: In G95 mode, the cutting feedrate will be uneven when the spindle speed is less than 1 rev./min. The following error will exist in the actual feedrate when the spindle speed vibration occurs.

To guarantee the machine quality, it is recommended that the spindle speed selected in machining is not less than the lowest speed of available torque exported by spindle servo or inverter.

$$f_x = \frac{d_x}{\sqrt{d_x^{\square} + d_y^{\square} + d_z^{\square} + d_{\square}^{\square} + d_{\square}^{\square}}} \bullet F$$

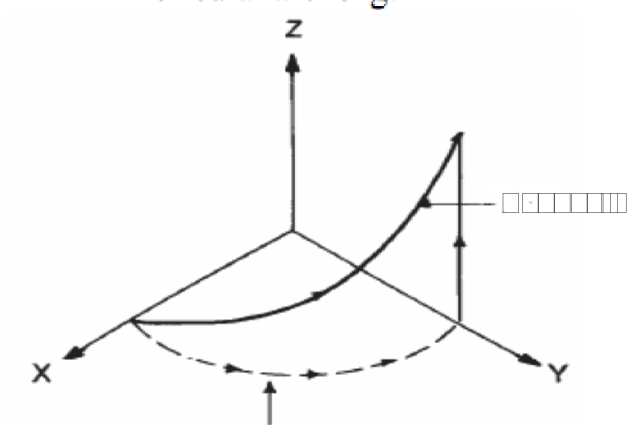
$$f_y = \frac{d_y}{\sqrt{d_x^{\square} + d_y^{\square} + d_z^{\square} + d_{\square}^{\square} + d_{\square}^{\square}}} \bullet F$$

$$f_z = \frac{d_z}{\sqrt{d_x^{\square} + d_y^{\square} + d_z^{\square} + d_{\square}^{\square} + d_{\square}^{\square}}} \bullet F$$

$$f_{\square} = \frac{d_{\square}}{\sqrt{d_x^{\square} + d_y^{\square} + d_z^{\square} + d_{\square}^{\square} + d_{\square}^{\square}}} \bullet F$$

$$f_{\square} = \frac{d_{\square}}{\sqrt{d_x^{\square} + d_y^{\square} + d_z^{\square} + d_{\square}^{\square} + d_{\square}^{\square}}} \bullet F$$

$$f = F \times \frac{\text{linear axis length}}{\text{circular arc length}}$$



Feedrate along the circle between 2 arc interpolation axes is the specified one.

Volume I Programming

[illegible]

2.4.2 Manual feed

[illegible]

Feedrate override(□)	[illegible]	[illegible]	[illegible]	[illegible]	[illegible]	[illegible]	[illegible]	[illegible]	[illegible]	[illegible]	[illegible]	[illegible]	[illegible]	[illegible]	[illegible]	[illegible]
Manual feedrate (mm□min)	[illegible]	[illegible]	[illegible]	[illegible]	[illegible]	[illegible]	[illegible]	[illegible]	[illegible]	[illegible]	[illegible]	[illegible]	[illegible]	[illegible]	[illegible]	[illegible]

Note: The manual feedrate of □ a□is is diameter variation per minute□the feedrate defined by □S□980M□a standard PLC ladder diagram is memori□ed when the power is turned off.

[illegible]

2.4.3 MP□□Step feed

[illegible]

2.4.4 □utomatic acceleration or deceleration

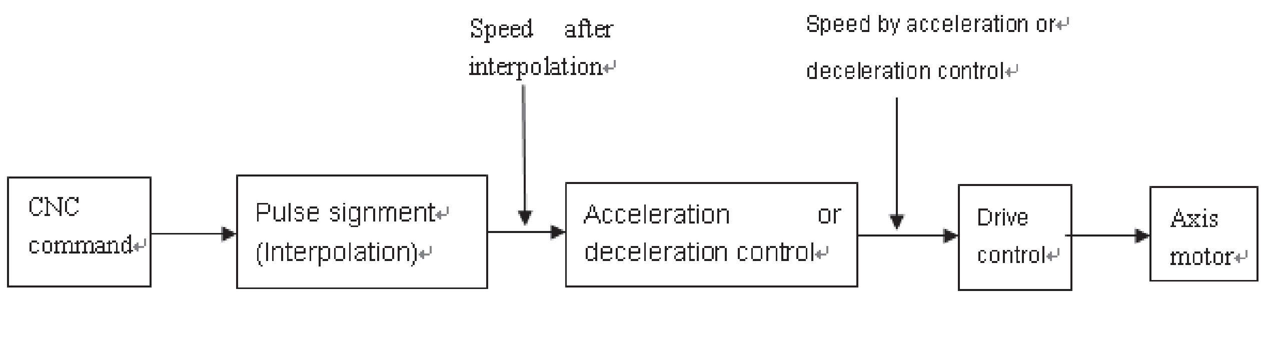
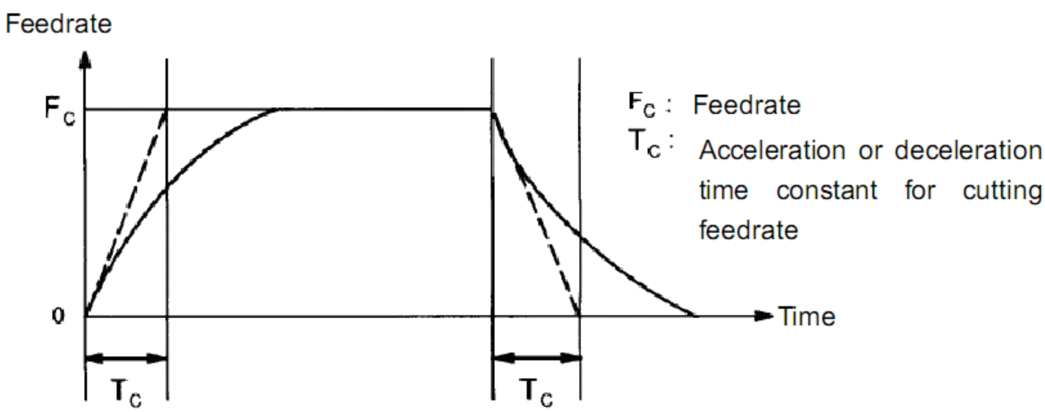


Fig. 2.9



F_C: feedrate

T_C: The acceleration or deceleration time constant of cutting feedrate

(Data parameter No.072 and No.074)

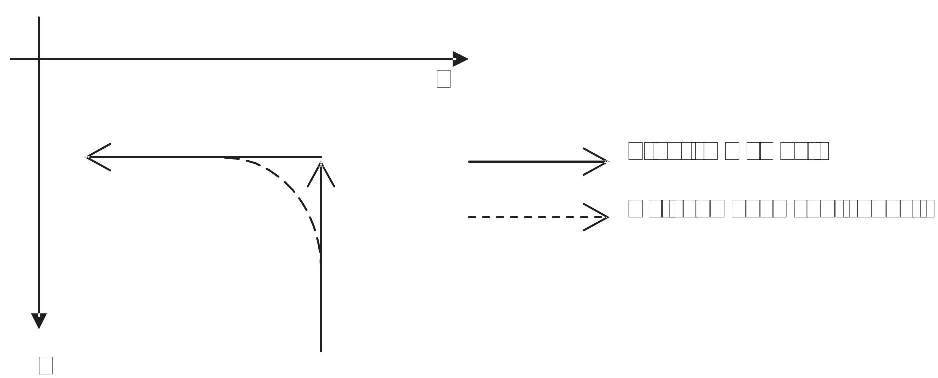
Fig. 2-11 Curves for cutting and manual feedrate

Volume I Programming

Previous block / Next block	Rapid Position	Cutting feed	Without move
Rapid positioning	□	□	□
Cutting feed	□	□	□
Without move	□	□	□

Note: □: The subse□uent block is performed after the previous block is accurately positioned at the end of the block.

□: □ach a□is speed is transmitted according to the acceleration or deceleration between the ad□acent blocks□an arc transition is formed at the meeting point of the tool path.



C[illegible]PT[illegible]R3 [illegible] C[illegible]MM[illegible]N[illegible]

[illegible]

[illegible]

[illegible]

[illegible]

[illegible]

[illegible]

[illegible]

Command word	[illegible]roup	Function	Remark
[illegible]	[illegible]	[illegible]	[illegible]
[illegible]		[illegible]	
[illegible]		[illegible]	
[illegible]		[illegible]	
[illegible]		[illegible]	
[illegible]		[illegible]	
[illegible]		[illegible]	
[illegible]	[illegible]	[illegible]	[illegible]
[illegible]		[illegible]	
[illegible]		[illegible]	
[illegible]		[illegible]	
[illegible]		[illegible]	
[illegible]		[illegible]	
[illegible]		[illegible]	
[illegible]		[illegible]	
[illegible]		[illegible]	
[illegible]		[illegible]	
[illegible]		[illegible]	
[illegible]		[illegible]	
[illegible]		[illegible]	
[illegible]		[illegible]	

Volume I Programming

[illegible]		[illegible]	
[illegible]		[illegible]	

3.1.1 Modal, non[illegible]modal and initial state

[illegible]

[illegible] **modal** [illegible] **command.** [illegible]

[illegible] **non[illegible]modal** [illegible] **command.**

[illegible] **initial** [illegible] **command**[illegible]

3.1.2 [illegible]amples

[illegible]

[illegible];

[illegible] (Move to G17 plane X100 Y100 at the rapid traverse rate; modal command G0 and G17 valid)

X20 Y30; (Move to X20 Y30 at the rapid traverse rate; modal command G0 can be omitted)

G1 X50 Y50 F300; (Linear interpolation to X50 Y50, feedrate is 300mm/min; modal command G1 valid)

X100; (Linear interpolation to X100 Y50, feedrate is 300mm/min; the Y coordinate is not input, use current value Y50; keep F300, the modal command G01 can be omitted)

G0 X0 Y0； (Move to X0 Y0 at the rapid traverse rate, modal G command G0 valid)

M30；

Example 2

O0002；

G0 X50 Y5； (Move to X50 Y5 at the rapid traverse rate)

G04 X4； (Time delay for 4 seconds)

G04 X5； (Time delay again for 5 seconds non-modal command G04 should be input again）

M30；

Example 3□ □the first operation after the po□er is turned on) O0003；

G□0 G□4 G01 X100 Y100 F500; （G□4 feed per minute，feedrate is 500mm/min）

G□1 G□5 G01 X10 F0□01; （G□5 feed per revolution, input the F value again）

G□0 G00 X□0 Y50；

M30；

3.1.3 Related definition

The □ords or characters □hich are not specially described in this manual are as follo□s□

Start point: the position before performing the current block;

End point: the position after performing of the current block;

X: the end point absolute coordinate of X axis for G□0, the incremental value of X axis against current point for G□1;

Y: the absolute coordinate of Y axis at the end for G□0, the incremental value of Y axis against current point for G□1;

Z: the absolute coordinate of □ axis at the end for G□0, the incremental value of □ axis against current point for G□1;

F: □utting feedrate□

3.1.4 Address definition

□sage of the address in system is as follo□s□

Address	Function	Value range	Rounding
A	□unching number of 1 and 3rd side for rectangle serial punch□G140/G141)	-□□□□□□□□□□□□□□□□□□□ □bsolute value for negative	□ecimal part omitted
	4th，5th axis，axis name address	-□□□□□□□□□□□□□□□□□□□	□ound-off
B	□unching number of 2nd and 4th side for rectangle serial punch□G140/G141)	-□□□□□□□□□□□□□□□□□□□ □bsolute value for negative	□ecimal part omitted
	□adius for arc serially punch □G142/143)	-□□□□□□□□□□□□□□□□□□□	□ound-off
	4th，5th axis，axis name address	-□□□□□□□□□□□□□□□□□□□	□ound-off
C	□unching number for arc serially punch □G142/143)	-□□□□□□□□□□□□□□ □bsolute value for negative	□ecimal part omitted
	4th，5th axis，axis name address	-□□□□□□□□□□□□□□□□□□□	□ound-off
D	Tool radius offset number	0□32	□ecimal

			alarm
E	□nused		
F	G□4 feed per minute	0□15000	□ecimal efficiency
	G□5 feed per rotation	0□0001□500	□ound-off
	Tooth pitch in G74,G□4 □unit：G21, mm/r; G20 , inch/r)	0□001□500	□ound-off
□	G code	G command in system	□ecimal alarm
□	Length offset number	0□32	□ecimal alarm
	Operation command in G□5	0□□□	□ecimal alarm
I	□istance from arc start point to center point in X direction	-□□□□□□□□□□□□□□□□□□	□ound-off
	G110□G115□radius value of circle	-□□□□□□□□□□□□□□□□□□ □bsolute value for negative	□ound-off
	G134□G13□□□idth of rectangle in X direction	-□□□□□□□□□□□□□□□□□□ □bsolute value for negative	□ound-off
	G74,G□4：inch scre□（unit：tooth/inch）	0□0□□25400 □bsolute value for negative	□ound-off
□	□istance from arc start point to center point in Y direction	-□□□□□□□□□□□□□□□□□□	□ound-off
	G112,G113□ distance from start point to center point	-□□□□□□□□□□□□□□□□□□ □bsolute value for negative	□ound-off
	G114,G115□distance from start point to circle	-□□□□□□□□□□□□□□□□□□ □bsolute value for negative	□ound-off
	G134□G13□□□idth of rectangle in Y direction	-□□□□□□□□□□□□□□□□□□ □bsolute value for negative	□ound-off
	G140,G141□length of 2nd side of rectangle	-□□□□□□□□□□□□□□□□□□ □bsolute value for negative	□ound-off
□	□istance from arc start point to the center point in □ direction	-□□□□□□□□□□□□□□□□□□	□ound-off
	G110,G111,G134,G135□ cutting increment in XY plane each time	-□□□□□□□□□□□□□□□□□□ □bsolute value for negative	□ound-off

Volume I Programming

	G13□□G13□□ distance from start point to rectangle side in X axis direction	-□□□□□□□□□□□□□□□□□□ □bsolute value for negative	□ound-off
□	The length of linear chamfering	-□□□□□□□□□□□□□□□□□□ □bsolute value for negative	□ound-off
	□unching number for linear serial punch □use together □ith the canned cycle punch)	-□□□□□□□□□□□□□□□□□□ □bsolute value for negative	□ecimal part omitted
	Tool life management, tool life value	0□ □□□□□□	□ecimal part omitted
□	M miscellaneous function	0□□□	□ecimal alarm
	M code subprogram call	□000□□□□□	□ecimal alarm
□	□rogram number	0□2^{31}	□ecimal alarm
	Tool life□tool life unit □0-time, non-0 -time)	0 or other number	□ecimal alarm
□	□rogram number	0□□□□□	
P	□elay time in G04 □ms)	-□□□□□□□□□ □□□□□□□ □gnore negative	□ecimal alarm
	□ hat kind of number reference return in G30	2□4	□ecimal part omitted
	□kip se□uence or alarm number in G□5	0□□□□□	□ecimal alarm
	M□□ subprogram call □times□program name)	0□□□□□□□□□	□ecimal alarm
	□e□uence number of M□□ subprogram return	0□□□□□	□ecimal alarm
□	□pecifying G73 and G□3 cut-in value per time	-□□□□□□□□□□□□□□□□□□ □bsolute value for negative	□ound-off
	The value of operation in G□5	-□□□□□□□□□ □□□□□□□□□□	□ecimal alarm
R	□adius value of arc	-□□□□□□□□□□□□□□□□□□	□ound-off
	□ plane value of canned cycle command	-□□□□□□□□□□□□□□□□□□	□ound-off
	The value of operation in G□5	-□□□□□□□□□ □□□□□□□□□□	□ecimal alarm
S	□nalog spindle	0□□□□□	□ecimal alarm
	□hift spindle	0□□□	□ecimal alarm

T	□umber of tool	0□32□ parameter set value	□ecimal alarm
	Tool compensation number	0~32	Decimal alarm
U	Corner radius value of arc corner	-9999.999~9999.999 Absolute value for negative	Round-off
	Corner radius value of rectangle in G134~G139	-9999.999~9999.999 Absolute value for negative	Round-off
V	Distance to unmachined surface, in rapid cut of rough milling command G110,G111,G134 and G135	-9999.999~9999.999 Absolute value for negative	Round-off
W	First cutting-in value in Z direction in rough milling command G110,G111,G134 and G135	-9999.999~9999.999 Absolute value for negative	Round-off
X	Delay time in G04 (s)	-9999.999~9999.999 Absolute value for negative	Round-off
	X axis coordinate value	-9999.999~9999.999	Round-off
Y	Y axis coordinate value	-9999.999~9999.999	Round-off
Z	Z axis coordinate value	-9999.999~9999.999	Round-off

3.2 Rapid PositioningG00

Format: G00 X____ Y____ Z____;

Function: X, Y and Z axes simultaneously move to end points from start at their rapid traverse rates. See Fig. 3-1.

Two axes move at their respective speeds, the short axis arrives at the end firstly, the long axis moves the rest of distance independently, and their resultant paths are possibly not linear.

Explanation: G00, which is initial G command;

The value ranges of X, Y and Z are indicated as -9999.999~+9999.999mm;

X, Y and Z axes, one of them can be omitted or all of them can be omitted. When one of them is omitted, it means that the coordinate value of start and end points are same. The start and end points share the same position when they are omitted at the same time.

Command path figure:

Tool positions at the rapid traverse rate independently for each axis. Usually, the tool path is not linear.

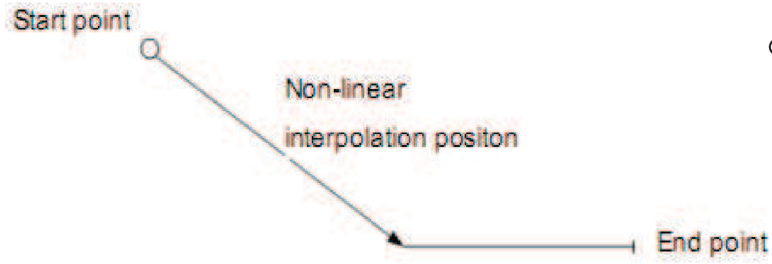


Fig. 3-1

X, Y and Z axes are separately set by the s□stem data parameter □o.059, □o.0□0 and □o.0□1 at their rapid traverse rate, the actual traverse rate can be modified b□ the rapid override □e□s on the machine panel.

The rapid traverse acceleration or deceleration time constant of X, Y and Z axes are separatel□ set b□ the s□stem data parameter □o.0□4, □o.0□5 and □o.0□□.

□xample□ tool traverses from point A to point □. See Fig.3-2.

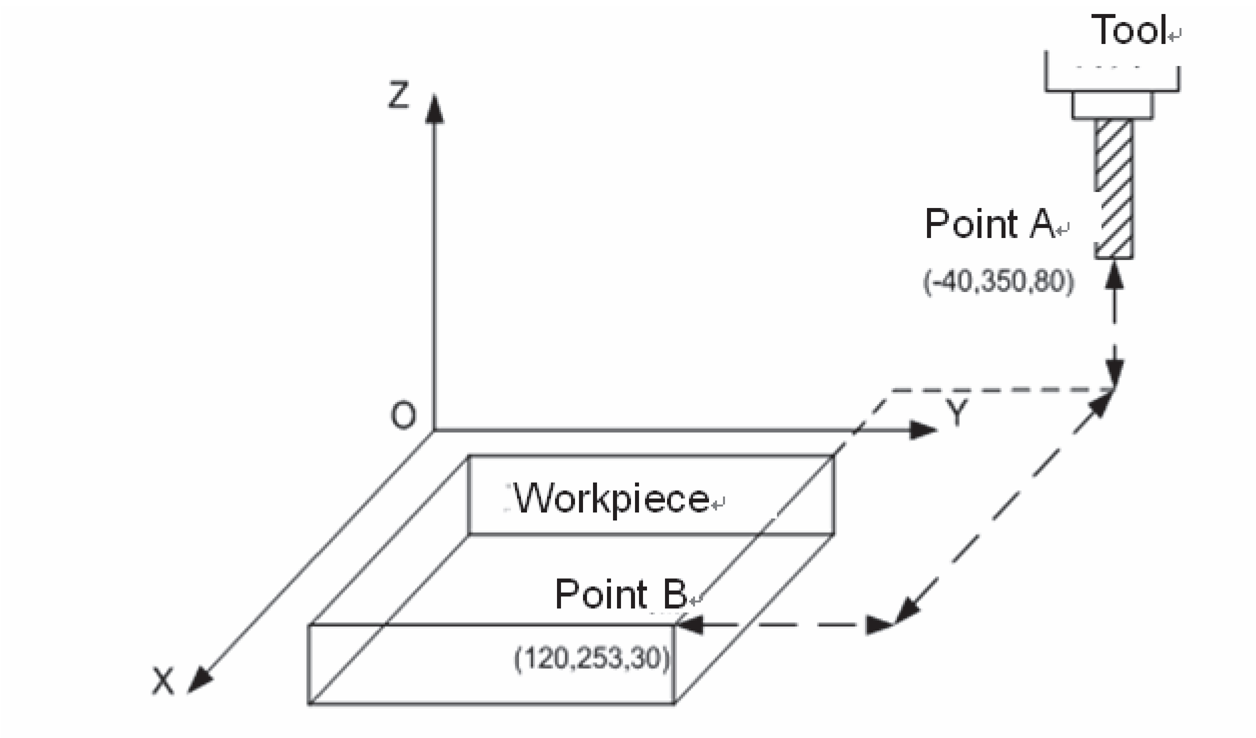


Fig. 3-2

G90 G0 X120 Y253 Z30; (absolute coordinate programming)

G91 G0 X1□0 Y-9□ Z-50; （relative coordinate programming）

3.3 □inear □nterpolation G01

Format: G01 X□Y□Z□F□;

Function: □ovement path is a straight line from start to end points.

Explanation: G01, which is modal G command;

The value range of X, Y and Z are indicated as -9999.999~+9999.999mm;

X, Y and Z axes which one of them can be omitted or all of them can be omitted.

When one of them

is omitted, it means that the coordinate value of start and end points are consistent. The start and end points share the same position when they are omitted at the same time.

F command value is vector resultant speed of instantaneous rates in X, Y and Z axes directions, the actual feedrate is the product of override and F command value;

F command value is invariable after it is performed till the new one is executed. The following G

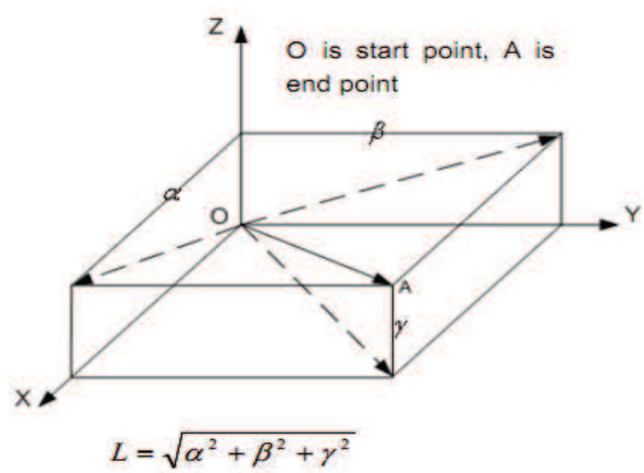
command with F command word uses the same function.

The value range is indicated as follows□

Command function	G94 (mm□min)	G95 (mm□rev)
□alue range	1~15000	0.001~500

Command path figure:

The linear interpolation is performed from point □ to point A□ G01 X α Y β Z γ F f__;



The feedrate specified by F is the tool movement speed along the line. The speed of each axis is as follows:

$$\text{Speed in X axis direction}: \quad F_X = \frac{\alpha}{L} \times f$$

$$\text{Speed in Y axis direction}: \quad F_Y = \frac{\beta}{L} \times f$$

$$\text{Speed in Z axis direction}: \quad F_Z = \frac{\gamma}{L} \times f$$

Note: The F initial default value is set by data parameter No.172 when the power is turned on.

□.□ □rc and □elical □nterpolation □ □□□□ □□

Fo□□ at:

□irc□lar interpolation:

□rc in the □□ plane:

$$G17\left\{\begin{matrix}G02\\G03\end{matrix}\right\}X\_\_\ Y\_\_\quad\left\{\begin{matrix}R\_\_\\I\_\_\ J\_\_\end{matrix}\right.\quad F\_\_$$

□rc in the □□ plane:

$$G18\left\{\begin{matrix}G02\\G03\end{matrix}\right\}X\_\_\ Z\_\_\quad\left\{\begin{matrix}R\_\_\\I\_\_\ K\_\_\end{matrix}\right.\quad F\_\_$$

□rc in the □□ plane:

$$G19\left\{\begin{matrix}G02\\G03\end{matrix}\right\}Y\_\_\ Z\_\_\quad\left\{\begin{matrix}R\_\_\\J\_\_\ K\_\_\end{matrix}\right.\quad F\_\_$$

□elical interpolation

□rc interpolation in □□ plane□□ axis linear interpolation lin□age□

$$G17\left\{\begin{matrix}G02\\G03\end{matrix}\right\}X\_\_\ Y\_\_\ Z\_\_\quad\left\{\begin{matrix}R\_\_\\I\_\_\ J\_\_\end{matrix}\right.\quad F\_\_$$

□rc interpolation in □□ plane□□ axis linear interpolation lin□age□

$$G18\left\{\begin{matrix}G02\\G03\end{matrix}\right\}X\_\_\ Z\_\_\ Y\_\_\quad\left\{\begin{matrix}R\_\_\\I\_\_K\_\_\end{matrix}\right.\quad F\_\_$$

□rc interpolation in □□ plane□□ axis linear interpolation lin□age□

$$G19\left\{\begin{matrix}G02\\G03\end{matrix}\right\}Y\_\_\ Z\_\_\ X\_\_\quad\left\{\begin{matrix}R\_\_\\J\_\_\ K\_\_\end{matrix}\right.\quad F\_\_$$

Fun□tion: □nly two axes of circ□lar interpolation can be lin□ed for controlling tool movement along with the arc on the selected plane in any time. □f the □rd axis is specified sim□ltaneo□sly in linear interpolation mode□ it will be lin□ed by linear interpolation type to constit□te helical interpolation. □□□ movement path is □□ from start to end points. □□□ movement path is □□□ from start to end points.

.

□□planation:

□□□ and □□□ are modal □ commands□

□ is arc radi□s□the val□e range are indicated as □□□□□.□□□～□□□□.□□□mm□

□ hen the circle center is specified by address □□□and □□they are corresponding with the □□□ and □ axes separately.

□is the difference between the center point and the arc start point in the □ axis direction□□□ center point coordinate □□ □ coordinate of arc start point□ the val□e range are indicated as □□□□□.□□□～□□□□.□□□mm□

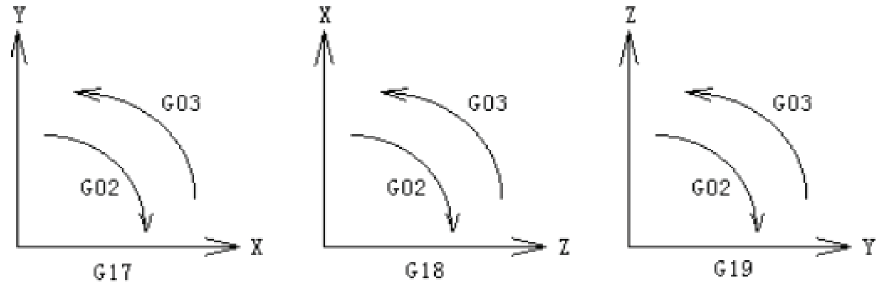
□ is the difference between the center point and the arc start point in the □ axis direction□□□center point coordinate □□ □ coordinate of circle arc start point□ the val□e range are indicated as □□□□□.□□□～□□□□.□□□mm□

□ is the difference between the center point and circle start point in the □ axis direction□ □□center point coordinate □□□ coordinate of circle start point□the val□e range are indicated as □□□□□.□□□～□□□□.□□□mm.

Note □hen □□□ and □ are for whole□ir□le that they have si□ns a□□ordin□ to the dire□tion. □nd they are positive values when □□□and □ share the same dire□tions with □□□ and □ a□es□otherwise they are ne□ative ones.

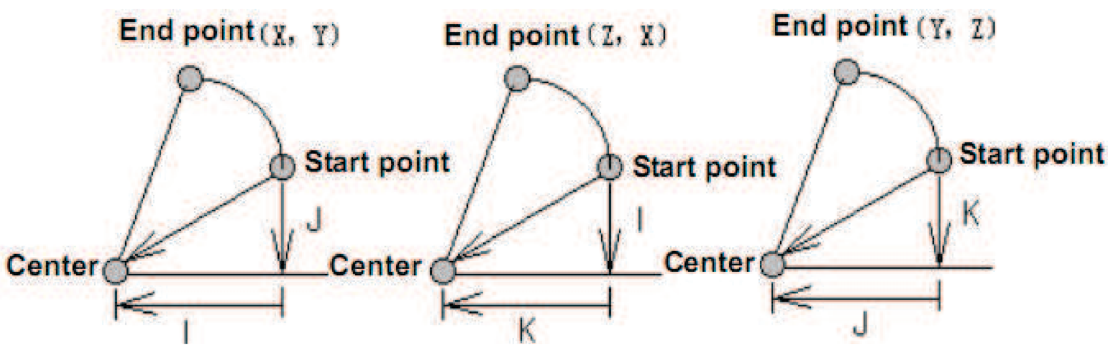
□tem	□pe□ified □ontent		□ommand	□eanin□
□	□lane specification		□□□	□pecifying □□ plane arc
			□□□	□pecifying □□ plane arc
			□□□	□pecifying □□ plane arc
□	□otating direction		□□□	□□
			□□□	□□□
□	□nd point	□□□ mode	Two axes of □□□ and □	□nd point in the part coordinate system
		□□□ mode	Two axes of □□□ and □	□istance from start to end points
□	□istance from start point to circle center point		□	□ axis distance from start point to the center point □with sign□
			□	□ axis distance from start point to the center point□with sign□
			□	□ axis distance from start point to the center point □with sign□
	□rc radi□s		□	□rc radi□s
□	Feedrate		F	Feedrate along the arc

□□loc□wise□ and □□o□nterclo□□wise□ are defined when □□ plane□□□ plane□ □□ plane□ is viewed in the positive□to□negative direction of the □ axis □□ axis□□ axis□in the □artesian coordinate system□see the following fig□re:



CW or CCW

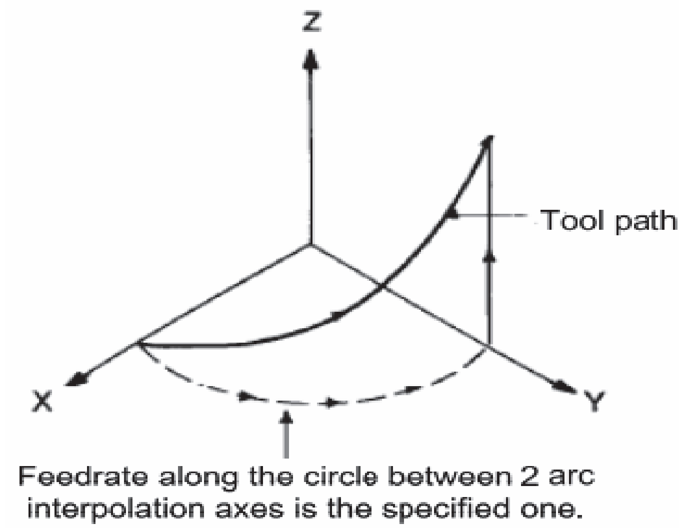
The end point of an arc is specified by □sing the address □□ □ or □□ and is expressed as an absol□te or incremental val□e according to □□□ or □□□. The incremental val□e is the distance val□e from start to end points of an arc. The arc center is specified by address □□ □ and □ against the □□ □ and □ respectively. The n□merical val□e following □□ □ and □□ however□ is a vector component from start point of an arc to the center point□ which is an incremental val□e with sign. □ee the following fig□re:



The F command is circ□lar interpolation rate in helical interpolation□ in order to achieve the lin□age interpolation between linear axis and arc□ the speed of linear interpolation by the □rd axis has the following relationship to the F command:

$$f = F \times \frac{\text{Length of linear axis}}{\text{Length of circular arc}}$$

□elical interpolation path is as follows:



□□□ and □ have signs according to the direction. The circ□lar center also can be specified by radi□s □ other than □□□ and □□ as follows:

□ □□ □□ □□ □ □ □

□ □□ □□ □□ □ □ □

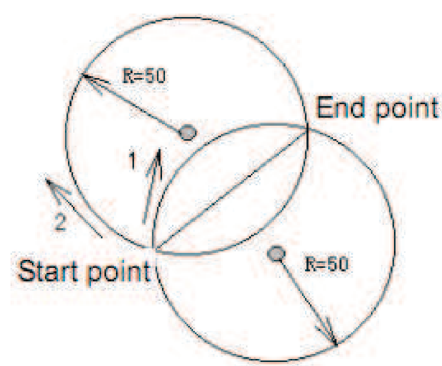
□ow□the following two arcs can be described□one arc is more than □□□□□the other is less than □□□□. The arc radi□s which is less than □□□□ is specified by the positive val□e□the arc radi□s which is more than □□□□ is specified by the negative val□e. The radi□s is either positive or negative when the arc command is e□□al to □□□□.

□□xample□□rc ① less than □□□□

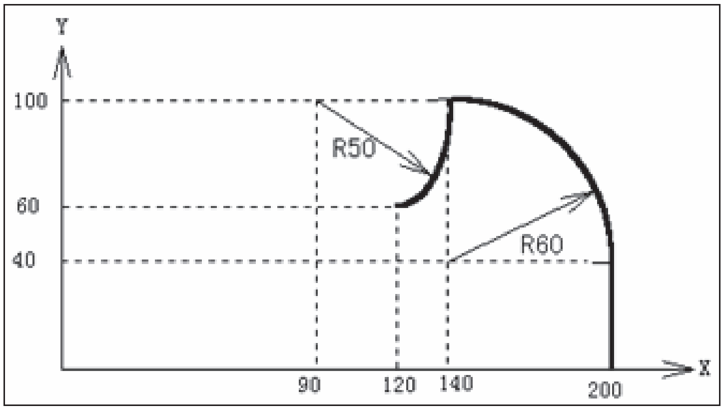
□ □□ □ □□ □□□.□ □□□.□ □ □□.□ F□□□.□□

□rc ② more than □□□□

□ □□ □ □□ □□□.□ □□□.□ □ □□□.□ F□□□.□□



□□xample for the programming□



To program the above paths □sing the absol□te mode and incremental mode respectively:

□□□□bsol□te mode

□ □□ □□□□.□ □□□.□ □□ ;

□ □□ □ □□ □□□□.□ □□□□.□ □□□.□ F□□□.□ ;

□ □□ □□□□.□ □□□.□ □□□.□ ;

□r □ □□ □□□□.□ □□□.□ □□ ;

□ □□ □ □□ □□□□.□ □□□□.□ □ □□.□ F□□□.□ ;

□ □□ □□□□.□ □□□.□ □ □□.□ ;

□□□□ncremental mode

□ □□ □ □□ □□□.□ □ □□.□ □□□.□ F□□□.□ ;

□ □□ □□□.□ □□□.□ □□□.□ ;

□r □ □□ □ □□ □□□.□ □□□.□ □ □□.□ F□□□.□ ;

□ □□ □□□.□ □□□□.□ □ □□.□ ;

The feedrate of circ□lar interpolation is specified by F command□it is the speed of the tool along the arc tangent direction.

Note 1: □□□□ and □□ □an be omitted□but□it is very ne□essary to input one of the addresses □□□□ □ or □□or the system alarm is □enerated.

Note 2: The □□□ and □ □an be omitted simultaneously when the end and start points share same position. □ hen the □enter point is spe□ified by address □□□ and □□it is a □□□□ar□.

□ □2 □□□ □Full □ir□le□

The □ir□le is □□when usin□ □.

□ □2 □□□ □not move□

□t is re□ommended that pro□rammin□ uses □. □n order to □uarantee the start and end points of the ar□ are □onsistent with the spe□ified value□the system will move by □ountin□ □ a□ain a□□ordin□ to the sele□ted plane□when pro□rammin□ usin□ the □□□ and □.

□lane sele□tion	□ount the radius □ value a□ain
□□□	$R=\sqrt{I^2+J^2}$
□□□	$R=\sqrt{I^2+K^2}$
□□□	$R=\sqrt{J^2+K^2}$

Note □: The error between the a□tual tool feedrate and the spe□ified feedrate is □2□ or less. The □ommand speed is movement speed after tool radius offset alon□ the ar□.

Note □: The □ is effe□tive when address □□□ and □ are □ommanded with the □□but the □□□ and □ are disabled at one time.

Note □: The a□is not e□ists is spe□ified on the set plane□the alarm o□□urs.

Note □: □f the radius differen□e between start and end points e□□eeds the permitted value by parameter □No.1□□□□a □□ alarm o□□urs.

□.□ □well □□□

Format: □□□ □□ □or

□□□ □□ □

Fun□tion: □□es stop□the □urrent □ □ommand mode and the data□status are invariable□after delayin□ time spe□ified□the ne□t blo□□ will be e□e□uted.

□□**planation:** □□□□which is a non□modal □□command□

□□□ delay time is specified by command words □□□□□□

□ee the following fig□re table for time □nit of □□ and □□ command val□e:

□ddress	□	□
□nit	□.□□□ s	s
□vailable □n	□～9999999	0~9999.999

Note:

- **X can be specified by the decimal but P not, or the alarm will be generated.**
- **When the P and X are not introduced or they are negative value, it means exact stop between the**
- **The P is effective when the P and X are in the same block.**
- **The operation is held on when feeding during the G04 execution. Only the delay time execution is finished, can the dwell be done.**

□olume □ □ro□rammin□

[illegible] 9

Format:

G[illegible] [illegible] X[illegible]

G[illegible] [illegible] X [illegible]

G[illegible] [illegible]

Function: [illegible]

Explanation: [illegible] 9 [illegible].

Command example:

[illegible] ; [illegible]

[illegible] ; [illegible]

Note:

Note [illegible]: The plane selection command can share the same block with other group G commands.

Note [illegible]: The move command is regardless of the plane selection. [illegible]or example, the [illegible] axis is not On X[illegible] plane, the [illegible] axis movement is regardless of the X[illegible] plane in command G[illegible] [illegible].

G[illegible] ;

[illegible] 0 [illegible]

Format:

[illegible] 0 [illegible];

Function: [illegible].

Explanation:

[illegible]	[illegible]	[illegible]. [illegible]
[illegible]	[illegible] 0	0.000 [illegible]
[illegible]	[illegible]	0.00 [illegible]

[illegible]. [illegible].

[illegible].

[illegible].

[illegible].

[illegible].

[illegible].

[illegible].

[illegible].

Note [illegible]: The G code for inch or metric conversion when the power is turned on is the same as that at the power off.

Note [illegible]: [illegible]hanging G[illegible]0 and G[illegible] are unallowed during programming. Or the alarm occurs.

Note [illegible]: When the unit systems between the machine and input are different, the max. error is 0.[illegible] of the min. move unit[illegible] and the error is not be cumulated.

Note 4: [illegible]s the inch input [illegible]G[illegible]0[illegible] and the metric input [illegible]G[illegible] switches each other, the offset should be suited to the reset of the input unit.

[illegible]

[illegible]ormat: [illegible]

[illegible]unction: [illegible]

[illegible]xplanation: [illegible]

[illegible] 90 [illegible] 9 [illegible]

[illegible] 90 [illegible] 9 [illegible]

[illegible] 90 [illegible] 9[illegible].

[illegible]

[illegible]ommand	[illegible]un
[illegible]	[illegible].
[illegible]	[illegible]
[illegible]	[illegible]
[illegible]	[illegible]
[illegible]	[illegible]
[illegible]	[illegible]
[illegible]	[illegible]
[illegible]	[illegible]

Process for command action [illegible]0[illegible]:

[illegible]

[illegible]

[illegible].

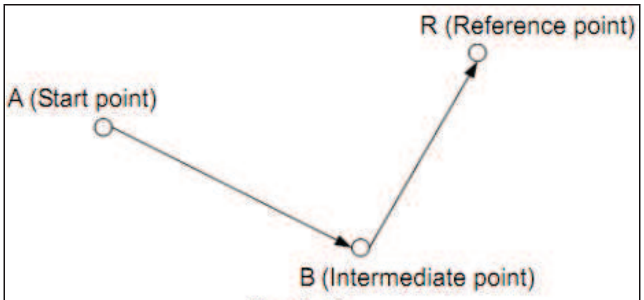


Fig. 3-10

[illegible]olume [illegible] Programming

Note:

- [illegible]
- [illegible] → [illegible] → [illegible]
- [illegible]
- [illegible]
- [illegible]
- [illegible]
- [illegible]

[illegible].9 [illegible] 9

[illegible]ormat: [illegible]9 [illegible]

[illegible]unction: [illegible]

[illegible]xplanation:

[illegible] 90 [illegible] 9 [illegible]

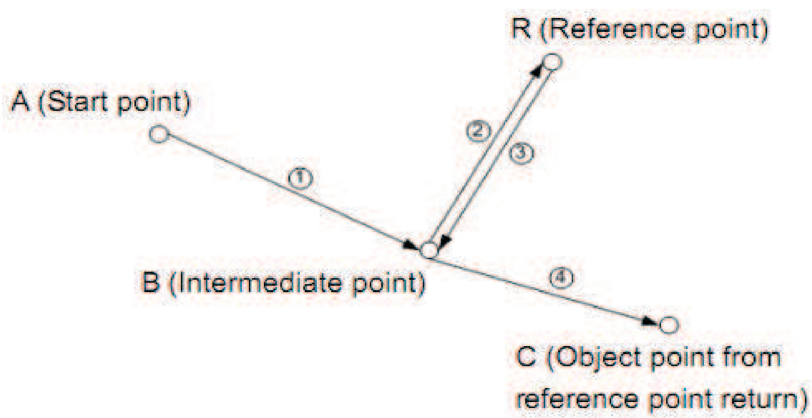
[illegible] 90 [illegible] 9 [illegible]

[illegible] 90 [illegible] 9 [illegible]

[illegible]

□ommand	□uncti
□□9	[illegible]
□□9 □	[illegible]
□□9 □	[illegible]
□□9 □	[illegible]
□□9 □_ □	[illegible]
□□9 □__ □	[illegible]
□□9 □___□	[illegible]
□□9 □___□__□___	[illegible]

Process for command action:



[illegible] ①→②.

[illegible] ③→④.

Note:

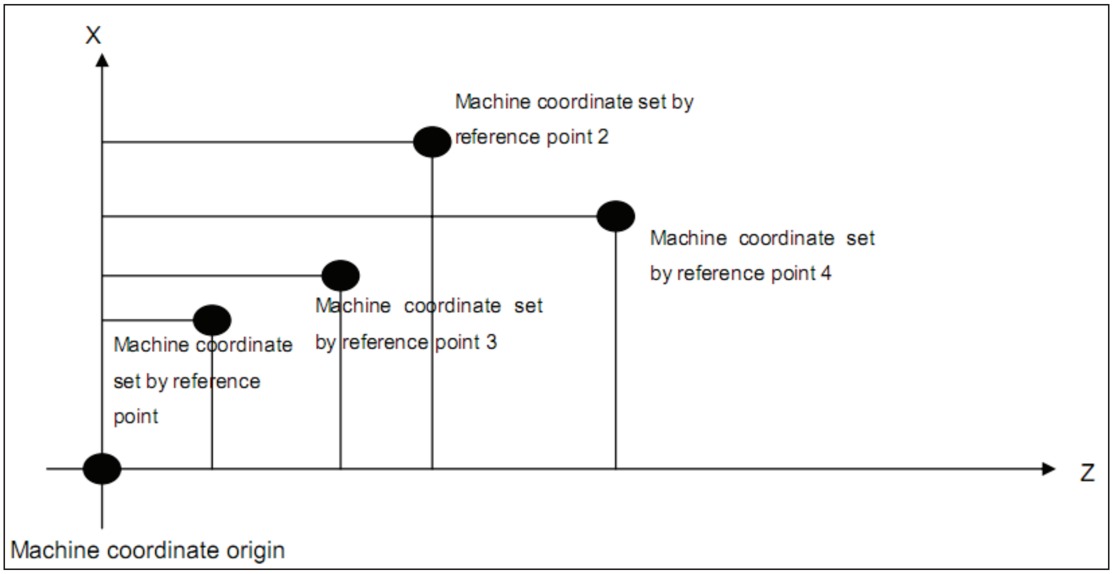
Note □:G□□ is specified after G□□, if an intermediate point is not specified by any of axes, the system alarm will be generated.

Note □: □t is incremental distance against the intermediate point in G□□ coordinate programming.

Note □: □urrent position is reference point when the G□□ command is followed to G□□ or G□0, it returns from reference point directly□or, it returns from current position if G□□ command is not followed by G□□ or G□0.

□.□0 [illegible] □□0

[illegible]~[illegible].



□ormat:

□□0 □□ □□ □□ □□ □□□□ □ □□□□□ □□□ □□□□□□□ □□□□□□□□ □□ □□□ □□ □□ □□□□□

□□0 □□ □□ □□ □□ □□□□ □ □□□□□ □□□ □□□□□□□ □□□□□□□□

□□0 □□ □□ □□ □□ □□□□ □ □□□□□ □□□ □□□□□□□ □□□□□□□□

□unction: □□□ □□□ □□□□□□□□□□□□□ □□□□ □□□□ □□□□□□ □□□ □□□ □ □□ □□□□□□ □□□ □□□□ □□□□□□ □□□□□□ □ □□□□□ □$^{□□}$□□$^{□□}$ □□□ □$^{□□}$ □□□□□□□ □□□□□ □□□ □□□□□□. □□□ □□□ □ □□□ □□□□ □□ □□□ □□ □□ □□□□ □□□□ □□□ □ □□□□□ □$^{□□}$ □□□□□□□ □□□□ □□ □□□□□□.

□xplanation: □□0□□ □□□ □□ □ □□□□ □□□□□ □□□□ □ □□□□

□□ □ □□□ □□□□□□□□ □□□□□□ □□□□□ □□□□

□□ □ □□□ □□□□□□□□ □□□□□□ □□□□□ □□□□

□□ □ □□□ □□□□□□□□ □□□□□□ □□□□□ □□□□

□□□ □□□□ □□□ □ □□□ □□□□□□ □□□ □□□ □ □□□□□□□□□ □□□ □□ □□ □□□□□□□ □□□ □□□□ □□□ □□□□□

□ommand	□unction
□□0 □□ □	□ □□□□□ □$^{□□}$ □□□□□□□ □□□□□□□□□ □□□□ □□□□□ □□□ □ □□□□ □□ □□ □□□□□□□□□□□□
□□0 □□ □□ □□	□ □□□□□ □$^{□□}$ □□□□□□□ □□□□□□□□□ □□□□ □□□ □ □□□□□ □□□□ □□ □□ □□□□□□□□□□□□
□□0	□ □□□□ □□ □□□ □□□□□□□ □□□□□□□ □□□ □□□□ □□□□□
□□0 □□ □□ □□ □□	□□ □ □□□ □ □□□□ □□□□□ □□ □□□ □ □□□□□ □$^{□□}$ □□□□□□□□

Note □:n is □, □or 4 in above table□

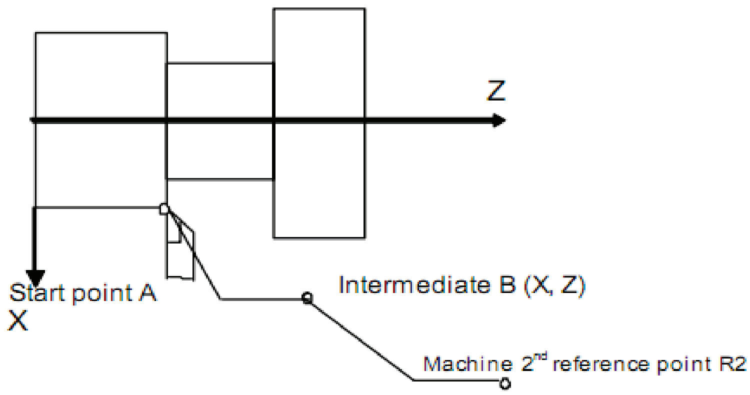
Note □: □eceleration and □ero signals check are not needed when the machine □nd, □rd and 4threference points are returned to.

□ommand action process [illegible]

[illegible]

[illegible]

[illegible]



Note □: □fter returning the machine reference point by manual or the G□□ command is performed, the machine □nd, □rd and 4th reference point return function can be employed only, or the □nd, □rd and 4th reference point operation of G□0 command , the system alarm will be generated.

Note □: □rom point □ to □ or from point □ to □□, the □ axes are moved at their separately rate, so the path is not straight line possibly.

Note □: □fter machine □nd, □rd and 4th reference point returned by the G□0 command, the system tool length compensation cancellation is defined by bit □ of the parameter No.□□.

Note 4: The □nd, □rd and 4th reference point operation of G□0 command can not be executed if the □ero switch is not installed on the machine tool.

Note □: The workpiece coordinate system is set after the machine □nd, □rd and 4th reference point are returned.

[illegible]

[illegible]

□ormat:

[illegible]

□xplanation:

[illegible]

[illegible]

□□ □□□□□□□. □□ □□□ □□□□□□□□ □ □□□□□ □□ □□□ □□□□□□□□ □ □□ □□□□□□ □□□□ □□□□ □□ □□ □□□□□ □□□□□□.

□. □□.

□ignal： □□□□□□□□□□□□□□□□□□□□□□□□□□□□□□.0 □□□0□9□.

Parameter：

0	□	□								□□P□	G□□P

□□□P □□ □□□□□□□□□□□□□□□□□

0□ □□□□□□□□□□□□□□.

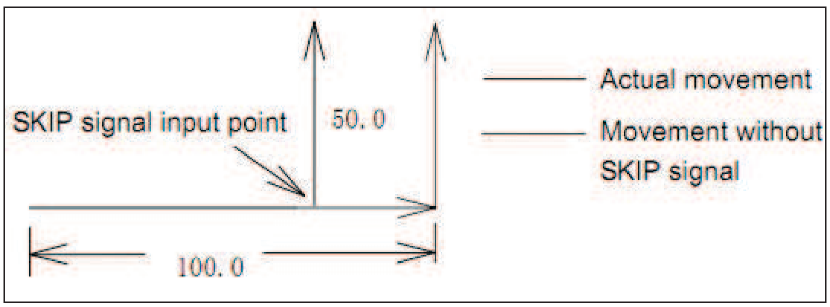
G□□P □□ □□□□□□□□□□□□□□□□□□□□□□□□□□□□□□□

0□ □□□□□□□□□□□□□□□□□□□□□□□□□□□□□□.

□. **The next block to G□□ is incremental command** □□ □ □□□□ □□ □□□□□□□ □□□ □□ □□ □□□□□□□□□□□□□□□□□□□.

□□□□□□ □□□□9□ □□00.0 □□00 ;

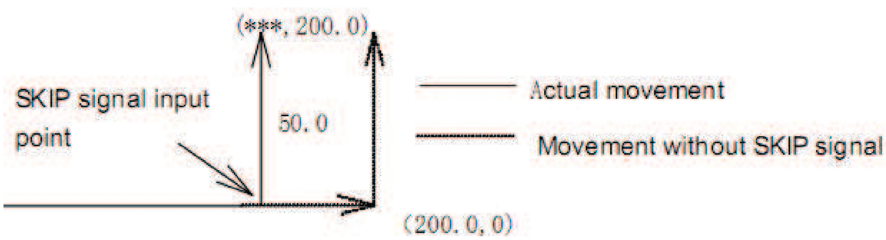
□□0.0 ;



□. **The next block to G□□ is absolute command for one axis:** □□□ □□□□□ □□ □□□□□ □ □□ □□□□□□□□□□□□□□□□□□□□□□□□□□□□□□□□□□□□□□□.

□□□□□□ □□□□90 □□00.0 □□00 ;

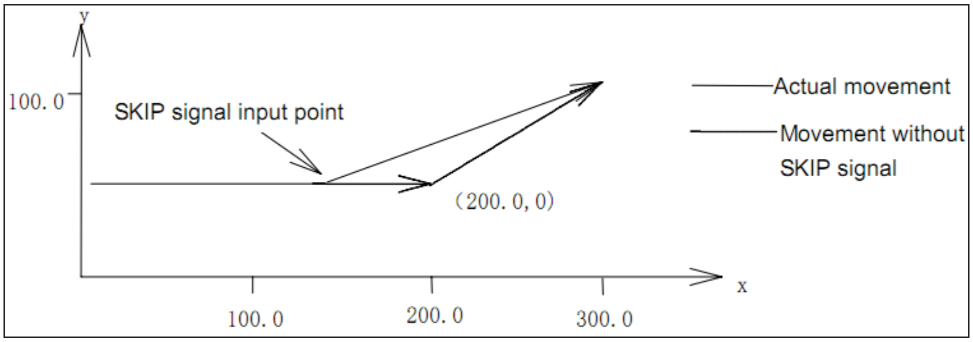
□□00.0 ;



□. **The next block to G□□ is absolute command for □axes:** □ □□.

□□□□□□ □□□□90 □□00.0 □□00 ;

□□00.0 □□00.0 ;



[illegible] 0 [illegible]

[illegible]ormat:

[illegible] [illegible] G42 [illegible]

[illegible]unctions:

[illegible]

[illegible] 0 [illegible] 00 [illegible] 0 [illegible] 0 [illegible] 0 [illegible].

G codes	[illegible]unctions
[illegible]0	[illegible]
[illegible]	[illegible]
[illegible]	[illegible]

[illegible] 0 [illegible].

[illegible]xplanation:

- [illegible]

[illegible].

Plane selection	Plane compensation
[illegible]	[illegible]—[illegible]
[illegible]	[illegible]—[illegible]
[illegible]9	[illegible]—[illegible]

- [illegible]

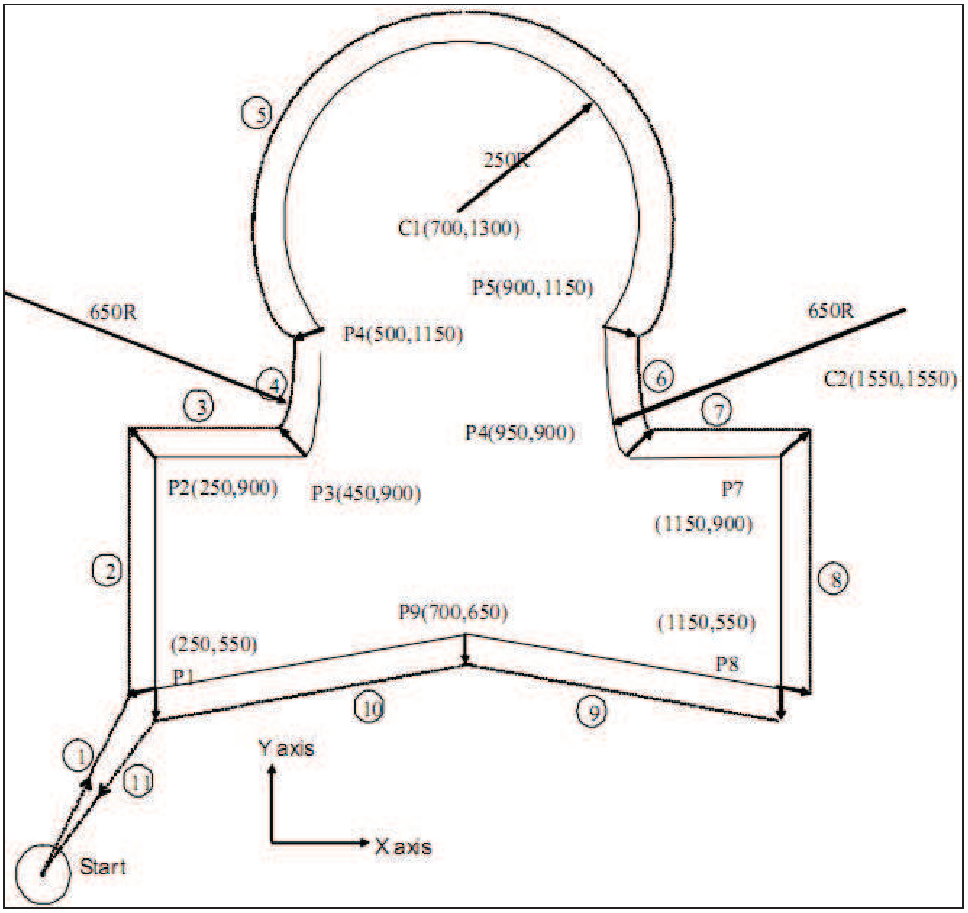
[illegible].

[illegible].00 [illegible].

olume Programming

[illegible]	0～+9999.999mm	0～+999.999 inch

Note:



Example :

□□c□ □□□i□n□m□□ □□□□□h□ c□m□□n□□i□n c□nc□□□i□n m□□□ □□c□m□□ c□m□□n□□i□n m□□□ □□ □□□ in □hi□ □□□c□. □□□h□ □n□ □□hi□ □□□c□□□□□c□n□□□i□ c□m□□n□□□□ in □h□ □i□c□i□n □h□□□□□ □□□i□□ i□ □□□ic□□□ n□□□□□□□□□m □□□h □□□m □□ □□ □□□. □□□□c□m□□n□□i□n □□□□□ i□ □□□ci□i□ □i□h □0□□ □□ □□□ □h□ c□m□□n□□i□n n□m□□□ □□ □□ □h□n □h□ □□□i□ in□ic□□□□ □i□h □□□□□□□h c□m□□n□□i□n □□□.

After the compensation begins, tool path compensation performs automatically when creating the workpiece as P1→P2……P8→P9→P1.

□00 □9□ □0 □0 □0;

□0□ □90 □□□ □00 □□□ □□ □□□0.0 □□□0.0 ;　（□h□ c□m□□n□□i□n □□□□ □h□□□ □□ □□□□□□ □i□h c□m□□n□□i□n n□m□□□）

□0□ □0□ □900.0 □□□0 ;

□0□ □□□0.0 ;

□0□ □0□ □□00.0 □□□□0.0 □□□0.0 ;

□0□ □0□ □900.0 □□□□0.0 ;

□0□ □0□ □9□0.0 □900.0 □□□0.0 ;

□0□ □0□ □□□□0.0 ;

□0□ □□□0.0 ;

□09 □□00.0 □□□0.0 ;

□□0 □□□0.0 □□□0.0 ;

□□□ □00 □□0 □0 □0 ;

[illegible] Tool length compensation G43, G44, G49

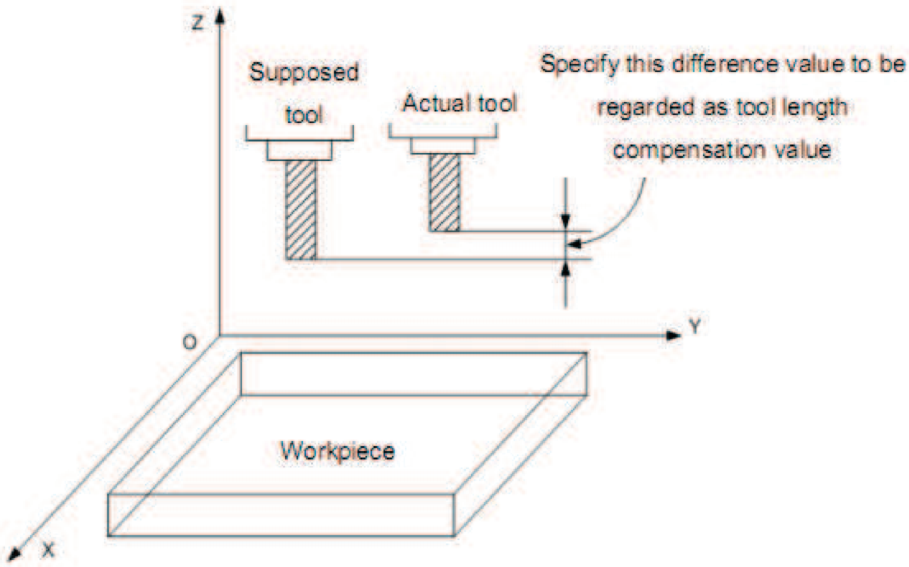
Function:

G17
G18
G19
G43
G44
H__

Tool length compensation function.

Explanation:

G43 and G44 [illegible] can [illegible] the [illegible] specified [illegible] machining [illegible] in the same [illegible].



The end point specified by a axis moves in [illegible] [illegible] by [illegible] is [illegible]. Difference between supposed and actual machine tool length is [illegible] the offset value when the program is compiled. Different length tools can be employed by changing tool length compensation without the program changed is [illegible].

Different offset directions are specified by G43 and G44, and the offset number is specified by H code.

Offset axis

The [illegible] specified by the specifying plane of G17, G18 and G19.

Specifying plane	Offset axis
G17	Z axis
G18	Y axis
G19	X axis

[illegible] [illegible] can be used to specify the offset axis and the offset axis changed by [illegible].

[illegible] and [illegible] compensation

G19 [illegible] axis [illegible]

[illegible] axis [illegible] compared with the [illegible] and [illegible] compensation.

Offset direction

G43: Positive offset

G44: Negative offset

Compensation value can be specified by H code. Either absolute or incremental command, the end point coordinate value specified by Z axis move command in program plus the offset value specified by H code in G43, and the offset value specified by H code is subtracted from the end point coordinate value specified by H code in G44. Then the value calculated is regarded as the end point coordinates.

The following command is indicated when Z axis move is omitting, when the offset is positive, it is shown offset in the positive direction; or it is shown offset in the negative direction.

It is the same when the offset is negative value.

Specifying the offset

An offset number is specified by H code and its corresponding offset value shall be subtracted from Z axis movement command value in program or added onto Z axis movement command value. The offset number is H00~H255.

The value corresponding with offset number is set in the offset page by using the panel MDI/LCD. Setting range of value is as follows:

	Millimeter input (mm)	Inch input (inch)
Offset	-9999.999～+9999.999	-999.9999～+999.9999

The offset number H00 i.e. H00 corresponds to the 0 offset. It is disabled to set offset value of H00.

Tool length compensation cancellation

G49 or H00 can be specified when the tool length compensation is cancelled. When the axis movement compensations are cancelled, then the axis compensation will be cancelled if the G49 is specified. Compensation value of the specified axis is cancelled when it is cancelled with H00. After G49 or H00 is specified, the system immediately cancels the compensation value.

Note:

1. In the block the tool length compensation is specified (G10, G40/G50/G49) and G28 cannot be specified at the same time, otherwise an alarm will occur.

2. Tool length compensation command can be specified in the block in which canned cycle is specified. Meanwhile the canned cycle is executed after the tool length compensation is disabled and is not modal.

Example:

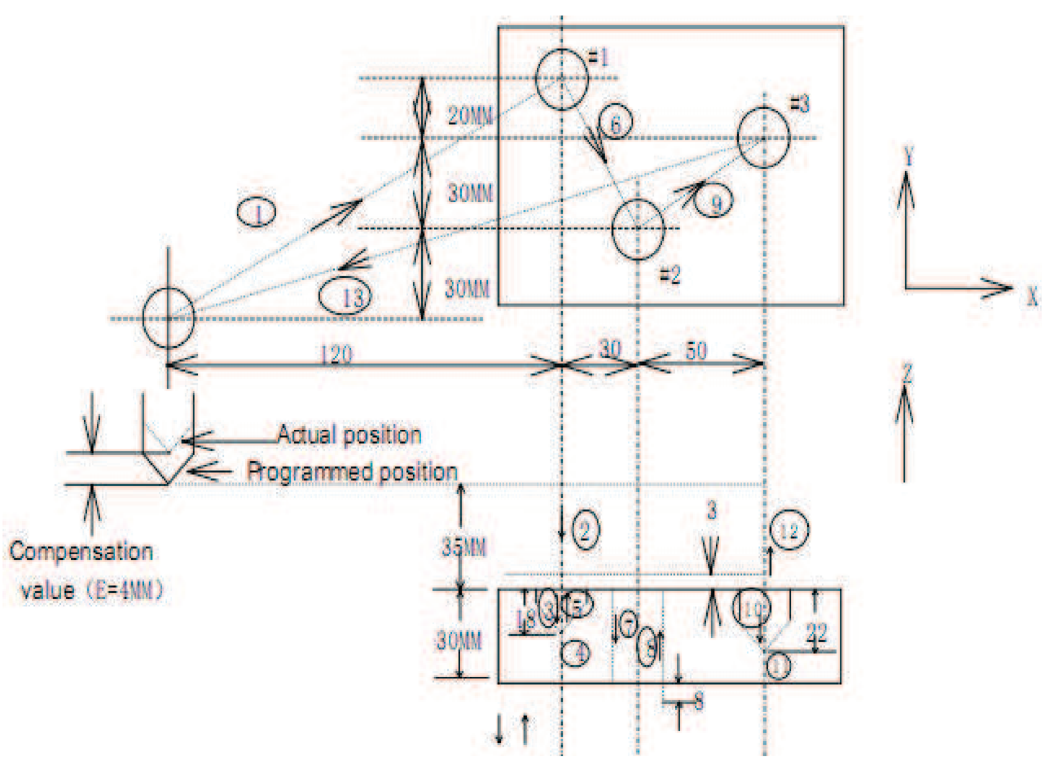
Command	Offset	Explanation (H1=20.0mm, H2=30.0mm)
G43 H1 G44 G01 X10 Y10 Z10 H2 G90 G00 X100 Y100 Z100 H00	[illegible] [illegible] [illegible]	Setting Z axis length compensation in the positive direction. Single incremental positioning, setting H2 length compensation in negative direction Positioning to X100 Y100 Z100(Z70) with the compensation cancelled.

□n □h□ □□m□ □□□c□ □i□h □0□□□0□□□0□□□ □□□□9□		
□□□□□ □□9 □0□ □□0 □□□□□	□□□□□ □□□□□	□□□in□ □□ □□□□□n□h c□m□□n□□i□n in □h□ □□□i□i□□ □i□□c□i□n. □□□m □cc□□□.
□n □h□ □□m□ □□□c□ □i□h c□nn□□ c□c□□ c□□□□		
□□□□□ □□□ □□□ □□0 □□ □□□0 □□ □90 □00 □□00 □□00 □□00	□□□□□ □□□□□ □□□□□	□□□in□ □□ □□□□□n□h c□m□□n□□i□n in □h□ □□□i□i□□ □i□□c□i□n. □□□in□ □□ □□□□□n□h c□m□□n□□i□n in □h□ n□□□□i□□ □i□□c□i□n. □□□□□ □h□ c□nn□□ c□c□□ □□m □□.

□□□ci□i□□ in □h□ c□nn□□ c□c□□		
□□□□□ □90 □□□ □□0 □□□ □□0 □□9 □□ □□9 □0 □□□ □□□ □□□ □0	□□□□□ □□□□□ □□□□□ □□9 □0	□□□in□ □□ □□□□ □n□h c□m□□n□□i□n in □h□ □□□i□i□□ □i□□c□i□n. □□m□□n□□i□n □□□□□□i□h □□□□n□□□ in□□ c□nn□□ c□c□□ m□□□. □h□ □□□□□n□h c□m□□n□□i□n □□ □□9□□□in □h□ c□nn□□ c□c□□ i□ in□□□ci□□□ □n□ □h□ □□□i□□□□ □□□c□ □□m□in□ m□□□□. □□nc□□□□□h□ □□i□ c□m□□n□□i□n□□ □n□ □□□□0 m□□□□. □□□i□i□n □□ □□□□ □□□□ □□□□□□□.

□omman□ Example:

□□□□□n□h c□m□□n□□i□n □□□□□□ □n□ □□h□□m□chinin□□



□□□□□□ □0□□ □.0

□□ □9□ □00 □□□0.0 □□0.0 ; .....□ . (1)

[illegible].0 [illegible]0[illegible] ; .... [illegible] (2)
N3 G01 Z-21.0 ; .... (3)
N4 G04 P2000 ; .... (4)
N5 G00 Z21.0 ; .... (5)
N6 X30.0 Y-50.0 ; .... (6)
N7 G01 Z-41.0 ; .... (7)
N8 G00 Z41.0 ; .... (8)
N9 X50.0 Y30.0 ; .... (9)
N10 G01 Z-25.0 ; .... (10)
N11 G04 P2000 ; .... (11)
N12 G00 Z57.0 H00 ; .... (12)
N13 X-200.0 Y-60.0 ; .... (13)
N14 M30 ;

Z, X or Y axis offsets a value at offset storage positively or negatively from the original end position according to the above command. Offset axes can be specified with G17, G18 and G19, offset direction can be specified with G43 and G44. Offset No. corresponding to the offset is specified by H code.

3.14 Workpiece Coordinate system G54~G59

Format:

G54 X__ Y__ Z__; Workpiece coordinate system 1
G55 X__ Y__ Z__; Workpiece coordinate system 2
G56 X__ Y__ Z__; Workpiece coordinate system 3
G57 X__ Y__ Z__; Workpiece coordinate system 4
G58 X__ Y__ Z__; Workpiece coordinate system 5
G59 X__ Y__ Z__; Workpiece coordinate system 6

Function:

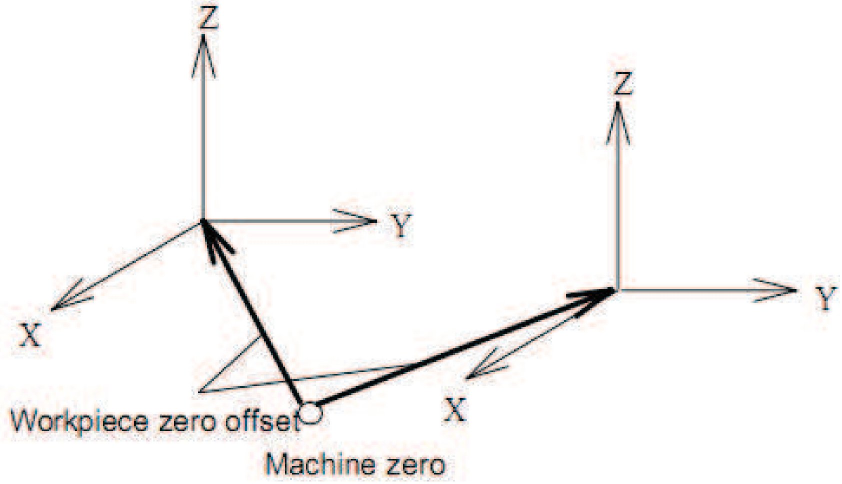
There are 6 workpiece coordinate systems for machine tool regardless of the G92, any of coordinate system can be selected by G54~G59.

Explanation:

X: New X axis absolute coordinate in current position;
Y: New Y axis absolute coordinate in current position;
Z: New Z axis absolute coordinate in current position.

These six workpiece coordinates are set by the distances (workpiece zero offset) from machine zero to each coordinate system origin.

Volume I Programming

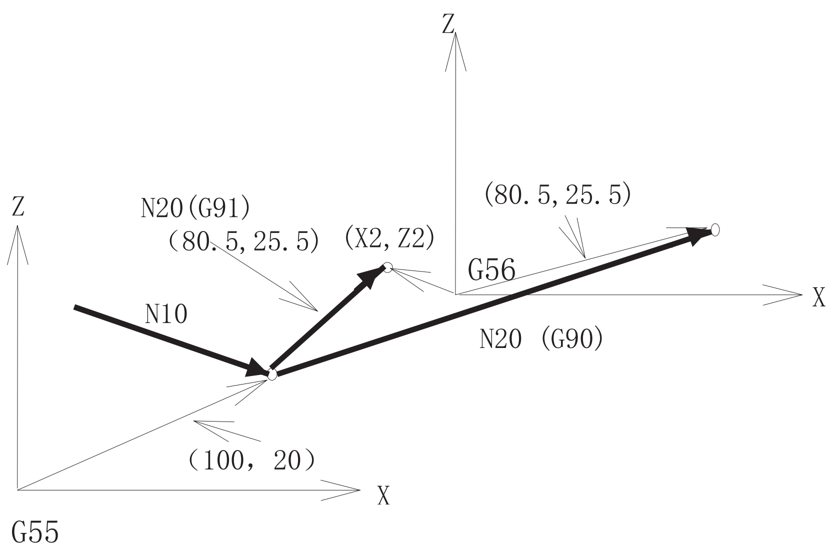


Exampl□□:

N10 G55 G90 G00 X100.0 Z20.0;

N20 G56 X80.5 Z25.5;

□apidly positioning to workpiece coordinate system 3 (X□80.5, Z□25.5) from workpiece coordinate s□stem □ (X□100.0, Z□□0.0). □or example, if N20 block is G91, it is incremental movement. The absolute coordinates automatically become the coordinates in coordinate system G56.



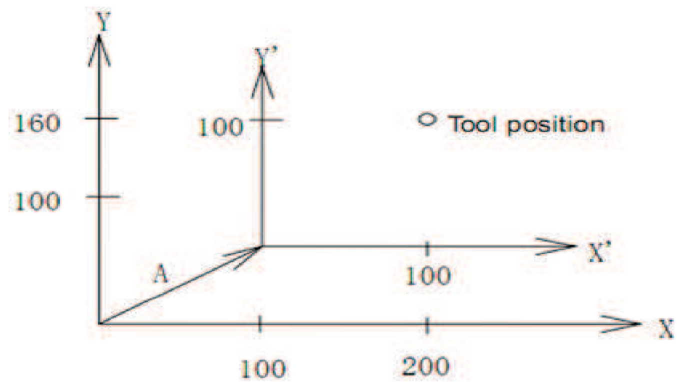
The absolute position for the figure is coordinate value under the current coordinate system.

□ot□:

- Workpiece coordinate systems 1~6 is set up as soon as machine zero return is executed after power-on. When the system is restarted, the coordinate system is the one set by parameter No. 13 bit 17.
- Whether the relative position varies with coordinate system depends on status parameter №005 PP□. when PP□□0, it changes; when PP□□1, it does not change.
- When the workpiece coordinate system function is determined, usually, G92 is not

needed to set coordinate system. if G92 is used, coordinate system 1~6 will be moved. □o not confuse with □9□ and □54~□59, unless workpiece coordinate systems G54~G59 are to be moved. When G54~G59 are in the same block with G92, G54~G59 are disabled.

- Workpiece coordinate system can be modified in the program run. The new coordinate system is effective till the system is restarted.



□f it performs □9□ X100 Y100 commands when the tool is positioned a（t □00，160）in the G54 coordinate system; the offset vector □ for workpiece coordinate system 1 is (X□,Y□). □nd the other workpiece coordinate systems offset for vector □.

3.15 Compound Cycle Command

□□□□□ □ri□□□or cann□□ c□cl□

Generally, the canned cycle is a machining movement completion from one block with G function to the completion of multi-block specified. Canned cycles make it easier for the programmer to create programs. With a canned cycle, a fre□uently-used machining operation can be specified in a single block with a G function; without canned cycles, multiple blocks are needed, and canned cycles can shorten the program to save memory.

□□□□□□ □ann□□ c□cl□ li□t

□ co□□□	□rillin□	□p□ration at t□□ □ottom o□a □ol□	□□traction	□pplication
G73	□ntermittent feed	——	□apid feed	High-speed peck drilling cycle
G74	□eed	□well, spindle CCW	□eed	□eft-hand tapping cycle
G80	——	——	——	Canned cycle cancellation
G81	□eed	——	□apid feed	□rilling, point drilling
G82	□eed	□well	□apid feed	□rilling, boring, counter boring
G83	□ntermittent feed	——	□apid feed	Peck drilling cycle
G84	□eed	□well, spindle CW	□eed	Tapping
G85	□eed	——	□eed	□oring
G86	□eed	□pindle stop	□apid feed	□oring
G88	□eed	□well, spindle stop	manual	□oring

G89	□eed	□well	□eed	□oring
G110	□ntermittent feed	□ull-circle helical rough milling	□apid feed	□ound groove internal rough milling CCW
G111	□ntermittent feed	□ull-circle helical rough milling	□apid feed	□ound groove internal rough milling CW
G112	□eed	□ull-circle fine milling	□apid feed	□ull-circle internal fine milling CCW
G113	□eed	□ull-circle fine milling	□apid feed	□ull-circle internal fine milling CW
G114	□eed	□ull-circle fine milling	□apid feed	□xternal round fine milling CCW
G115	□eed	□ull-circle fine milling	□apid feed	□xternal round fine milling CW
G134	□ntermittent feed	□ectangle rough milling	□apid feed	□ectangle groove internal rough milling CCW
G135	□ntermittent feed	□ectangle rough milling	□apid feed	□ectangle groove internal rough milling CW
G136	□eed	□ectangle fine milling	□apid feed	□ectangle groove internal fine milling CCW
G137	□eed	□ectangle fine milling	□apid feed	□ectangle groove internal fine milling CW
G138	□eed	□ectangle fine milling	□apid feed	□ectangle groove external fine milling CCW
G139	□eed	□ectangle fine milling	□apid feed	□ectangle groove external fine milling CW

□□□□□□ □ann□□ circl□ □xplanation□

Generally, a canned cycle consists of a se□uence of the following operations, see the right figure.

Operation 1… Positioning of axes X and Y

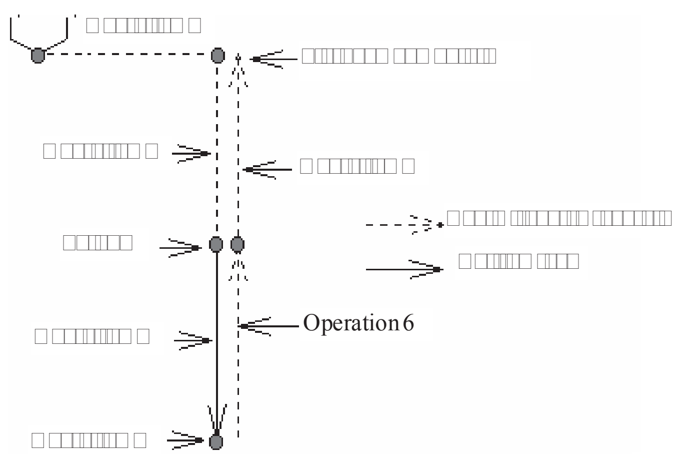
Operation 2…□apid traverse to point □ plane

Operation 3…Hole machining;

Operation 4…Operation at the bottom of hole;

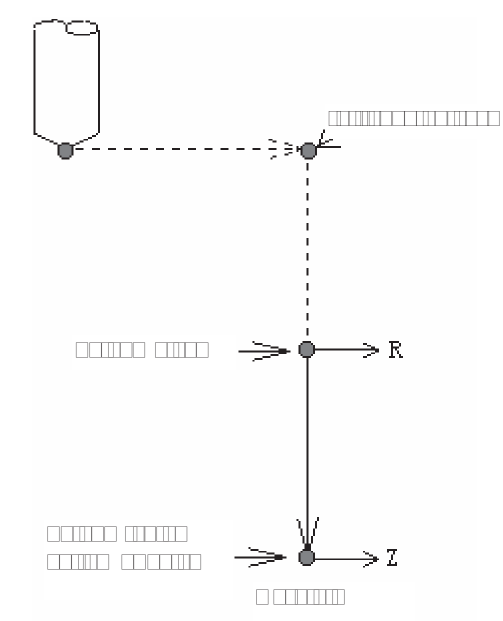
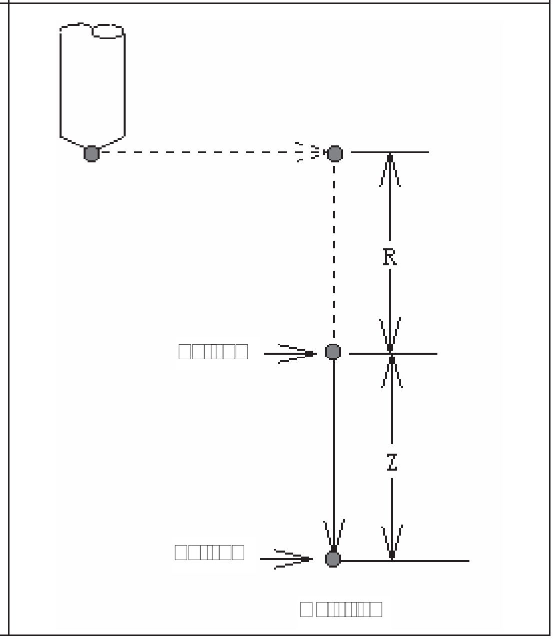
Operation 5…□etraction to point □ plane

Operation 6…□apid traverse to the initial Point



□□□□□□ □ □□□ □□

The data mode corresponded with G90 and G91 are different. The point □ plane and the absolute position machined at the bottom of the hole are specified by □ and Z values, when the command is G 90. The specified □ value is the distance relative to the initial plane, and the Z value is the distance relative to the □ point plane when the command is G91. □ee the □ig. 13.1 (□)

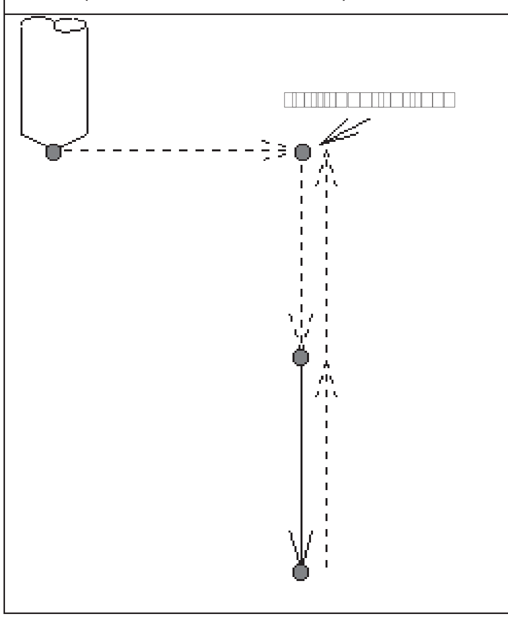
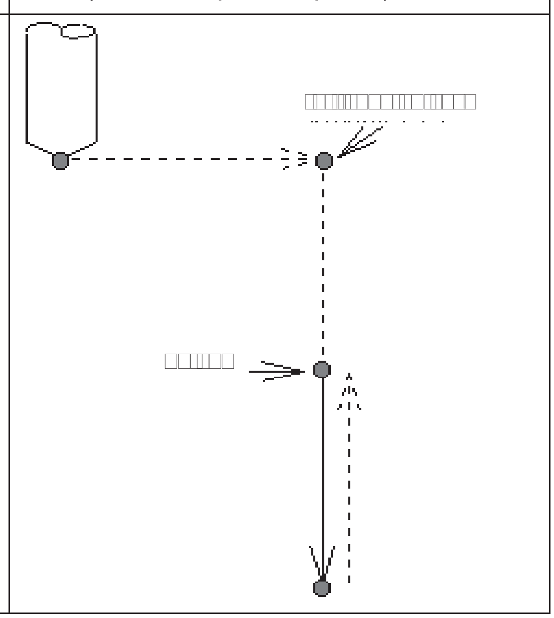
G90 (□bsolute command)	G91 (□ncremental command)
	

□ig. 13.1 (□) □bsolute and incremental commands for canned cycle

□□□□□□ □□turnin□ point l□□□l □ □□□□ □□

Tool can be returned to the initial plane or point □ plane according to G98 and G99 during returning. □ee the following figure □ig. 13.1 (C).

Normally, the initial hole machining is used by G99, the last machining is used with G98. The initial level will not be changed when the hole machining is done by G99.

G98 (□eturn to initial level)	G99 (□eturn to point □ plane)
	

□ig.13.1 (C) □evels for initial and point □

Note: The initial point level is an absolute position for hole machining axis direction which is indicated from the canned cycle cancellation to start.

3.15.1.5 Canned cycle cancellation

There are two ways for canned cycle cancel are listed below:

1. Canceling the canned cycle with the G80
2. The canned cycle is cancelled by the G00, G01, G02 and G03 command in group 01.

(1) When the canned cycle is cancelled by the command G80, if the G00, G01, G02 and G03 of the 01 group are not specified, then the reserved modal command (G00 or G01) performs motion before using canned cycle.

For example:

N0010 G01 X0 Y0 Z0 F800; （The modal command is G01 before entering the canned cycle）

N0020 G81 X10 Y10 R5 Z-50; （Entering canned cycle）

N0030 G80 X100 Y100 Z100; (The modal G01 command reserved before canned cycle performs cutting feed ）

If the G01 is not specified in the abovementioned program N0010, but G00, the G00 performs rapid positioning for N0030.

When both command G80 and commands G00, G01, G02 and G03 are specified in block, actions are performed by the latter, G00, G01, G02 and G03.

For example:

N0010 G01 X0 Y0 Z0 F800; (The modal command is G01 before entering the canned cycle)

N0020 G81 X10 Y10 R5 Z-50; (Entering canned cycle)

N0030 G00 G80 X100 Y100 Z100; (The G00 performs positioning at the rapid rate, and the modal command G00 is saved)

Note: The cutting feedrate by F command is still held on even if the canned cycle is cancelled.

3.15.1.6 General command format for canned cycle

Once the hole machining data is specified in the canned cycle, it is held until the canned cycle is cancelled. So the hole machining data should be outright specified at the beginning of the canned cycle, only the modified data is specified in the following canned cycle.

The general command format of canned cycle: G_ X_ Y_ R_ Z_ Q_ P_ F_ L;

All commands for canned cycle are listed in above-mentioned format. But it is not needed to specify the above-mentioned format in each canned cycle. For example, the canned cycle can be performed as long as the G command (hole machining) and any of X, Y, Z and R are specified; additionally, Q or P is not available in some canned cycle G command (hole machining), the command is disabled even if these data are specified, they are regarded as modal data memories only.

Table 13.1.□ □ommand e□□lanations for canned cycle

□□ecifying content	□ddress	□□□lanation for command address
□ole machining	G	Refer to the canned cycle list.
□ole position data	X，Y	Specifying the hole position with the absolute and incremental value, control is same with G00 position. □nit: mm;
□ole machining data	R	See the fig.13.1 (B), the distance from initial point level to point R plane is specified by using the incremental value, or specifying the coordinate value of the point □ by absolute value. □nit: mm;
	Z	□ole depth. See the fig.13.1 (A), the distance from R point to the bottom of a hole is specified by using the incremental value or specif□ing the coordinate value of the hole bottom by absolute value. □nit: mm;
	Q	Specifying each cut-in in G□3 and G83 or translational value in G□□ and G87. □nit: mm;
	P	Specifying the dwell at the bottom of a hole. Relation of time and the numerical specified are same with G0□. □nit: ms;
	□	□achining cycle for □holes is performed from start (start position of block) to XY coordinate position.
	F	The cutting feedrate is specified, tooth pitch is indicated in G□□ and G8□.

A part of command of canned cycle such as G110, G111, G112, G113, G11□, G115, G13□, G135, G13□, G13□, G138 and G13□ are explained in the following chapters or sections.

3.15.□ □escri□tion for canned cycle

3.15.□.1 □igh□s□eed □ec□ drilling cycle G□3

Format: **G□□/G□□ G□3 X_ Y_ R_ Z_ Q_ F_ L_;**

Function: This kind of cycle performs high-speed peck drilling, it performs intermittent cutting feed to the bottom of a hole, and eliminating the chips from the hole simultaneously.

□□□lanation: Refer to the command explanation of canned cycle in Table 13.1.□.

□ycle □rocess:

(1) Positioning to XY plane level at the rapid traverse;

(2) □own to the point R plane at the rapid traverse rate;

(3) □utting feed for Q distance;

(□) Retract *d* distance in rapid traverse;

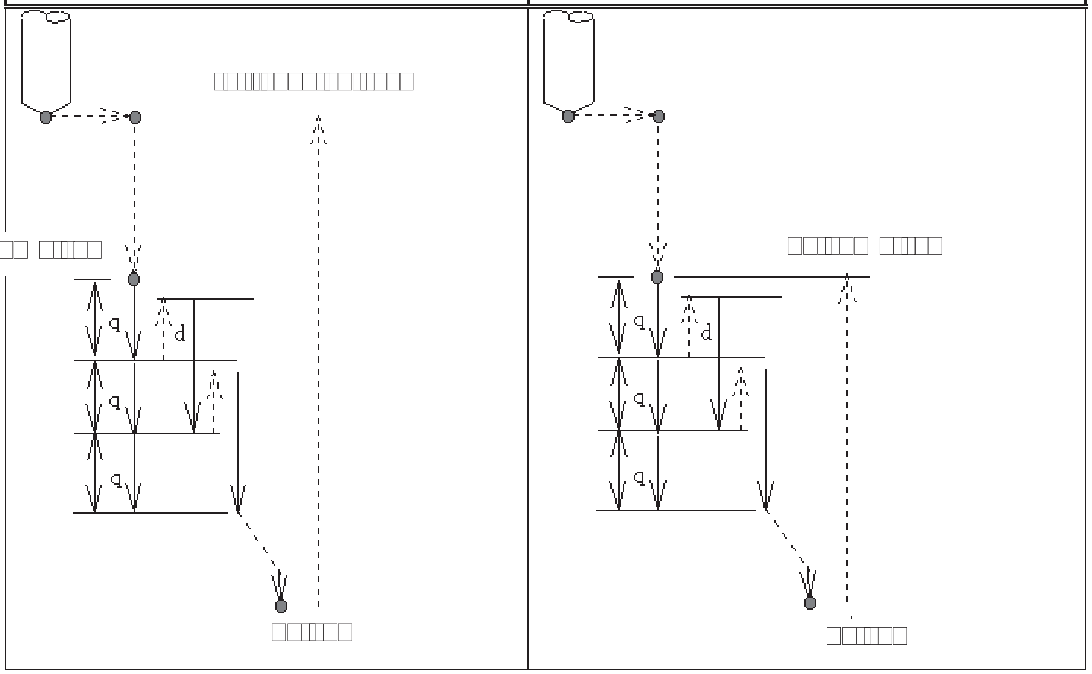
(5) □utting feed for (Q□d) distance

(□) □achine to the Z axis hole bottom by cycling the (□) and (5);

(□) Return to the start point level or point R plane according to G□8 or G□□ at the rapid traverse.

Command Path:

G98 Return to the initial plane at the rapid traverse	G99 Return to the point R plane at the rapid traverse



Related Explanation:

(1) This kind of cycle is peck drilling for Q value intermittent feeding along the Z-axis direction. The Q value should be positive, the sign is ineffective even if the negative value is specified. If the Q value is not specified, then it defaults 0.1mm. If a depth to be cut is less than the Q value, then cut to the bottom of the hole without tool retraction at the rapid traverse for the first time.

(2) To remove chips from the hole easily, a small value can be set for retraction. This allows drilling to be performed efficiently. The tool is retracted in rapid feed, the retraction amount d is set by parameter No.51, the default is 1000, unit: 0.001mm.

(3) The command P is disabled, but its value is reserved as canned cycle modal value.

3.15.□.□ Left-handed tapping cycle G74

Format: G98/G99 G74 X_ Y_ R_ Z_ P_ F_ L

Function: This cycle performs left-handed tapping. In the left-handed tapping cycle, the spindle rotates clockwise for tapping till the bottom of the hole has been reached, then retracts by counter-clockwise after dwell.

Explanation: For canned cycle explanation, see the Table 13.1.□

Thereinto, the F is indicated for tooth pitch. The value range are indicated as 0.001～500.00mm (metric), 0.0□□～25□00 teeth/inch (inch)

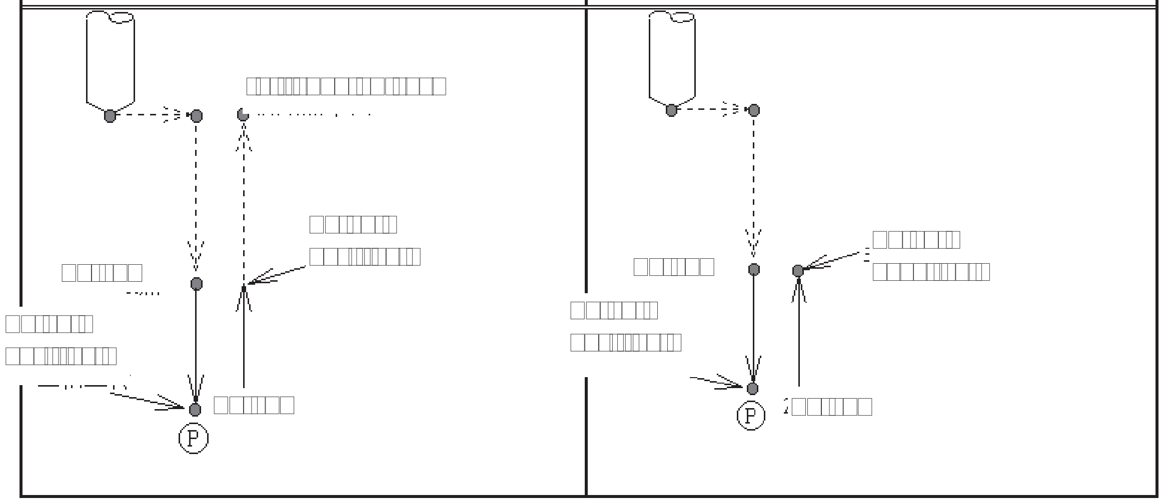
Cycle Process:

(1) Positioning to XY plane level at the rapid traverse;
(2) Down to the point R plane at the rapid traverse;
(3) Tapping to the bottom of a hole;
(4) The spindle stops;
(5) Pause for time P if dwell is specified;
(6) The spindle rotates CCW, and then retracts to point R plane;

(7) The spindle is stopped; pause for time P if dwell is specified;
(8) Spindle rotates CW;
(9) Return to the initial plane if it is G98.

Command Path:

G98 (Mode for returning to initial plane)	G99 (Mode for returning to R point plane)



Related Explanation:

(1) Tapping to the bottom of a hole it will not be returned immediately even if the P is omitted or regarded as 0 in this cycle, it will be returned after a dwell time (2s), and this time is set by system.

(2) The F is tapping modal value, the last tapping F value is taken when it is omitted, or alarm will be generated if it does not exist.

(3) The metric or inch of the F value is determined by G20 (metric) or G21 (inch).

(4) The command Q is disabled in this cycle, but its value will be reserved as canned cycle modal value.

3.15.□.3 Tapping cycle G84

Format: **G98/G99 G84 X_ Y_ R_ Z_ P_ F_ L_ ;**

Function: This cycle is used to machine a thread. The tapping is performed by spindle rotating positively, when the bottom of a hole has been reached, the spindle is retracted in the reverse direction.

Explanation: For command explanation of canned cycle, see the Table 13.1.□

Thereinto, the F is tooth-pitch. The value range is 0.001□500.00mm (metric), 0.0□□25□00 tooth/inch (inch).

Cycle Process:

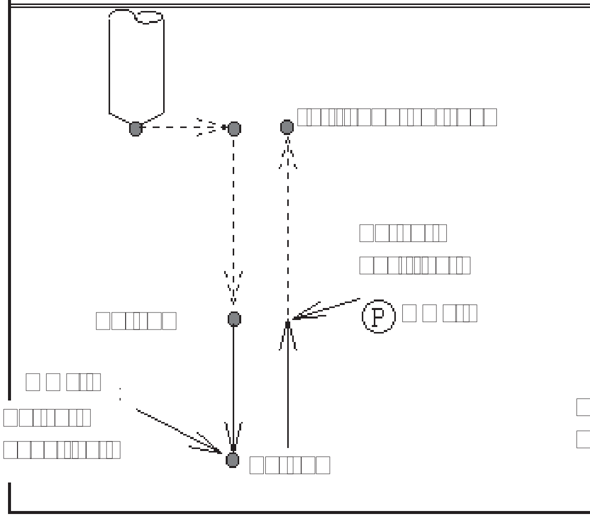
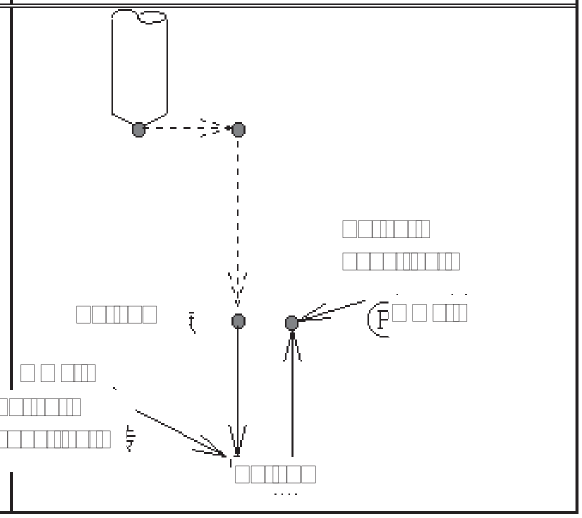
(1) Positioning to the XY plane level at the rapid traverse;
(2) Down to the point R plane at the rapid traverse;
(3) Tapping to the bottom of a hole;
(4) Spindle stops;
(5) For dwell time P if it is commanded
(6) Spindle returns to the point R plane in reverse direction;

(7) Spindle stops; for dwell time P if the P is commanded;

(8) The spindle is rotated in the positive direction;

(9) Returning to the initial point level if it is G98.

Command Path:

G98 (Mode for returning to initial point level)	G99 (Mode for returning to point R plane)
	

Related Explanation:

Please refer to the related explanation for G□□ (Counter tapping cycle)

3.15.□.□ Drilling cycle, spot drilling cycle G□1

Format: G□□/G□□ G□1 X□ Y_ R_ Z_ F_ L_ ;

Function: 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.

Explanation: For the command explanation of canned cycle, see the Table 13.1.□.

Cycle Process:

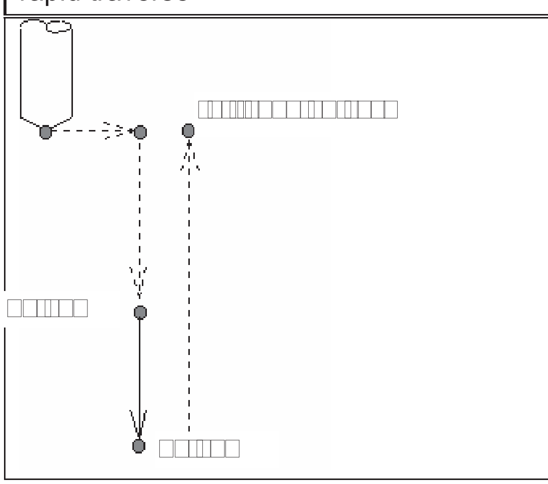
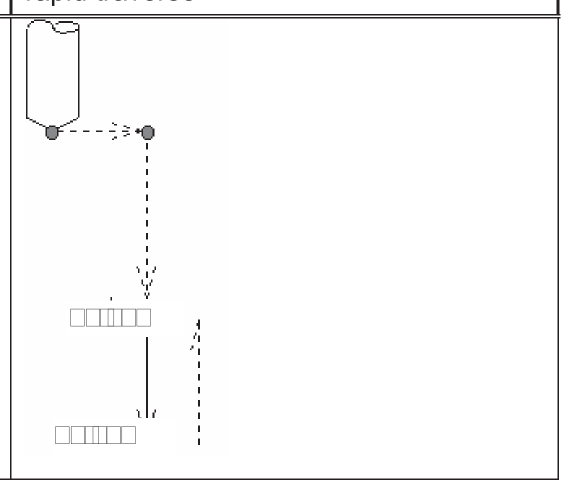
(1) Positioning to the XY plane level position at the rapid traverse;

(2) Down to the point R plane at the rapid traverse;

(3) Cutting feed to the bottom of the hole;

(□) Returning to the initial point or point R plane at rapid traverse according to the G98 or G99;

Command Path:

G98 Return to the initial plane at the rapid traverse	G99 Return to the R point plane at the rapid traverse
	

Related Explanation:

The command Q or P is disabled in this cycle, but its value will be saved as canned cycle modal value.

3.15.□.5 Drilling cycle/counter boring cycle G82

Format：G98/G99 G82 X_ Y_ R_ Z_ P_ F_ L_；

Function: Cutting feed is performed to the bottom of the hole. Hole depth precision is added when the dwell is performed, and then the tool is retracted from the bottom of the hole at rapid traverse.

Explanation: For the command explanation of these canned cycles, see the Table 13.1.□

Cycle Process:

(1) Positioning to the XY plane level at the rapid traverse;

(2) Down to the point R plane at the rapid traverse;

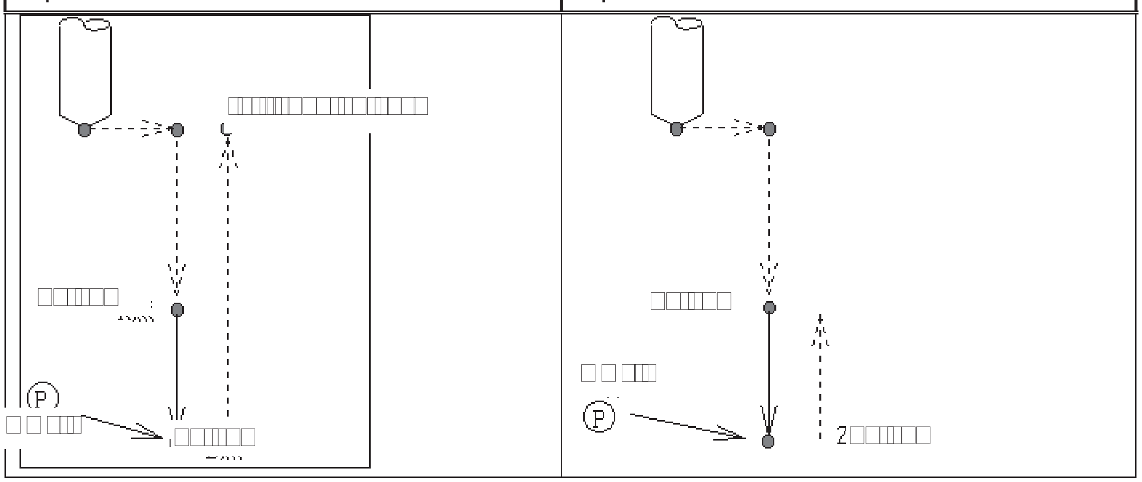
(3) Cutting feed to the bottom of a hole

(4) Dwell for P time if it is commanded.

(5) Returning to the initial point or point R plane according to G98 or G99 at the rapid traverse;

Command Path:

G98 Return to the initial point level at the rapid traverse	G99 Return to the point R plane at the rapid traverse



Related Explanation:

(1) They are basically the same as G81 (drilling and spot-drilling machining), it is up after dwell at the bottom of a hole only (the dwell time is specified by P, the dwell will not be executed if it is not specified, and the command action is same as that of G81). In the blind hole, the accuracy of hole can be improved by the dwell.

(2) The command Q is disabled in this cycle, but its value will be reserved as the canned cycle modal value.

3.15.□.6 Peck drilling cycle G83

Format: G98/G99 G83 X_ Y_ R_ Z_ Q_ F_ L_ ;

Function: This cycle performs high-speed peck drilling; it performs intermittent cutting feed to the bottom of a hole while removing chips from the hole.

Explanation: The command explanation for canned cycle, see the table 13.1.□.

Cycle Process:

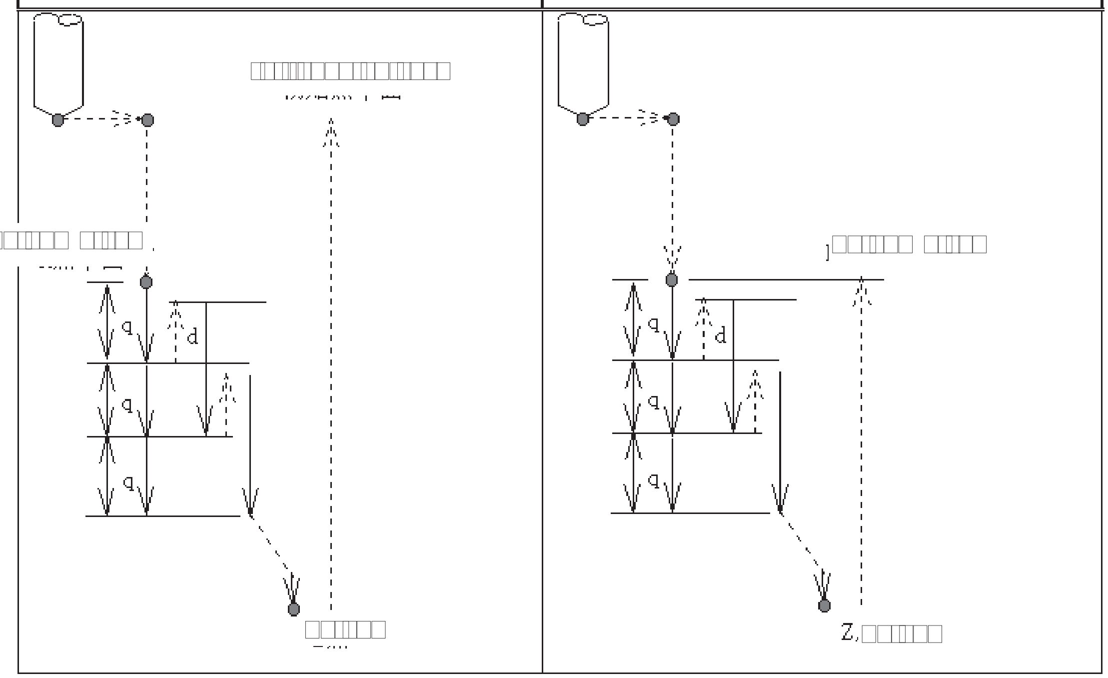
(1) Positioning to the XY plane level at the rapid traverse;

Volume □ Programming

(2) □own to the point R plane at the rapid traverse;
(3) □utting feed for Q distance;
(□) Retract to the point R plane at the rapid traverse;
(5) Rapid feed to d distance to the end surface
(□) □utting feed for (Q□d) distance;
(□) □ycling (□) (5) and (□) to the bottom of a hole along Z-axis;
(8) Return to the initial point or point R plane according to the G□8 or G□□ at the rapid traverse;

□ommand Path:

G□8 returned to the initial plane at the rapid traverse	G□□ returned to the point R plane at the rapid traverse



Related □□□lanation:

(1) Same as G□3, after feeding for Q, it returns to the point R plane at the rapid traverse firstly, and then rapid feeds to d mm to the end surface, then cutting feed is applied and the cycle is performed in turn. The Q value should be positive, even if the negative value is specified, and the sign is also disabled. Q value 0.001mm is defaulted if Q value is not specified; d, is set by the parameter No.52, its default value is 1000, and the unit is 0.001mm. If the cutting depth is less than the Q value, then cutting to the bottom of a hole at the first time, and rapid traverse retraction is not performed.

(2) The command P is disabled in this cycle, but its value will be reserved as canned cycle modal value.

3.15.□.□ □oring cycle G□5

Format: **G□□/G□□ G□5 X_ Y_ R_ Z_ F_ L_ ;**

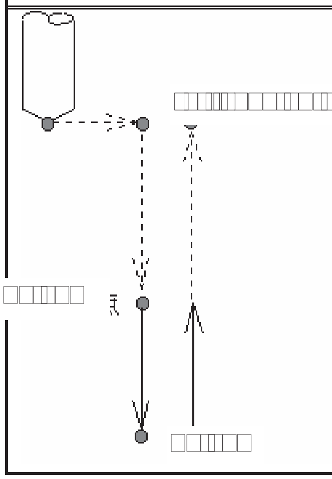
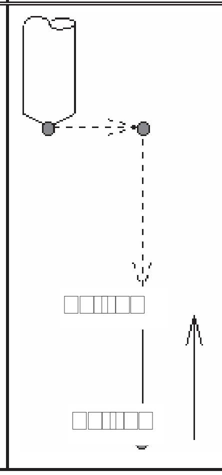
Function: After positioning along X and Y axes, rapid traverse is performed to point R; the boring is performed from point R to point Z thereafter. □utting feed is performed to return point R plane when the Z point has been reached the bottom of a hole.

□□□lanation: □ommand explanation for the canned cycle, see the table 13.1.□.

□ycle □rocess:

(1) Positioning to the XY plane level at the rapid traverse;
(2) □own to the point R plane at the rapid traverse;
(3) □utting feed to the bottom of a hole;
(□) □utting feed to the point R plane;
(5) Returning to the initial point level if it is G□8;

□ommand Path:

G□8 (□ode for returning to initial point level)	G□□ (□ode for returning to point R plane)
	

Related □□□lanation:

(1) This cycle is used to bore a hole. The command motion is basically same as the G81 (□rilling, Spot-drilling cycle), the difference is that by the G81 it returns to the point R plane in rapid traverse rate, while by the G85 it returns to the point R plane in feedrate when the cutting feed reaches the bottom of a hole.
(2) The Q and P commands are disabled in this cycle, but its value is reserved as the canned cycle modal value.

3.15.□.□ □oring cycle G□6

Format: **G□□/G□□ G□6 X_ Y_ R_ Z_ F_ L_ ;**

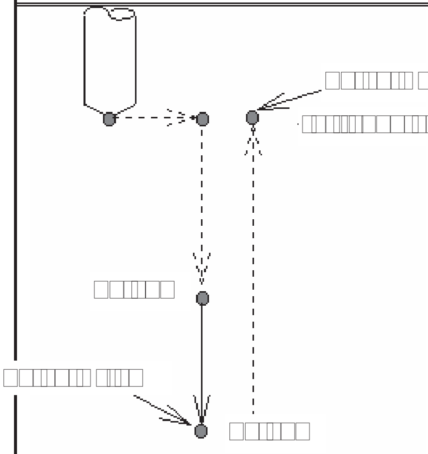
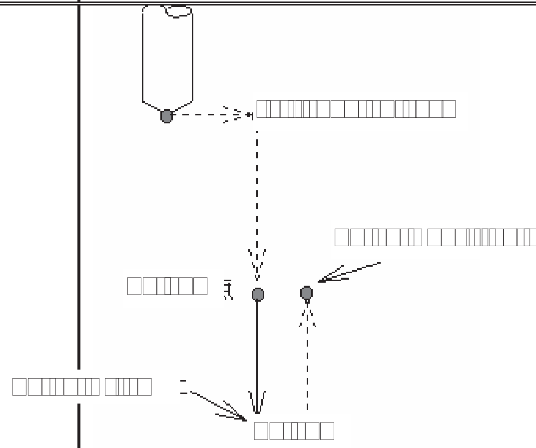
Function: After positioning along X and Y axes, rapid traverse is performed to R point, and the boring is performed from point R to point Z. The tool is retracted in rapid traverse and spindle is rotated positively when the spindle is stopped at the bottom of the hole.

□□□lanation: For command explanation for canned cycle, see the table 13.1.□.

□ycle □rocess:

(1) Positioning to the XY plane level at the rapid traverse;
(2) □own to the point R plane at the rapid traverse;
(3) □utting feed to the bottom of a hole;
(□) The spindle stops;
(5) Returning to the initial point or point R plane at rapid traverse according to the G□8 or G□□;
(□) The spindle is rotated in the positive direction;

□ommand Path:

G□8 (□ode for returning to start point level)	G□□ (□ode for returning to point R plane)
	

Related □□□lanation:

(1) This cycle is used to be bore a hole. The command operation is basically same with G81, only spindle rotation status is different. After cut feeds to the bottom of a hole, the □05 is executed (spindle stops), then the point R plane is retracted at the rapid traverse, the □03 is then performed (spindle rotates positively) regardless of the currently spindle rotation status and the positive or negative rotation are specified before the canned cycle.

(2) The command Q and P are disabled in this cycle, but its value is reserved as canned cycle modal value.

3.15.□.□ □oring cycle G□□

Format: G□□/G□□ G□□ X_ Y_ R_ Z_ P_ F_ L_ ;

Function: A dwell is performed at the bottom of a hole, the spindle is stopping. If the manual operation is applied now, tool can be removed manually. It is better to retract the tool safely from the hole regardless of any kind of manual operation. It is rapidly retracted to point R or initial plane when the automatic operation is performed again, the spindle is stopped and G88 is finished.

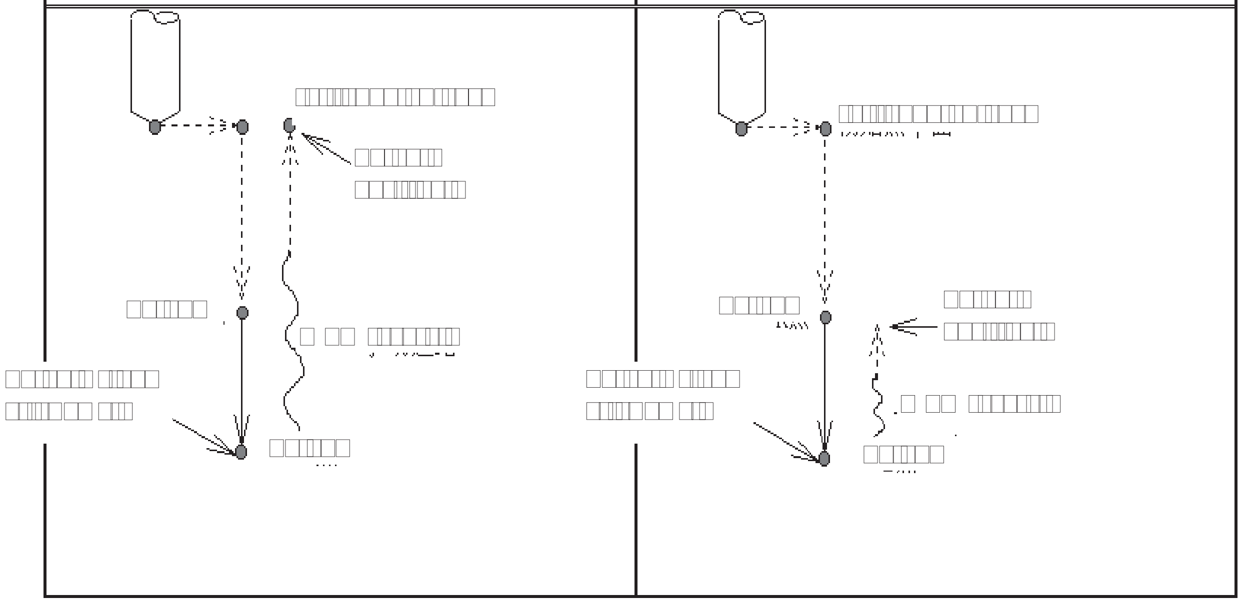
□□□lanation: For the command explanation of the canned cycle, see the table 13.1.□.

□ycle □rocess:

(1) Positioning to the XY plane at the rapid traverse rate;
(2) □own to the point R plane at the rapid traverse rate;
(3) □utting feed to the bottom of hole;
(□) The spindle is stopped;
(5) P time is delayed if it is specified.
(□) □anual operation will be performed if the dwell is executed.
(□) Restoring the automatic mode, retracting to initial point or point R plane according to the G□8 or G□□ at the rapid traverse rate.
(8) The spindle rotates positively;

□ommand Path:

G□8 (□ode for returning to initial plane)	G□□ (□ode for returning to point R plane)



Related □□□lanation:

The command Q is disabled in this cycle, but its value is reserved as the canned cycle modal value.

3.15.□.1□ □oring cycle G□□

Format: G□□/G□□ G□□ X_ Y_ R_ Z_ P_ F_ L_ ;

Function: This cycle is used to bore a hole normally. This cycle performs a dwell at the bottom of the hole; the tool is then retracted from the bottom of the hole at the rapid traverse rate.

□□□lanation: For the command explanation of the canned cycle, see the table 13.1.□.

□ycle □rocess:

(1) Positioning to XY plane at the rapid traverse rate;

(2) □own to the point R plane at the rapid traverse rate;

(3) □utting feed to the bottom of a hole;

(□) For dwell time P if the P is specified;

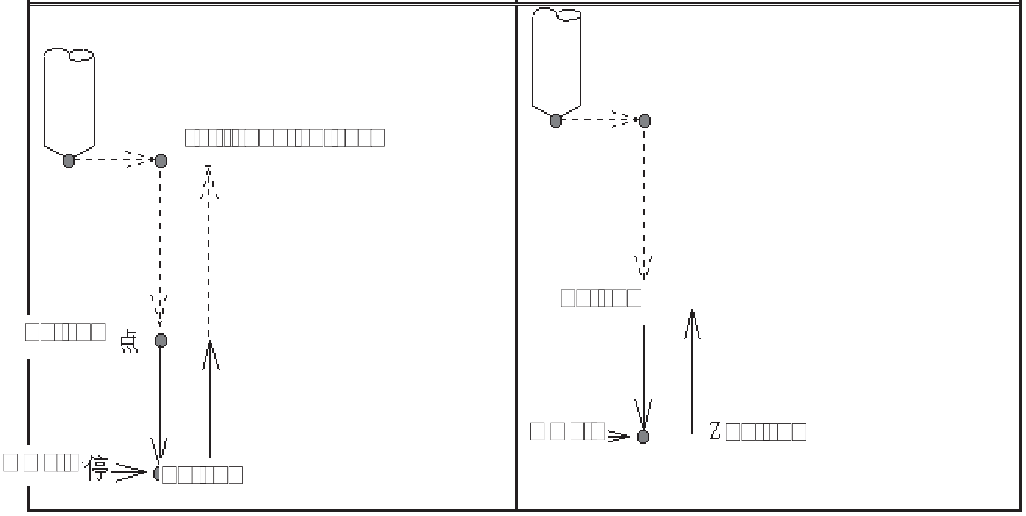
(5) □utting feed to the point R plane;

(□) Returning to the initial point level if it is G□8;

(□) Returning to the initial point or point R plane at the rapid traverse according to the G□8 or G□□;

□ommand Path:

G□8 (□ode for returning to initial point level)	G□□ (□ode for returning to point R plane)



Related □□□lanation:

(1) G8□ (Boring cycle) is basically same as the G85, a dwell is applied at the bottom of a hole (□well time is specified by P, if it is not specified, the dwell is not applied, the command operation is same to the G85)

(2) The command Q is disabled in this cycle, but its value is reserved as canned cycle modal value.

3.15.□.11 Groove rough milling inside the round G11□G111

Format:

	G11□	
G□□G□□		**X_ Y_ R_ Z_ □_ □_ Q_ □_ □_ □_ F_**
	G111	

Function: From the beginning of the center point, arc interpolations are performed helically till the round groove of programming dimension has been machined.

□□□lanation: For command explanation of the canned cycle, see the table 13.1.□.

G110: Groove rough-milling inside the round in □□W;

G111: Groove rough-milling inside the round in □W;

I: I is radius inside the round groove, it should be more than the radius of current tool.

W: The firstly cutting depth is from the R reference level to the undersurface along the Z axis direction, it should be more than 0 (The first cutting position is over the bottom of the groove, then bottom position is regarded as machining position);

Q: The cutting incremental value each time along Z axis direction;

□: The width increment of cut inside XY plane, it should be less than the tool radius, and more than 0;

□: The distance to the end machining plane at the rapid traverse, it should be more than 0 when cutting;

□: Tool radius serial number, the value range is 0□32, 0 is the default of □0. The current tool radius is determined by the specified serial number.

□ycle □rocess:

(1) Positioning to the XY plane level at the rapid traverse rate;

(2) □own to the point R plane at the rapid traverse rate;

(3) □ut W depth downwards in cutting feedrate

(□) □ill a round face with radius I helically by □ increment each time from center point to outside.

(5) The Z axis is retracted to the R reference surface at the rapid traverse rate;

(□) X and Y axes are positioned to the center at the rapid traverse rate;

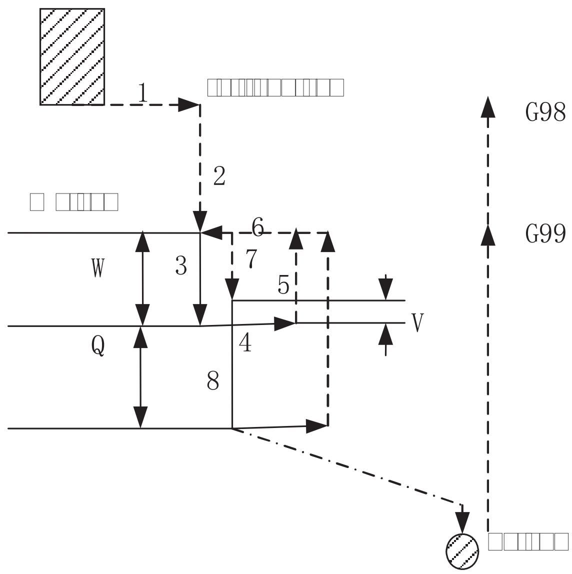
(□) □own to distance □ to the end machining surface along Z axis at the rapid traverse rate;

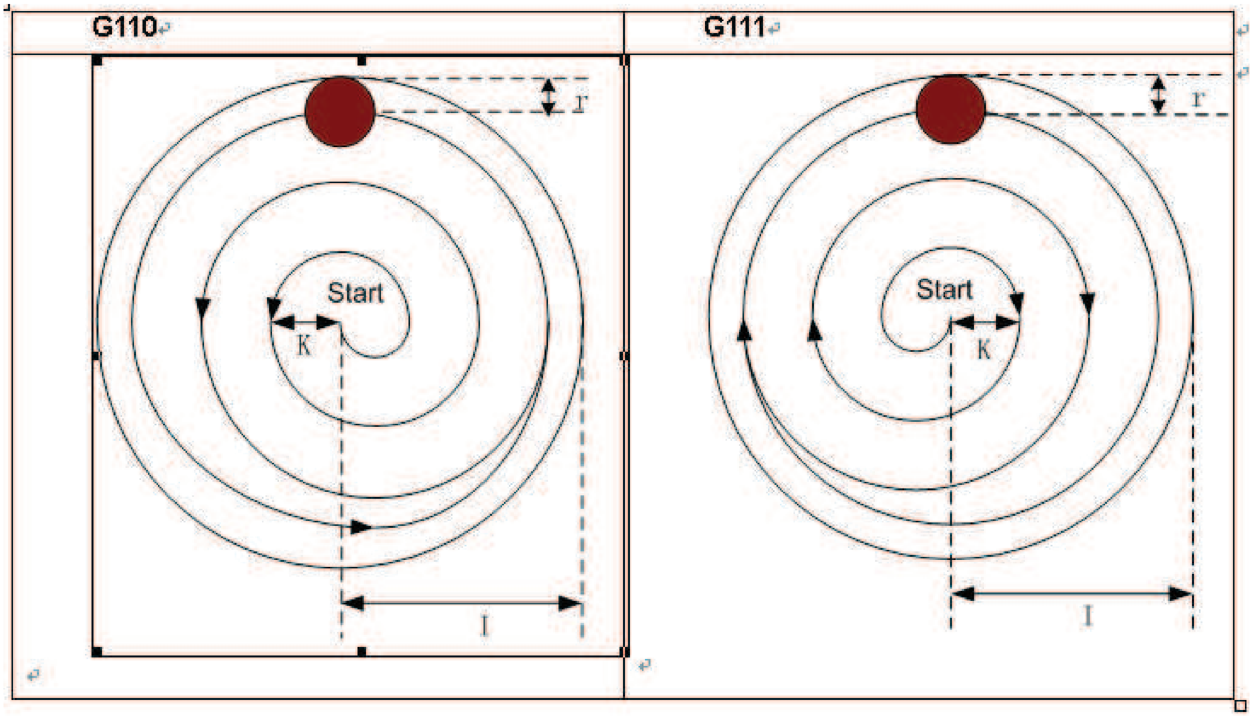
(8) □ut along Z axis for (Q□□) depth;

(□) □ycling the operations from (□) □ (8) till the round surface of total depth is finished.

(10) Return to the initial plane or point R plane according to G□8 or G□□.

□ommand Path:



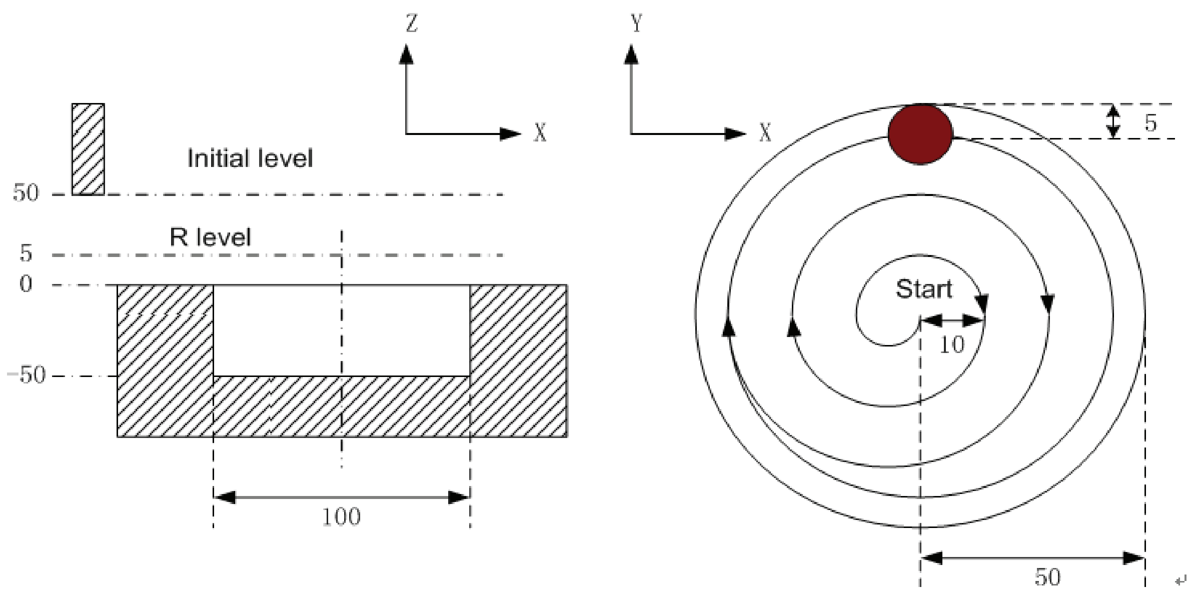


Related □□□lanation:

The P and □ are disabled in this cycle, but the P value will be reserved as canned cycle modal value.

For e□am□le: A round inside groove rough-milling is specified in canned cycle G111, see the following

Figure



G□0 G00 X50 Y50 Z50; (G00 positioning at the rapid traverse rate)

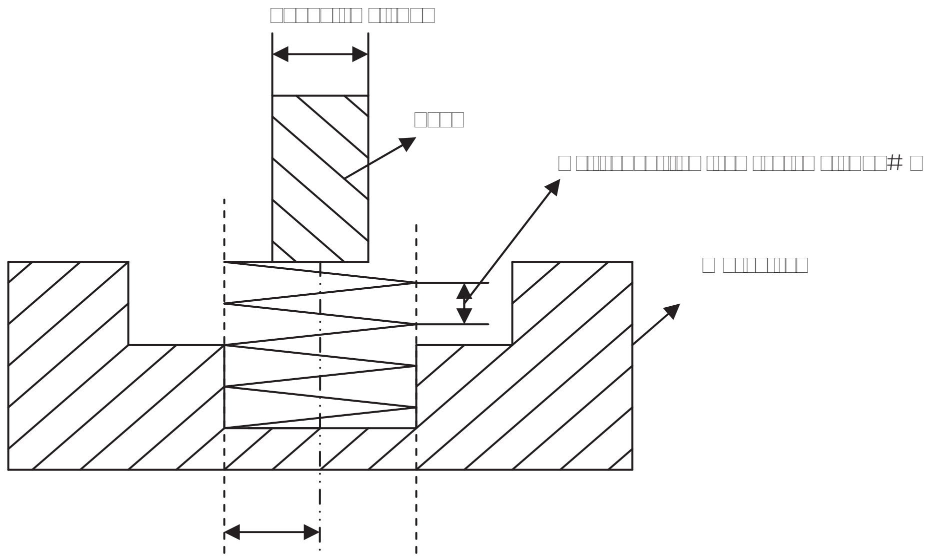
G□□ G111 X25 Y25 R5 Z-50 150 W20 Q10 □10□10 F800 □1; (Rough-milling cycle inside the round groove □1□5)

G80 X50 Y50 Z50; (□anceling canned cycle, returning from the point R plane)

□30;

Note: □et the □□□ □arameter value to one □hich is more than 1□□by G11□ and G111 it feeds helically along Z a□is. Rough□milling machining can be directly □erformed for non□groove □or□□iece.

See the following figure for helical cutting path:



3.15.□.1□ Fine□milling cycle inside full circle G11□G113

Format:

G11□

G□□G□□　　X_　Y_　R_　Z_　□_　□_　□_　F_

G113

Function: A fine-milling inside the full circle is finished with the specified radius value I and direction, the tool is retracted after the fine-milling.

□□□lanation: For command explanation of canned cycle, see the table 13.1.□.

G112: Fine-milling cycle inside the full circle in □□W.

G113: Fine-milling cycle inside the full circle in □W

I: Fine-milling circle radius, the value range is indicated as 0□□□□□.□□□mm, the absolute value is taken when it is negative.

□: Fine-milling distance from start point to the center point, the value range is indicated as 0□□□□□.□□□mm, the absolute value is taken when it is negative

□: Se□uence number of tool radius, the value range is indicated as 0□32, the 0 is default of □0. The current tool radius value is taken according to the specified se□uence number.

□ycle □rocess:

(1) Positioning to the XY plane level at the rapid traverse rate;

(2) □own to the point P level at the rapid traverse rate;

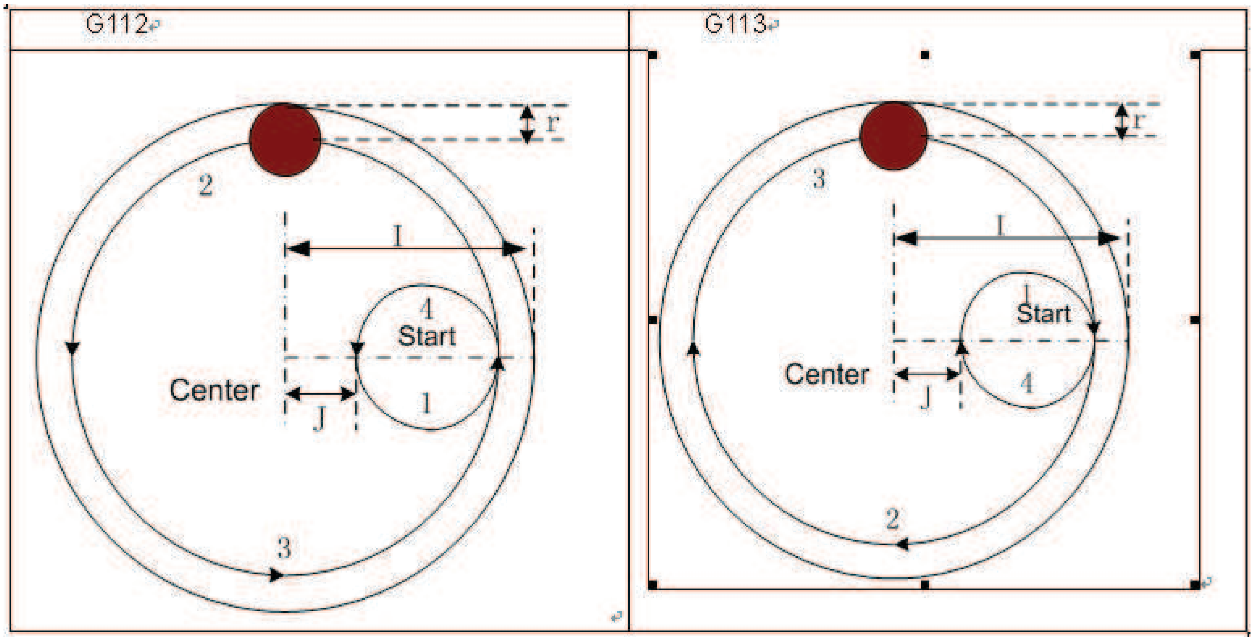
(3) Feed to the bottom of a hole;

(□) Perform the circle interpolation by the path of transit arc 1;

□olume □ Programming

(5) Perform the full circle interpolation by the path of arc 2 and arc 3;
(□) Perform circular interpolation by the path of transit arc □and return to the start point;
(□) Return to the initial point level or point R plane according to G□8 or G□□.

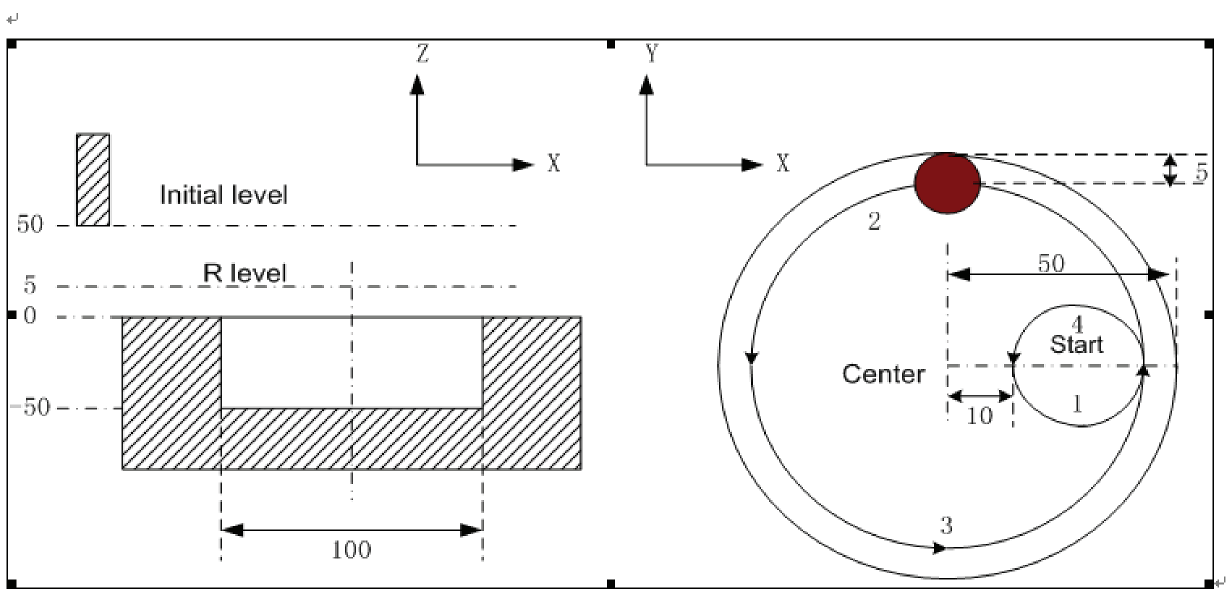
□ommand Path:



Related □□□lanation:

The commands Q, P and □are disabled in this cycle, but the Q and P value will be reserved as the canned cycle modal value.

For e□am□le: Fine-mill a finished rough-milling round groove by the canned cycle G112 command, see the following figure:



G□0 G00 X50 Y50 Z50; (G00 rapid positioning)
G□□ G112 X25 Y25 R5 Z-50 150 □10 F800 □1; (Start canned cycle, fine-milling cycle

inside the circle at the bottom of a hole
(1□5)

G80 X50 Y50 Z50;　　(The canned cycle is cancelled, returning from the point P level)
□30;

3.15.□.13 Fine-milling cycle outside circle G114, G115

Format:

G114
G98/G99　　X_　Y_　R_　Z_　I_　J_　D_　F_;
G115

Function: A fine-milling outside the full circle is performed by the specified radius value and the direction, and the tool is retracted after the fine-milling is finished.

Explanation: For command explanation of canned cycle, see the table 13.1.□.

G114: Finish-milling cycle for outside circle in CCW.

G115: Finish-milling cycle for outside circle in CW.

I: A fine-milling circle radius, the value range is indicated as 0□□□□□.□□□mm, the absolute value is taken when it is negative.

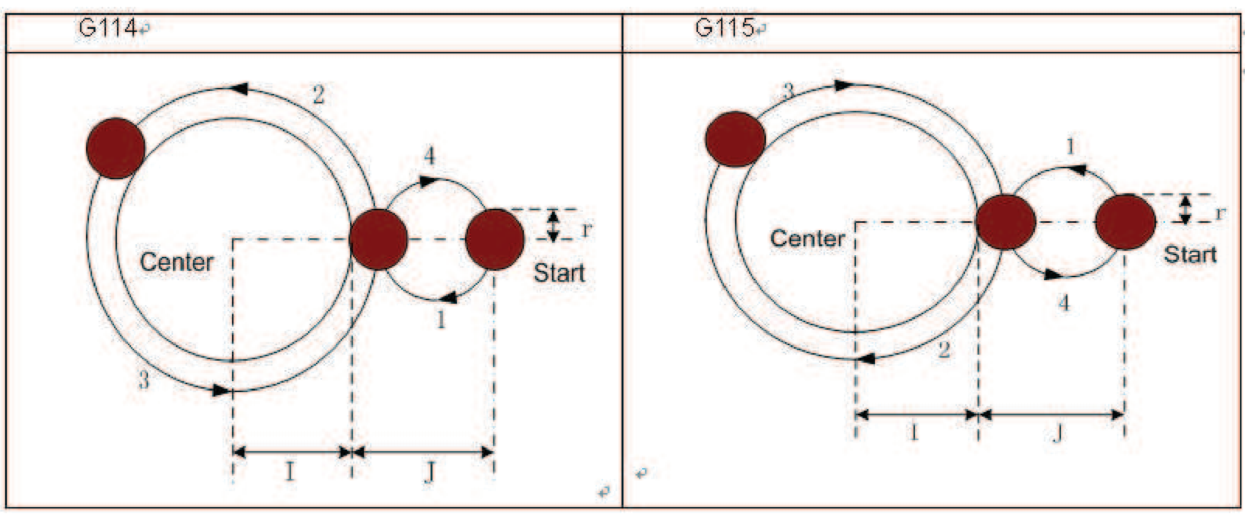
J: Distance of fine-milling between the start point and the circle, the value range is indicated as 0□□□□□.□□□mm; the absolute value is taken when it is negative.

D: The sequence number of tool radius, the value range is 0□32, 0 is the default of D0. The current tool radius value is taken according to the specified sequence number.

Cycle Process:

(1) Positioning to the XY plane level at the rapid traverse rate;
(2) Down to the point R plane at the rapid traverse rate;
(3) Cutting feed to the bottom of a hole;
(4) Perform the circle interpolation by the path of transit arc 1;
(5) Perform the full circle interpolation by the path of arc 2 and arc 3;
(6) Perform circular interpolation by the path of transit arc 4 and return to the start point;
(7) Return to the initial point level or point R plane according to G98 or G99.

Command Path:



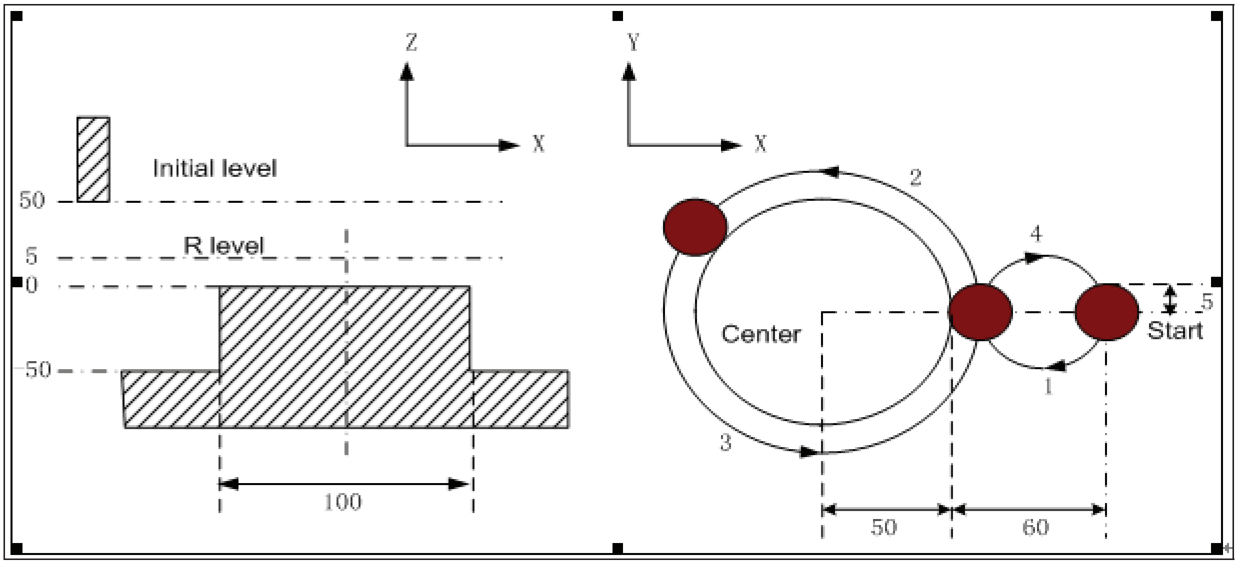
Related Explanation:

(1) The interpolation direction of between transit arc and fine-milling arc are different when the fine-milling outside circle is performed, the interpolation direction in command explanation is

the interpolation direction of fine-milling arc.

(2) The command Q, P and □are disabled in this cycle, but the Q and P value are reserved as canned cycle modal value.

For e□am□le: A finished rough-milling round groove is performed by fine-milling with the canned cycle G11□command, see the following figure :



G□0 G00 X50 Y50 Z50; (G00 rapid positioning)

G□□ G11□X25 Y25 R5 Z-50 150 □□0 F800 □1; (Start canned cycle, the fine-milling cycle is performed outside the circle at the bottom of a hole □1□5)

G80 X50 Y50 Z50; (The canned cycle is cancelled, returning from the point R plane)

□30;

3.15.□.1□Rectangle groove rough□milling G13□G135

Format: G13□ G□□G□□

X_ Y_ Z_ R_ □_ □_ □_ □_ Q_ □_ □_ □_ F_

G135

Function: From the center of the rectangle, the linear cutting cycle is applied by the specified parameter

data, till the rectangle groove with programmed dimension is made out.

□□□lanation: For command explanation of canned cycle, see the table 13.1.□.

G13□: Rectangle groove rough-milling in □□W

G135: Rectangle groove rough-milling in □W

I: The width of rectangle groove along the X axis direction

□: The width of rectangle groove along the Y axis direction.

□: The cut width increment inside XY plane, it is less than the tool radius, but, more than 0.

W: For the first cutting along the Z axis direction, the distance is downward to the R reference surface, it is more than 0 (if the first cutting is over the position of the bottom of the groove, then the bottom of the groove is taken as the machining position)

Q: The cutting incremental value each time along Z axis.

□: □istance to the end machining surface, which is more than 0, when the rapid traverse is executed.

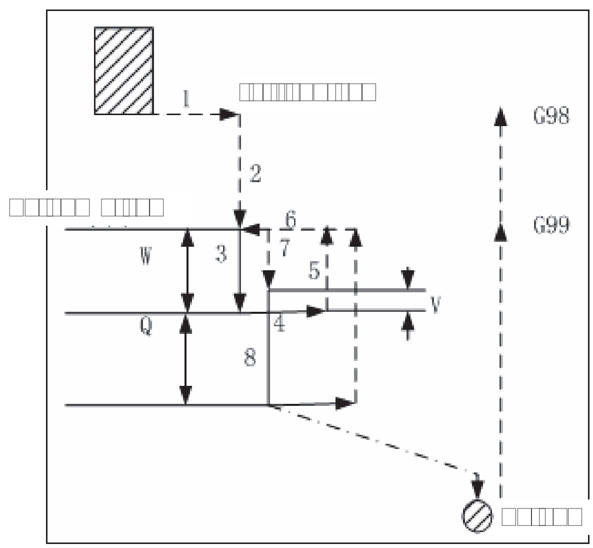
□: Corner arc radius, if it is omitted, that is no corner arc transition is not shown.

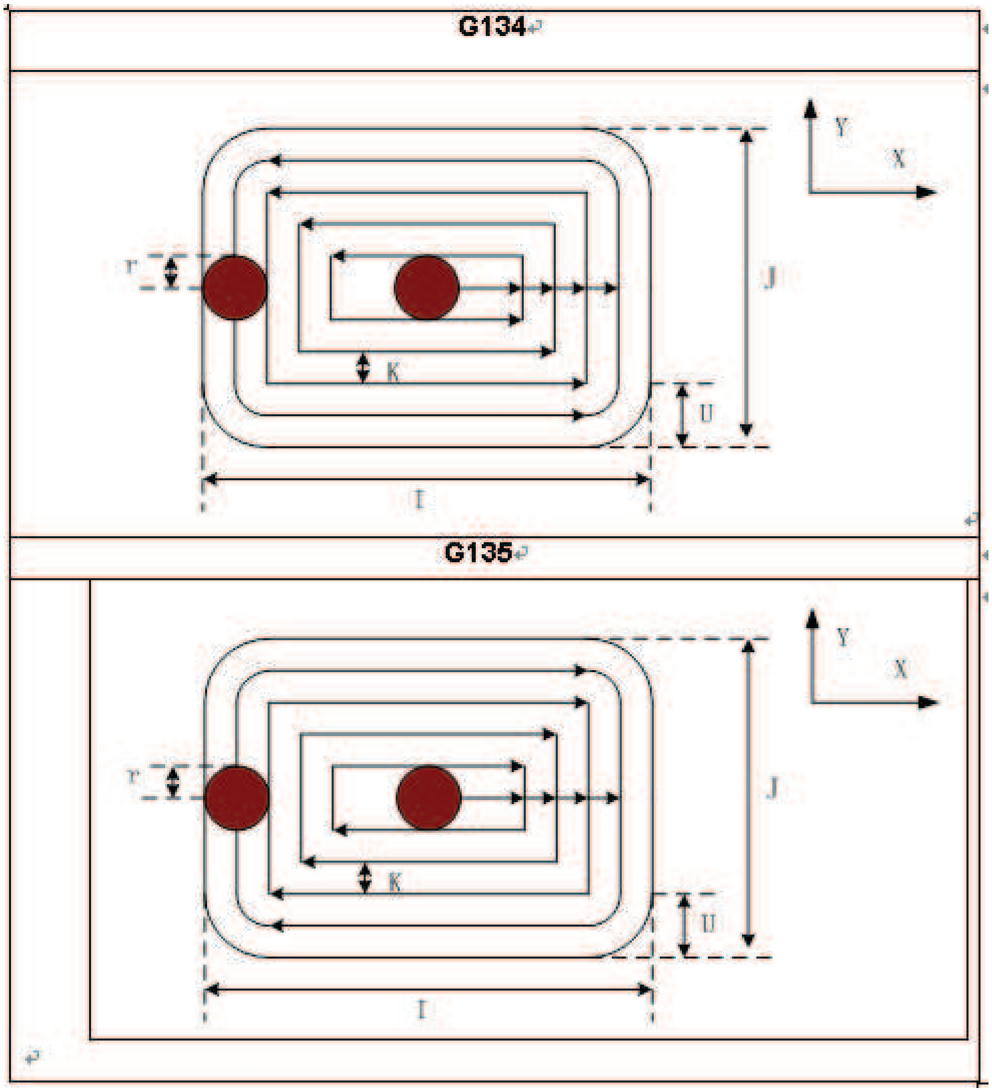
D: Sequence number of tool radius, its value range is indicated as 0 ~ 32, thereunto, the 0 is default of D0. The current tool radius value is taken out according to the specified sequence number.

Cycle Process:

(1) Positioning to the XY plane at the rapid traverse rate;
(2) Down to the point R plane at the rapid traverse rate;
(3) W distance depth is cut downwards by cutting feedrate
(4) Mill a rectangle face helically by □ increment each time from center point to outside.
(5) R reference surface is retracted along the Z axis at the rapid traverse rate.
(6) The center of rectangle is positioned along the X and Y axes at the rapid traverse rate.
(7) Down to distance V to the end machining surface along Z axis at the rapid traverse rate;
(8) Cut along Z axis for (Q+V) depth;
(9) Cycling the operation from (4) ~ (8) till the surface of total cutting is performed.
(10) Return to the initial plane or point R plane according to G98 or G99.

Command Path:

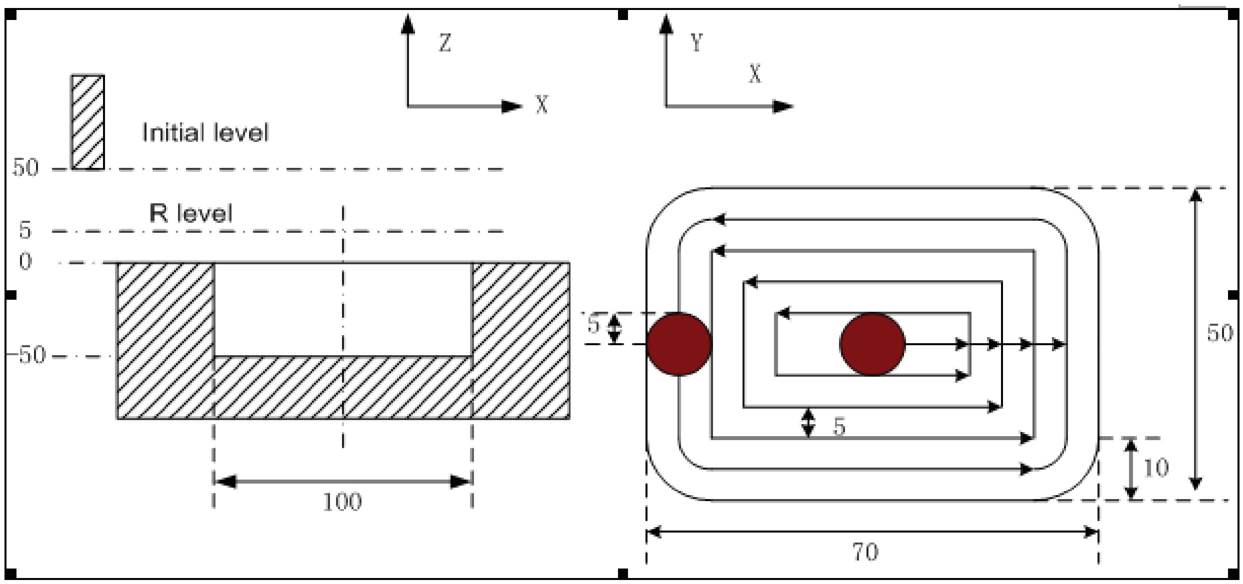




Related Explanation:

The commands P and □ are disabled in this cycle, but the P value is reserved as canned cycle modal value.

For example: An inside rectangle groove rough-milling is specified by G13□ in canned cycle, see the following figure:



G□0 G00 X50 Y50 Z50; (G00 rapid positioning)

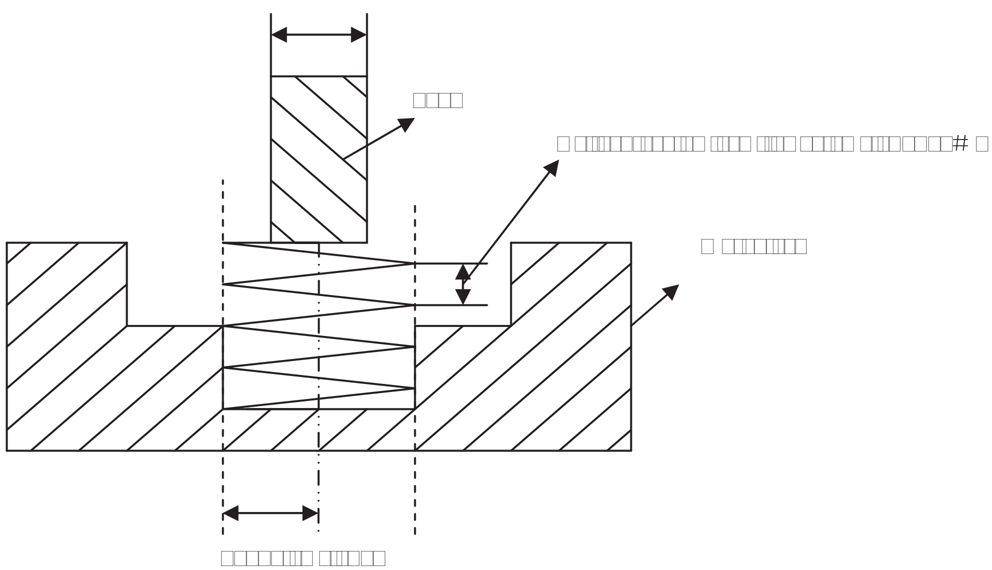
G□□ G13□ X25 Y25 R5 Z-50 I□0 □50 W20 Q10 □5 □10 □10 F800 □1; (Groove rough-milling cycle inside rectangle is performed □1□5)

G80 X50 Y50 Z50; (The canned cycle is cancelled, returning from the point R plane)

□30;

Note □f the □arameter value of □□# is set for more than 1□□the helical cutting feed along the Z a□is □ill be □erformed by G11□ and G111. □o□the □or□□iece □ithout groove can be machined by rough□milling directly.

The helical feeding path is as follows:



3.15.□.15 Rectangle groove inner fine□milling cycle G136□G13□

Format:

G136

G□□G□□ X_ Y_ R_ Z_ □_ □_ □_ □_ □_ F_;

G13□

Function: The tool performs fine-milling inside the rectangle with the specified width and direction, it is returned after finishing the fine-milling.

□□□lanation: For command explanation of canned cycle, see the table 13.1.□.

G13□: Finish-milling cycle inside groove of rectangle in □□W.

G13□: Finish-milling cycle inside groove of rectangle in □W.

I: The rectangle width along the X axis, the value range is indicated as 0□□□□□.□□□mm.

□: The rectangle width along the Y axis, the value range is indicated as 0□□□□□.□□□mm.

□: Se□uence number of tool radius, the value range is 0□32, the 0 is default value of □0. The current tool radius value is taken out according to the specified se□uence number.

□: The distance between the finish-milling start point and the rectangle side in X axis direction, the value range is indicated as 0□□□□□.□□□mm.

□: □orner arc radius; no corner arc transition if it is omitted. When the □ is omitted or it is e□ual to 0 and the tool radius is more than 0, the alarm is generated.

□ycle □rocess:

(1) Positioning to XY plane at the rapid traverse rate;

(2) □own to point R plane at the rapid traverse rate;

(3) □utting feed to the bottom of a hole;

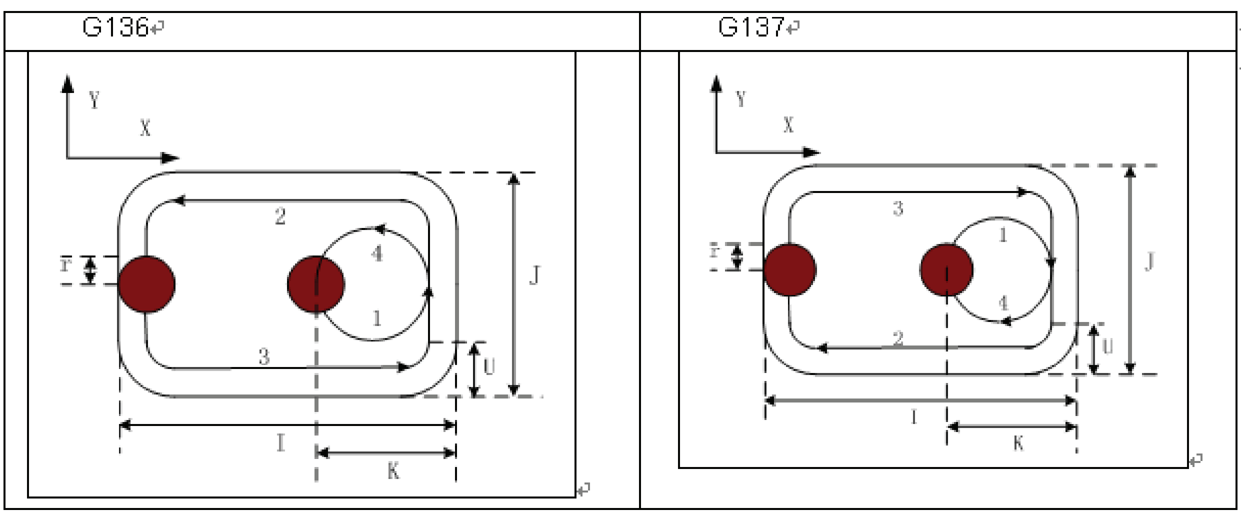
(□) Perform the circle interpolation by the path of transit arc 1;

(5) Perform the circular and linear interpolation by the path of 2-3-□-5-□;

(□) Perform circular interpolation by the path of transit arc □ and return to the start point;

(□) Returning to the initial plane or point R plane according to G□8 or G□□.

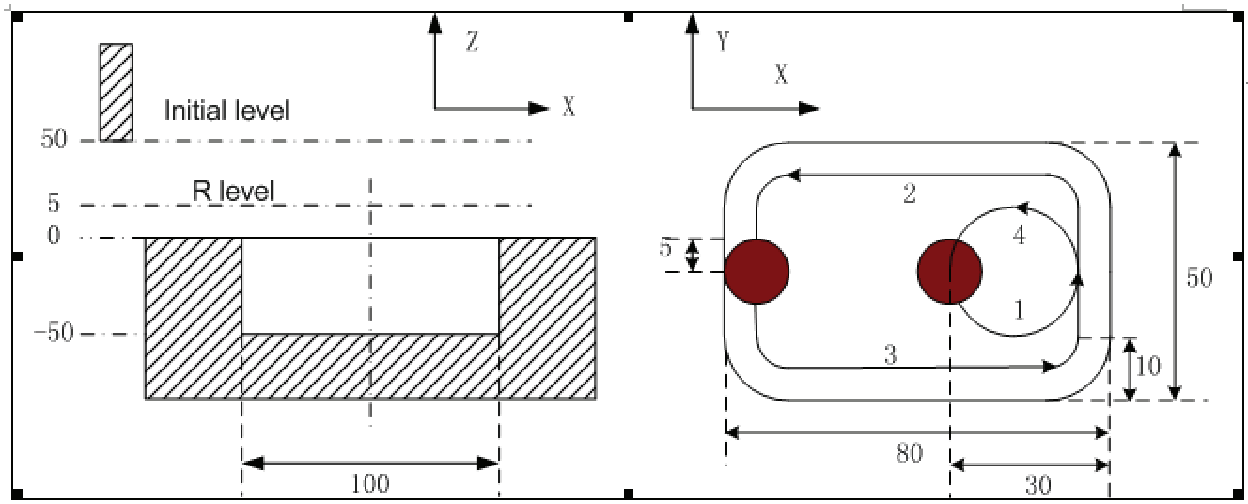
□ommand Path:



Related □□□lanation:

The commands Q, P and □ are disabled in this cycle, but the Q and P values are reserved as the canned cycle modal value.

For e□am□le: To perform a fine-milling for the finished rough-milling rectangle groove with the canned cycle G13□ command, see the following figure:



G□0 G00 X50 Y50 Z50; (G00 rapid positioning)

G13□ X25 Y25 R5 Z-50 I80 □50 □30 □10 F800 □1; (Perform finish-milling inside the rectangle groove at the bottom of a hole in the canned cycle □1□5)

G80 X50 Y50 Z50; (The canned cycle is cancelled, returning from the point R plane)

□30;

3.15.□.16 Finish□milling cycle outside the rectangle G13□G13□

Format:

	G13□	
G□□G□□		**X_ Y_ R_ Z_ □_ □_ □_ □_ □_ F_**
	G13□	

Function: The tool performs fine-milling outside the rectangle by the specified width and direction, it is returned after finishing the fine-milling.

□□□lanation:

G138: Finish-milling cycle outside the rectangle in □□W.

G13□: Finish-milling cycle outside the rectangle in □W.

I: The width of rectangle along the X axis, the value range is indicated as 0□□□□□.□□□mm.

□: The width of the rectangle along the Y axis, the value range is indicated as 0□□□□□.□□□mm.

□: Se□uence number of tool radius, its value range is indicated as 0 □32, thereinto, the 0 is default of □0. The current tool radius value is taken out according to the specified se□uence number.

□: The distance between the finish-milling start point and the side of rectangle along the X axis, the value range is indicated as 0□□□□□.□□□mm.

□: □orner arc radius, if it is omitted, no corner arc transition.

□ycle □rocess:

(1) Positioning to the XY plane at the rapid traverse rate;

(2) □own to the point R plane at the rapid traverse rate;

(3) □utting feed to the bottom of a hole;

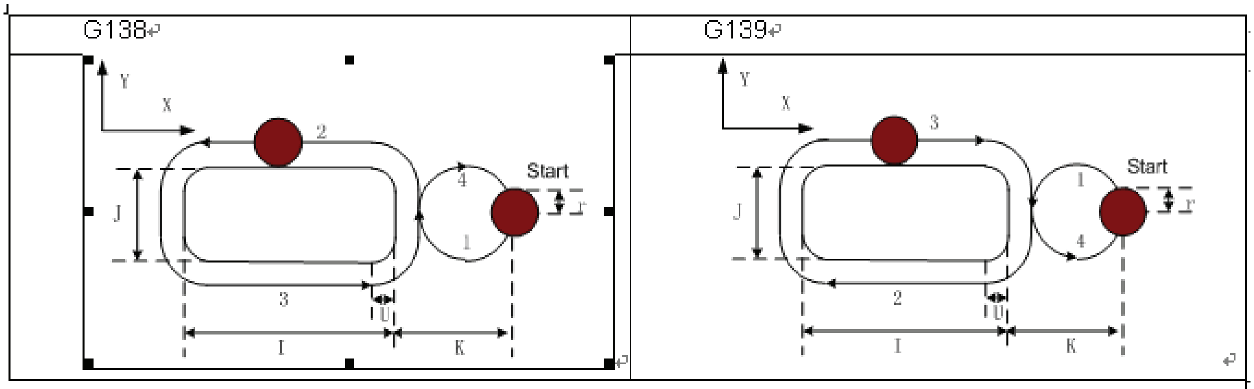
(□) Perform the circle interpolation by the path of transit arc 1;

(5) Perform the circular and linear interpolation by the path of 2-3-□-5-□;

(□) Perform circular interpolation by the path of transit arc □and return to the start point;

(□) Returning to the initial plane or point R plane according to G□8 or G□□.

□ommand Path:

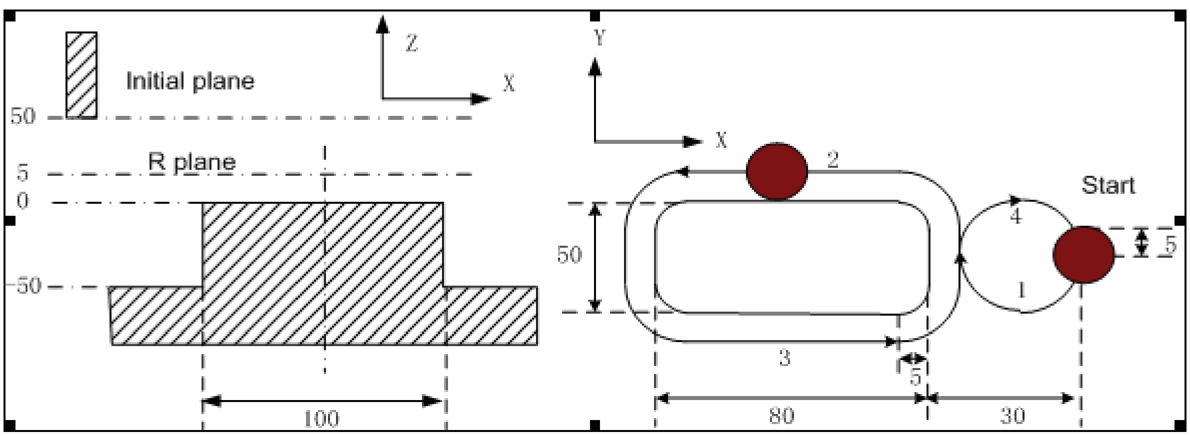


Related □□□lanation:

(1) The interpolation direction of transition arc is inconsistent to that of the fine-milling arc when a fine-milling is performed outside the rectangle. The interpolation direction is the one for the fine-milling arc in the command explanation.

(2) The commands Q, P and □are disabled in this cycle, but, the value of Q and P are reserved as canned cycle modal value.

For e□am□le: A finished rough-milling rectangle groove is performed by the fine-milling by the command G138 in canned cycle. See the following figure.



G□0 G00 X50 Y50 Z50; (G00 rapid positioning)

G□□ G138 X25 Y25 R5 Z-50 180 □50 □30 □5 F800 □1; (The rectangle outside finish milling is performed under the canned cycle at the bottom of a hole □1□5)

G80 X50 Y50 Z50; (The canned cycle is cancelled, it returns from the point R plane)

□30;

3.15.3 □ontinous □rilling

□ontinuous e□ual interval drilling cycle is performed in the way that canned cycle is called according to the specified linear, rectangular or arc path.

Parameters related to continuous drilling

0	1	5		□PT□	RPT□		BR□□	□□□	□□□	□□□	□□□

□PT□ □1：□ocating with G01 in line interval drill；

□0：□ocating with G00 in line interval drill；

RPT□ □1: □ocating with G01 in circle and rectangle interval drill；

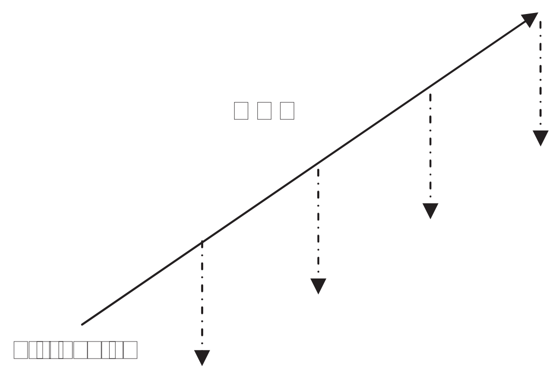
□0：□ocating with G00 in circle and rectangle interval drill；

BR□□ □1：the return plane when continuous drilling is selected by G□8, G□□.

□0：the return plane when continuous drilling is selected by G□□.

3.15.3.1 Line series □unch □L function□

□ holes machining cycle should be performed from current plane position to end point specified by X and Y are indicated if the □ word is specified in canned cycle, so the current position (block start and end) will not be drilled, the end point position is regarded as the last hole, holes are e□ual-spaced, as follows:



□value setting	System execution result
□alue is negative	Ineffective, the value should be positive
The value is unspecified or e□uals to 1	Normal drilling cycle 1 time
The value is 0	No change of axes, the system reserves relevant cycle modal data
The value is decimal	When □>1,using round number When □<1, it is processed as □□0, not moving but reserving its modal data and relevant cycle parameter values.

Note 1：the ma□imum in□ut value of command L is □□□□□.□□□～□□□□.□□□; □ecimals is ignored and absolute value is used instead of negative value. L code is effective only in current bloc□.

Note □：□n continuous drilling□the return □lanes are R □oint □lan. □fter the last hole is □rocessed□the return □lane is s□ecified by G□□□G□□.

Note 3：□ hen there is no a□is □osition command in the s□ecified L bloc□□it means drilling cycle is □erformed L times in the original □lace.

Note □：□anned cycle command G11□□G111□G11□□G113□G11□□G115□G13□□G135□G136□G13□□ G13□□G13□ has no continuous drilling function.

Note 5：□ hen L□ is s□ecified□no drilling □ill be □erformed.

3.15.3.□Rectangle series □unch □G1□□G1□1□

Format:

G1□□
G□□G□□ G□□ X_ Y_ R_ Z_ □_ □_ □_ F_
G1□1

Function: Performing series punch on each side of the rectangle according to the punch number specified.

□□□lanation:

G1□0 □Punching in □W
G1□1 □Punching in □□W
Gxx □Punching type (G□3, G□□, G81, G83, G8□, G85, G8□, G88, G8□)
X, Y □End coordinate of the first rectangle side
R □R plane position
Z □ □ole depth
A □The punching number on the 1st and 3rd side
B □The punching number on the 2nd and □th side
□- The length of the 2nd side
F □ □utting feedrate

Related Parameter:

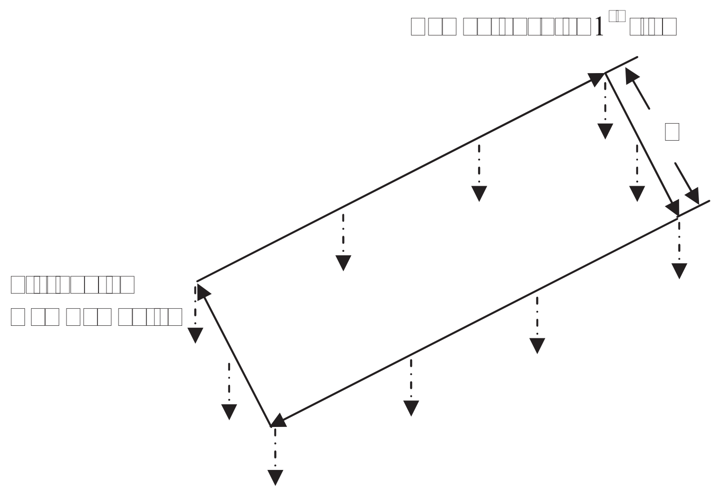
Bit □of the parameter 01□
1: □ole positioning of serial punching is performed by cutting path (G01□G03).
0: □ole positioning of serial punching is performed by the rapid traverse path (G00).

For example:

The end point coordinate of the rectangle first side is X□0, Y□0; the length of the 2nd side is 20mm as for the rectangle path punching. The punching holes are machined by G81, to punch 3 holes at 1st and 3rd side each other; punch 2 holes at 2nd and □th side each other, the hole depth is 25mm;

□ts □rogramming is as follo□s:

G□□ G1□ G□ X□ Y□ Z□5;
□ □3;
G1□□ G□1 X□□ Y□□ R5 Z□□5 □3 □□ □□□ F□□□;
G□□ G□ X1□□ Y1□□ □ □5;
□3□



There are 10 holes such as A1□A3, B□, B5, A□□A8, B□ and B10 to be machined as in above figure.

Note 1: □f the G1□□ or G1□1 is s□ecified in the canned cycle□it is indicated that the rectangle serial □unching □ill be □erformed. The rectangle data are defined according to s□ecified X□ Y coordinates and □ value in a □rogram□and the serial □unching cycle is □erformed

according to the □unch mode □canned cycle command□.

Note □: The command value of ma□imum □unching number □ and □ at each side is □□□□; the command is disabled □hen it is negative. The decimal □art □ill be rounded off if the command is decimal; if the □ or □ is not s□ecified□then □is a default.

Note 3: The rectangle is defined by the current start □oint□the end of the 1st side and the length of the □nd side; the default is current start □oint if the end of 1st side is not s□ecified; the alarm □ill be generated if the length □namely□the □ is not s□ecified□of □nd side is not s□ecified.

Note □: The returned levels are all R □oint □lane in serial □unching□the corres□onding □lane □ill be retracted according to G□□□G□□ s□ecified in a bloc□ □hen the last hole is □erformed.

Note 5: □anned cycles□such as G11□□G111□G11□□G113□G11□□G115□G13□□G136□G13□□G13□ and G13□ have no serial □unching functions.

Note 6: The command □ords G1□□□G1□1□□□□ and □ are only effective in current bloc□. The alarm □ill be generated if the G1□□ and G1□1 are s□ecified □ithout the canned cycle □□unching□. The □□□ and □ □ill be ignored if □□□ and □ are s□ecified instead of the G1□□ or G1□1.

3.15.3.3 □rc serial □unching □G1□□□G1□3□

Format:

G1□□

G□□□G□□ G□□ X_ Y_ R_ Z_ □_ □□_ □_□ □_ F_

G1□3

Function: Serial punching is performed according to the specified punching number on specified arc.

□□□lanation:

G1□□ □Punching in □□

G1□3 □Punching in □□□

G□□ □Punching ty□e（G□3□G□□□G□1□G□□□G□3□G□□□G□5□G□6□G□□□G□□）

X□Y □ □nd □oint coordinate for the arc□it is fi□ed for G1□ □lane.

R □R □lane □osition

Z □ □ole de□th

□ □Radius of arc□□hen a negative value is s□ecified□it is ma□or arc.

□_ □_□□The circle center and radius are calculated by □or □ □hen the R value is not s□ecified.

□ □Number of □unching

F □ □utting feedrate

Related Parameter:

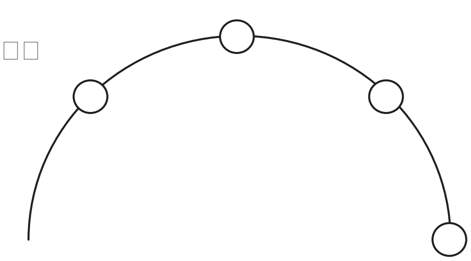
Bit □ of the parameter 01□

1: □ole positioning for serial punching is performed by cutting path (G01□G03).

0: □ole positioning for serial punching is performed by the rapid traverse path (G00).

For example:

G□1 G1□2 G81 X100 R50 Z-50 □□



Example 2：when drilling □holes in full circle, the start points and end points are coordinate origins, and the radius is 50, hole depth is 50.

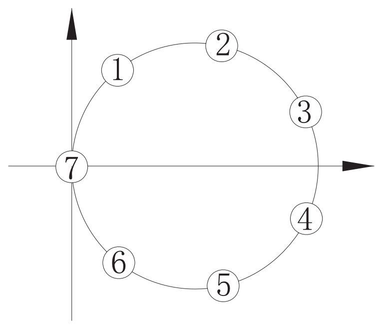
O0001；

G00 G□0 X0 Y0 Z0 G1□；

G□8 G1□2 G82 I50 □0 R-10 Z-50 □□F3000；

□30；

□



Note 1: □n continuous drilling□□hen the start □oint is identical to end □oint□no drilling □ill be □erformed.

Note □: □anned cycle G11□□G111□G11□□G113□G11□□G115□G13□□G135□G136□G13□□G13□□ G13□ has no continuous drilling function.

Note 3: The ma□imum drilling number □ is □□□□; the negative value is □rocessed as absolute value; the decimals are rounded.

Note □: □hen □ is not s□ecified or e□uals to □□it reaches the end □oint directly and no drilling □ill be □erformed.

3.15.□□autions for canned cycle

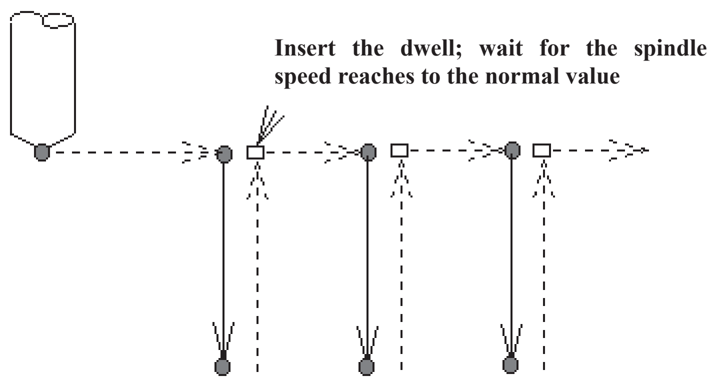
(1) The spindle should be rotated (The □ code should be correctly specified, or, the alarm will be generated, the G□□ by □0□, G8□ by □03) by using the miscellaneous function (□ code) before the canned cycle is executed.

(2) Specifying any command of the X, Y, Z and R data, the hole machining can be performed in the canned cycle of G□3□G8□. If neither data is contained in the block, the hole machining is not performed (G110, G111, G112, G113, G11□, G115, G13□, G135, G13□, G13□, G138 and G13□ are still needed to specify the corresponding address I, □and □, or the alarm occurs). But the hole machining is not performed when the G0□X□is specified in the circumstance of X, because the X indicates for time when the G0□is specified.

G00 X□;	(G00 rapid positioning)
G81 X□Y□Z□R□F□□□;	(□ole machining performs)
;	(Without hole machining)
F□;	(F value is refreshed without the hole machining)
□□;	(Performing the miscellaneous function only)

(3) When the canned cycle (G□□or G8□) is employed in spindle rotation consolation, if the hole

position (X, Y) or distance from initial point level to the point R plane is short, and it is necessary to machine serially, or sometimes the spindle can not reach the specified speed before the hole machining operation, for delaying the time, the dwell block by G0□ is inserted into each hole machining, which is shown as follows:



G8□X□Y□Z□R□F□ ;
G0□P □; (For dwell time P, without hole machining)
X□Y□; (The next hole is machined)
G0□P □; (For dwell time P, without hole machining)
X□Y□; (The next hole is machined)
G0□P□; (For dwell time P, without hole machining)

Sometimes, this issue will not be considered according to different machine tool, refer to the manual supplied by the machine tool builder.

(□) As stated above, the canned cycle can also be cancelled only when G00□G03 codes are read. So, there are two cases (□ expresses for 0□3, □□ for canned cycle code) will be shown when they share the same block with the canned cycle G code.

G□G□□ X- Y- Z- R- Q- P- F- □-; (For canned cycle)

G□□ G□X- Y- Z- R- Q- P- F- □-; The X, Y and Z axes are moved by G□, the R, P, Q and □ are disabled, the F is stored. The principle, which the last G code is effective when G codes of same group share the same block, is met by cases above.

(5) When the canned cycle and miscellaneous function are specified at the same block, The □ and □F codes are delivered at the beginning of positioning (see the Fig.13.1 (A) for the operation 1). The next hole machining can be performed till the ending signal (FIN) occurs.

(□) When the canned cycle is applied, if the tool compensation □ is current state, the tool compensation information □ is then temporarily cancelled and saved; the tool compensation □ status is restored when the canned cycle is cancelled.

(□) If the tool length offset commands (G□3, G□□ and G□□) are specified in a canned cycle block. Then, the offset is performed when the point R plane is positioned (operation 2). The tool length offset commands are disabled after the canned cycle is entered till it is cancelled.

(8) The cautions for the operation of canned cycle:

a. Single block

When the canned cycle operation is performed by using the single block mode, normally, it is separately stopped at the terminal of the movements 1, 2, 3, 4, 5 and 6 in the Fig. 13.1 (A). And the single block is somewhat different according to corresponding canned cycle action at the bottom of a hole. For example, the single block is stopped when the dwell is applied. The operation at the bottom of the hole for fine-milling and rough-milling are divided into multiple single stop. So, it is necessary to startup for several times to machine a hole in a single block.

b. Feed hold

The feed hold is disabled between the movement 3 ~ 5 in commands G74 and G84, but the indicator of feed hold will light up. But the control stops till the operation 6. If the feed hold is performed again in operation 6, then it is stopped immediately.

c. Override

The feedrate override is considered for 100 percent in the operation G74 and G84, the override change is disabled.

(9) When the bit 1 of parameter 3 (D_R) is set to 1, the D value in tool compensation page indicates diameter value.

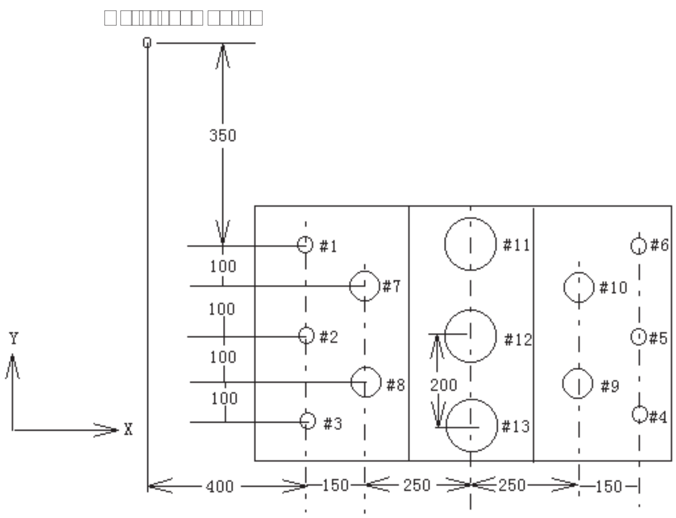
3.15.5 Examples for modal data specified in canned cycle

No.	Data Specification	Explanation
N0010	G00 X_ M3 ；	G00 positioning at the rapid traverse, and rotating the spindle;
N0020	G81 X_ Y_ Z_ R_ F_；	Because it is the beginning for the canned cycle, so the value needs to be specified for Z, R and F.
N0030	Y_；	The corresponding hole machining data is same to the previous hole, only the position Y is different, so G81Z_R_F_ can be omitted. As for the hole position is shifted for Y, hole machining is performed further by using the G81;
N0040	G82 X_ P_；	The hole position needs to be moved along the X axis as for the pervious one. The Z, R and F of previous hole and the P specified by this hole are taken as hole machining data by the G82;
N0050	G80 X_ Y_ M5 ；	The hole machining is not executed, all of the hole machining data are cancelled (except for the F); The GO positioning is performed with XY;
N0060	G85 X_ Z_ R_ P_；	The Z and R are needed to be specified newly because all of the data in previous block are cancelled, the above value specified is applied when the F is omitted. Although the P value is commanded, but it is not needed for this hole machining, so the P value is saved.
N0070	X_ Z_；	The Z is different compared with the previous hole, and the hole position just moves along the X axis;
N0080	G89 X_ Y_ D_；	The Z and R, P values separately specified by N0070 and N0060, the F value specified in N0020 are taken as hole machining data, which are used for G89 hole machining.
N0090	G112 I_ J_ F_ D_；	The fine-milling hole machined by G89 is performed by G112.
N0100	G0 X_ Y_ Z_；	positioning for a rectangle machining

N0110	G134 Z_R_I_J_K_U_D_；	Start machining the rectangle;
N0120	Y_I_J_K_U_D_；	Begins machining the second rectangle;
N0130	X_ Y_ I_J_K_U_D_；	Begins machining the 3rd rectangle;
N0140	G138 X_ Y_ R_ Z_ I_	The fine-milling inside the machined rectangle groove is to be performed, the corresponding data are needed;
	J_ K_ U_ D_ F_；	
N0150	G01 X_ Y_，	Cancel the hole machining mode and data (except for F); the G01 cutting feed is performed by XY.

□ot□□□□□r□□ □□□□ an□ □ o□cann□□ c□cl□ □□1□□□111□□11□□□113□□11□□□115□□13□□□135□ □13□□□13□□□13□ and □13□ are not sa□ed as canned cycle modal data□so t□e I□□ and □ □alues need to □e specified in eac□ □loc□□or t□e alarm □ill □e generated.

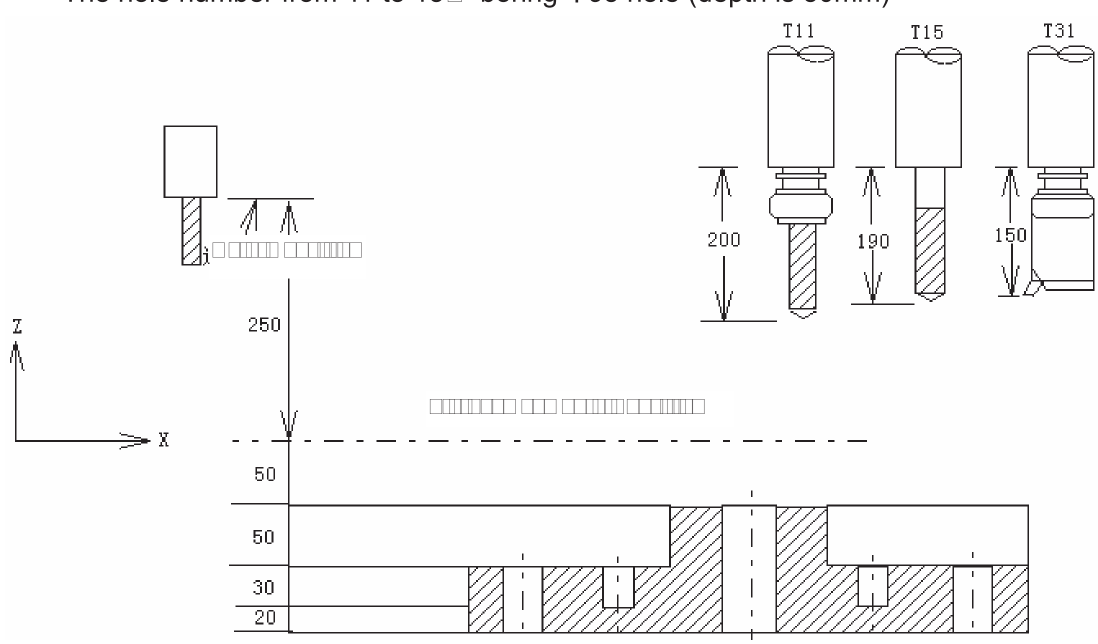
3.15.□ Examples for canned cycle and tool lengt□ compensation



□h□ h□□ n□m□□□□□m □□ □□ □□in□ Φ□0

The hole number from 7 to 10… drilling Φ20

The hole number from 11 to 13□ boring Φ95 hole (depth is 50mm)



The values of offset numbers □11, □15 and □ 31 are separately set to 200.0, 190.0 and 150.0, the program is as following:

N001 G92 X0 Y0 Z0 ；	The coordinate system is set at the reference point
N002 G90 G00 Z250.0 ；	
N003 G43 Z0 □11 ；	Plane tool length compensation is performed at the initial plane.
N004 S30 M3 ；	The spindle starts.
N005 G99 G81 X400.0 Y-350.0 ； Z-153.0 R-97.0 F120.0 ；	□1 hole is machined after positioning.
N006 Y-550.0 ；	□2 hole is machined after positioning, point R plane returned.
N007 G98 Y-750.0 ；	□3 hole is machined after positioning, initial plane returned.
N008 G99 X1200.0 ；	□4 hole is machined after positioning, point R plane returned.
N009 Y-550.0 ；	□5 hole is machined after positioning, point R plane returned.
N010 G98 Y-350.0 ；	□6 hole is machined after positioning, initial plane returned
N011 G00 X0 Y0 M5 ；	Reference point return, the spindle stops.
N012 G49 Z250.0 ；	Tool length compensation cancellation
□□□□ □□□ □□ □□□ ；	Initial plane, tool length compensation.
N014 S20 M3 ；	Spindle starts
N015 G99 G82 X550.0 Y-450.0 ； Z-130.0 R-97.0 P30 F70 ；	□7 hole is machined after positioning, point R plane returned.
N016 G98 Y-650.0 ；	□8 hole is machined after positioning, initial plane returned.
N017 G99 X1050.0 ；	□9 hole is machined after positioning, point R plane returned.
N018 G98 Y-450.0 ；	□10 hole is machined after positioning, initial plane returned.
N019 G00 X0 Y0 M5 ；	Reference point return, the spindle stops.
N020 G49 Z250.0 ；	Tool length compensation cancellation.
□□□□ □□□ □□ □□□ ；	Tool length compensation at initial plane.
N022 S10 M3 ；	Spindle starts.
N023 G85 G99 X800.0 Y-350.0 ； Z-153.0 R47.0 F50 ；	□11 hole is machined after positioning, point R plane returned.
N024 G91 Y-200.0 ； Y-200.0 ；	□12 and □13 are machined after positioning, point R plane returned.
N025 G00 G90 X0 Y0 M5 ；	Reference point return, the spindle stops.
N026 G49 Z0 ；	Tool length compensation cancellation
N027 M30 ；	Program stops.

3.16 Absolute and Incremental Commands G90 and G91

□ormat□

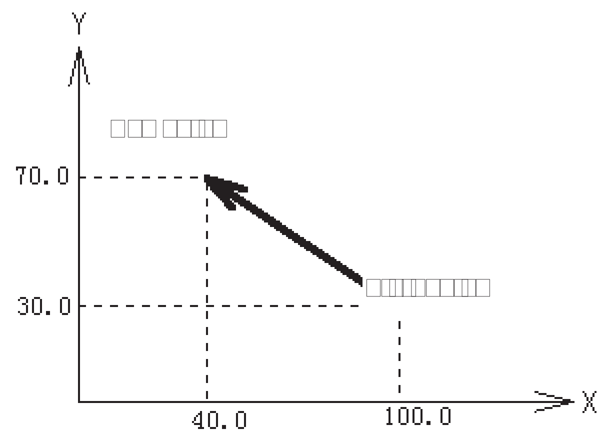
G90; Absolute command

G91; Incremental command

□unction□

There are two kinds of modes for commanding axis offset, one is absolute command the other is incremental command. The absolute command is programmed by coordinate value of the terminal position by the axis movement. The incremental command is directly programmed by the movement value of the axis. They are separately specified by G90 and G91 commands.

Example□



The above movement is programmed by absolute and incremental commands, which is as follows:

G90 X40.0 Y70.0 ; or G91 X－60.0 Y40.0;

3.17 Workpiece Coordinate System Setting G92

□unction□ The workpiece coordinate system is set by setting the absolute coordinate in current position in the system (It is also called floating coordinate system). After the workpiece coordinate is set, the coordinate value is input in absolute programming in this coordinate system till the new workpiece coordinate system is set by G92.

□ommand explanation□ G92, which is a non-modal G-command;

X: The new X axis absolute coordinate of current position;

Y: The new Y axis absolute coordinate of current position;

Z: The new Z axis absolute coordinate of current position;

Note□In □□□ command□current coordinate □alue □ill □e not c□anged if t□e □□□ and □ are not input□t□e program □ero is set □y t□e current coordinate □alue. □ □en t□e □□□ or □ is not input□t□e coordinate axis not input □eeps on t□e original set □alue.

3.18 Feed per min. G94, Feed per rev. G95

□ormat□ G94 Fxxxx; □F□□□□~ F□□□□□the leading □ero can be omitted□the feedrate per min. is offered, mm□min.)

Function: The cutting feedrate is offered in mm/min unit when the G94 is modal G command. The G94 can be omitted if the current mode is G94.

Format: G95 F<u>xxxx</u>; (F0.0001～500, The leading zero can be omitted)

Command Function: The cutting feedrate is offered in mm/rev unit when the G95 is modal G command. The G95 can be omitted if the current mode is G95. The product of F command value (mm/r) and current spindle speed(r/min) is regarded as the command cutting feedrate to control the actual feedrate when the G95 F<u>xxxx</u> is performed by system. The actual cutting feedrate varies with the spindle speed. The spindle cutting feed value per rev is specified by G95 F<u>xxxx</u>, it can form even cutting grain on the surface of the workpiece. The machine should be installed spindle encoder when the G95 mode is used.

G94 and G95 are modal G commands in same group, one of them is effective in one time. G94 is initial modal G command, it is defaulted effective when the power is turned on.

The conversion formula for feed value per rev and per min is as following:

$$F_m = F_r \times S$$

Thereinto: F_m: Feed value per min (mm/min);

F_r: Feed value per rev per rev (mm/r);

S: Spindle speed (r/min).

The feedrate value is set by system data parameter No.030 when the power is turned on for the system; an F value is invariable after the F command is performed. The feedrate is 0 after the F0 is executed. The F value is invariable when the system is reset or emergency stop. **The feed override is memorized when the power is turned off.**

Related parameter:

System data parameter No.029: the exponential acceleration or deceleration time constant for cutting and manual feed;

System data parameter No.030: the lower value of exponential acceleration or deceleration on cutting feed;

System data parameter No.031: The upper limit value for cutting feedrate (X, Y and Z axes)

Note:

The cutting feedrate becomes uneven when the spindle speed is less than 1 rev/min in G95 mode; the actual feedrate has following error when the spindle speed fluctuates. In order to guarantee the machining quality, it is recommended that the spindle speed can not be lower than spindle servo or the lowest speed of effective torque introduced by inverter during machining.

3.19 G98, G99

Format:

G98;

G99;

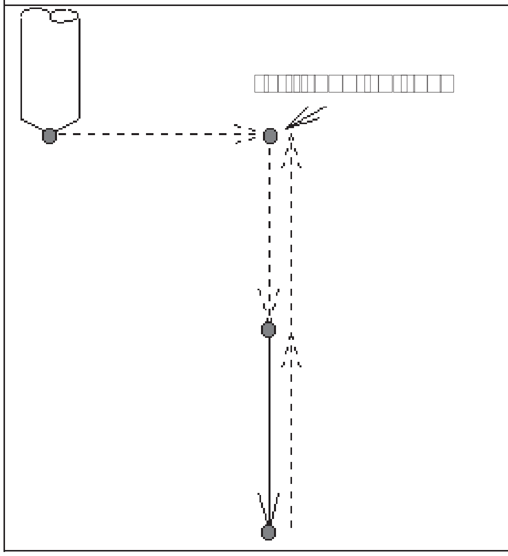
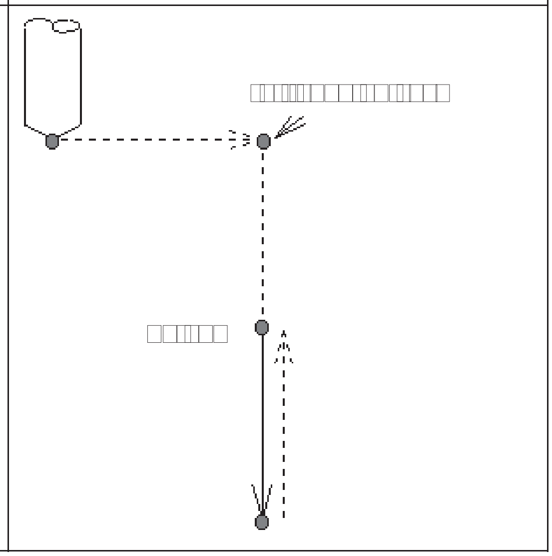
Function:

G98; Tool returns to the initial plane when the hole machining is returning.

G99; Tool returns to the point R plane when the hole machining is returning.

Explanation:

Modal G command

G98 (Return to initial plane)	G99 (Return to point R plane)
	

Refer to the explanation for canned cycle command.

3.20 Chamfering Function

A straight line or an arc is inserted into two figures; this is called Chamfering function. The tool can be smoothly transferred from one figure to another. GSK980MD owns two chamfering functions, one is linear chamfering, and the other is arc chamfering.

3.□□.1 □inear c□amfering

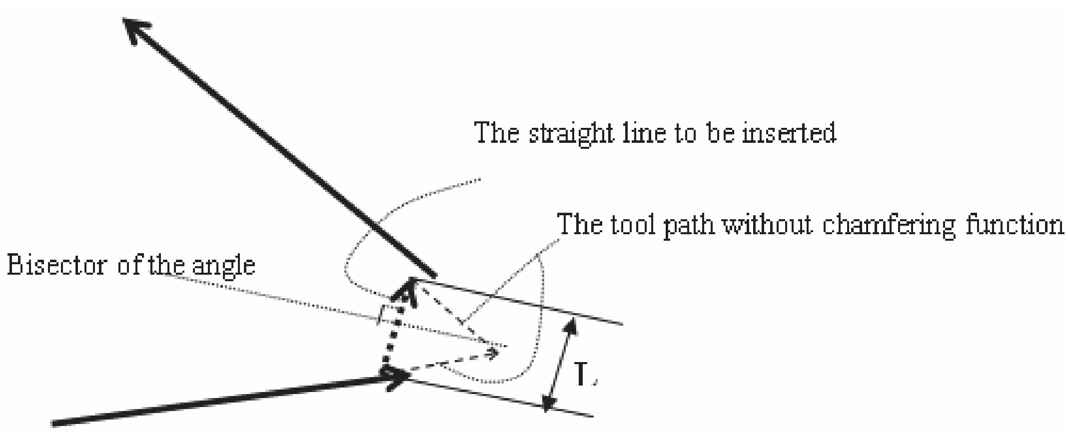
The linear chamfering is that a straight line is inserted between figures of the straight lines, the arcs, as well as the straight line and arc. The command address for linear chamfering is □. The data followed by command address □ is the length of chamfering straight line. The linear chamfering should be employed in the G01, G02 or G03 command.

- **□inear to linear**

  □ormat□ G01 IP_ □_; (IP is axis movement command)

  G01 IP_；

  □unction□ A straight line is inserted into interpolation between 2 straight lines.

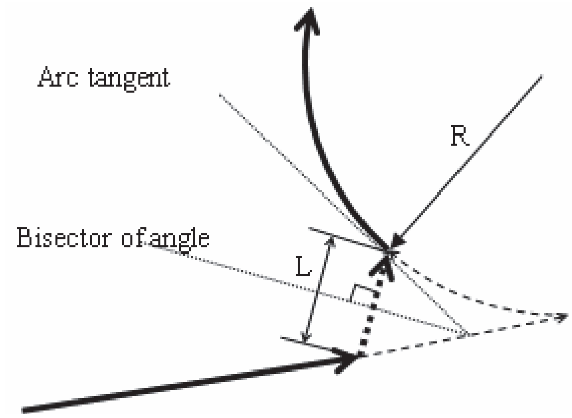


- **□inear to circular**

□ormat□ G01 IP_ □_；

G02□G03 IP_ R_(I_ J_ K_)；

□unction□ A straight line is inserted between straight line and arc interpolation.



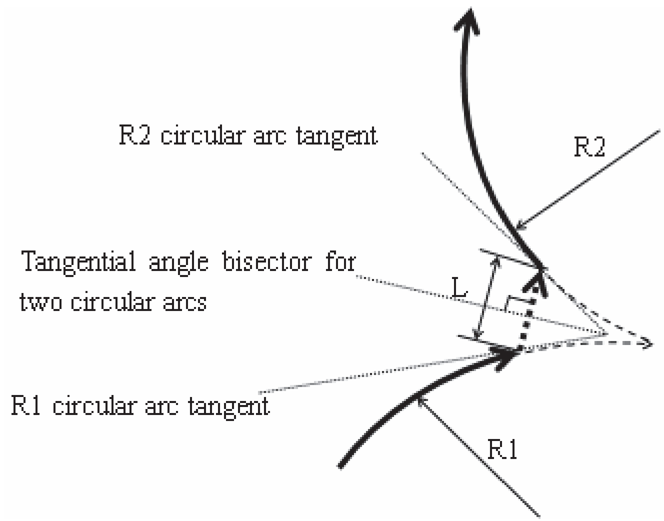
- **□ircular to circular**

□ormat□

G02□G03 IP_ R_ (I_ J_ K_) □_；

G02□G03 IP_ R_(I_ J_ K_)；

□unction□ A straight line is inserted between two arc interpolations.

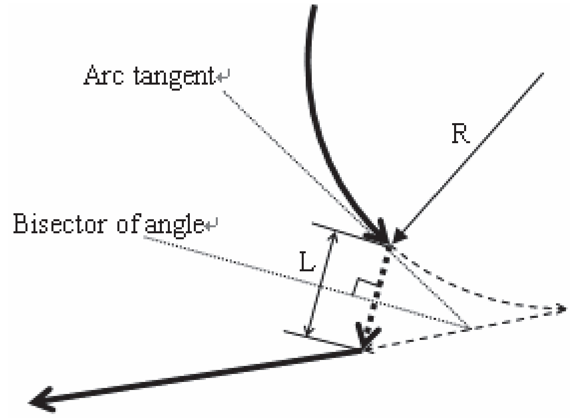


- **□ircular to linear**

□ormat□ G02□G03 IP_ R_(I_ J_ K_) □_；

G01 IP_；

□unction□ A straight line is inserted between the arc and linear interpolation.



3.□□.□ □ircular c□amfering

An arc is inserted between the two linear figures, arc figures or linear and arc figures, this is called circular chamfering. Tangent transition is performed between arc and figure line. The command address is C for the arc chamfering, the data followed by command address C is the radius of chamfering arc. The arc chamfering should be employed in command G01, G02 or G03.

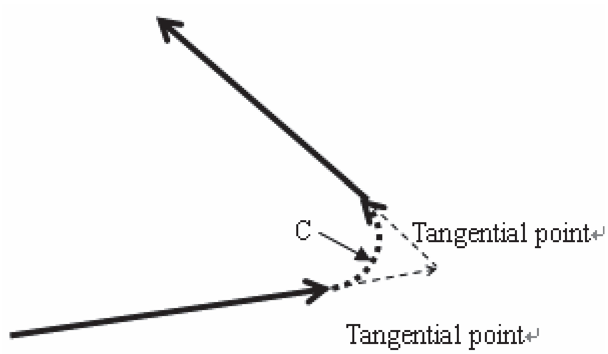
- **1. □inear to linear**

  □ormat□

  G01 IP_ C_；

  G01 IP_；

  □unction□ An arc is inserted between two linear interpolations, which it is tangential with two linear lines, the data followed by command address C is radius.



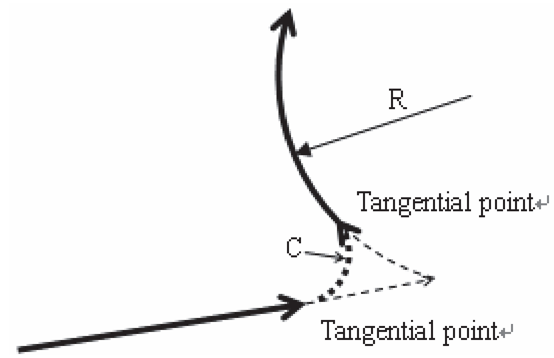
- **□. □inear to □ircular**

  □ormat□

  G01 IP_ C_；

  G02□G03 IP_ R_(I_ J_ K_)；

  □unction□ An arc is inserted at the intersection of straight line and arc, this arc is tangential with both the straight line and arc, the data followed by command address C is radius.



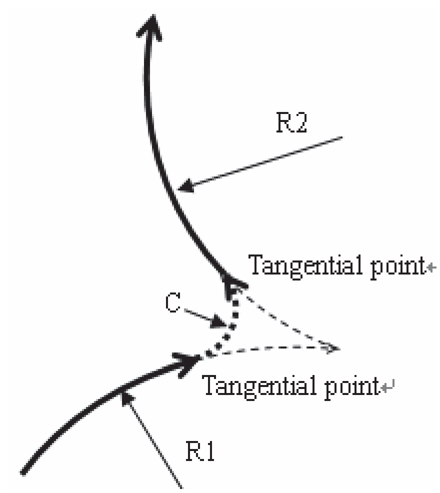
- **3. □ircular to □ircular**

□ormat□

G02□G03 IP_ R_(I_ J_ K_) C_;

G02□G03 IP_ R_(I_ J_ K_);

□unction□ An arc is inserted between two arc interpolations which it is tangential with two circulars, the data followed by the command address C is radius.



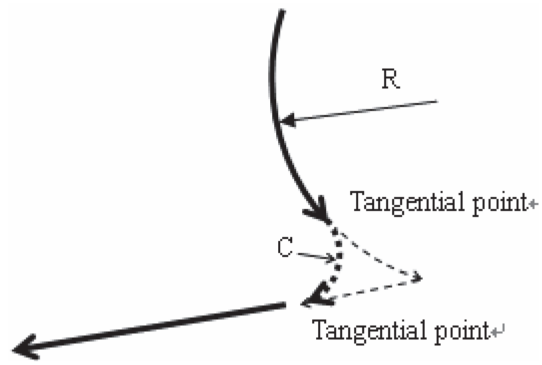
- **□. □ircular to □inear**

□ormat□

G02□G03 IP_ R_(I_ J_ K_) C_;

G01 IP_;

□unction□ An arc is inserted at the intersection of arc and straight line, which is tangential with the
arc and straight line; the data following the command address C is radius.



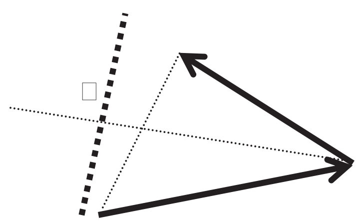
3.□□.3 Exceptional □ases

The chamfering function is ineffective or alarm is issued in the following circumstances:

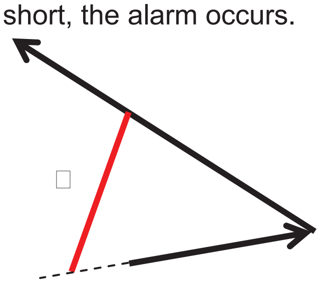
1. □inear c□amfering

□. The chamfering function is ineffective when two interpolation lines is shown on the same line.

□. If the chamfering linear length is too long, and the CNC alarm occurs.



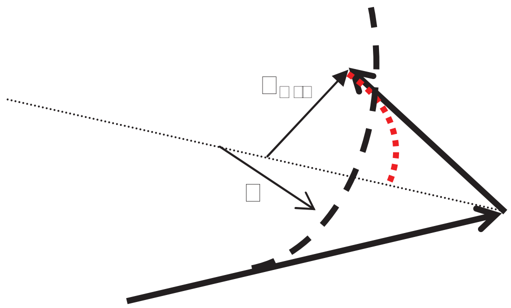
□. If some line (arc) is too short, the alarm occurs.



□. □rc □□amfering

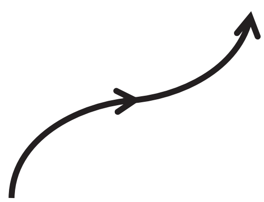
□. The arc chamfering function is disabled when two interpolation lines are shown on the same line.

□. If the chamfering radius is excessive, the CNC alarm occurs.



□. The arc chamfering function is disabled when the line is tangential with arc or the arc is tangential with line.

D. The arc chamfering function is disabled when the arcs are tangent.



Note 1□□□e c□amfering function can □e performed only in t□e plane specified □y □1□□□1□ or □1□□t□ese functions can not □e performed in parallel axes.
Note □□□□anging t□e coordinate system □y □□□ or □5□ to □5□□or□t□e □loc□ follo□ed □y performing t□e reference point return from □□□to □3□can not specify t□e c□amfering.
Note 3□□□amfering function can not □e employed in t□e DN□ mode.

3.21 Rigid Tapping

The right-handed tapping cycle (G84) and left-handed tapping cycle (G74) may be performed in standard mode or rigid tapping mode. In standard mode, the spindle is rotated and stopped along with a movement along the tapping axis using miscellaneous functions M03 (rotating the spindle cloclwise), M04 (rotating the spindle counterclockwise), and M05 (stopping the spindle) to perform tapping.

In rigid mode, tapping is performed by controlling the spindle motor as if it were a servo motor and by interpolating between the tapping axis and spindle. When tapping is performed in rigid mode, the spindle rotates one turn every time a certain feed (thread lead) which takes place along the tapping axis. This operation does not vary even during accleration or deceleration.

3.□1.1 □igid □apping

□ode format□

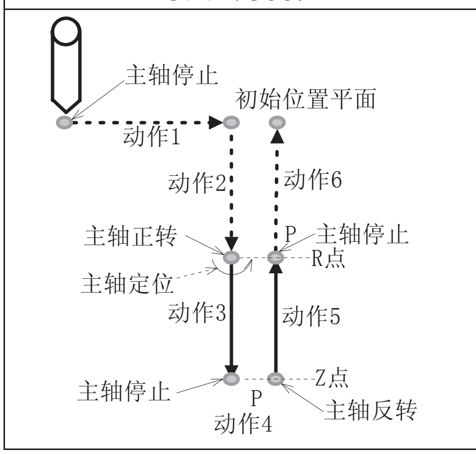
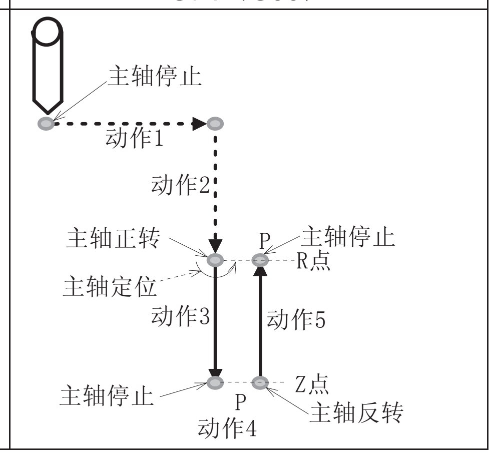
□eft-handed rigid tapping: G74 X_ Y_ Z_ R_ P_ F（I）_ □_ C_

Right-handed rigid tapping: G84 X_ Y_ Z_ R_ P_ F（I）_ □_ C_

□ode function□In rigid mode, tapping is performed by controlling the spindle motor as if it were a servo motor and by interpolating between the tapping axis and spindle. When tapping is performed in rigid mode, the spindle rotates one turn every time a certain feed (thread lead) which takes place along the tapping axis. This operation does not vary even during accleration or deceleration.

□ycle process□(1) Position to the XY plane at the rapid traverse rate;
(2) Reduce to the point R plane rapidly, then to the position where the C is specified at the rapid traverse rate;
(3) Tapping is performed to the bottom of the hole, then the spindle stops;
(4) Dwell time P is performed if the P is specified;
(5) Spindle rotates reversely returns to the point R plane, the spindle then stops; dwell time P is performed if the P is specified;
(6) Return to the origin plane if the command is G98;

□ode pat□□□ □□s□o□s a sample□

<table>
<tr><th>G74（G98）</th><th>G74（G99）</th></tr>
<tr><td>
</td><td>
</td></tr>
</table>

Explanations□

When the tapping operation 3 is being performed, the feedrate override can not be adjusted; when the operation 5 is perfoming, the speed override value is set by the data parameter 084, when the data parameter 084 is set to 0, the override value is fixed as 100□

When the tapping operation 3 is being performed, the linear acceleration or deceleration constant value is set by the data parameter 082; when the tapping operation 5 is performed, the linear acceleration constant value is set by data parameter 083, if the data parameter 083 is se to 0, the linear acceleration□deceleration time constant in operation 5 is set by the data parameter 082.

3.□1.□Pec□ □igid □apping

□ode format□

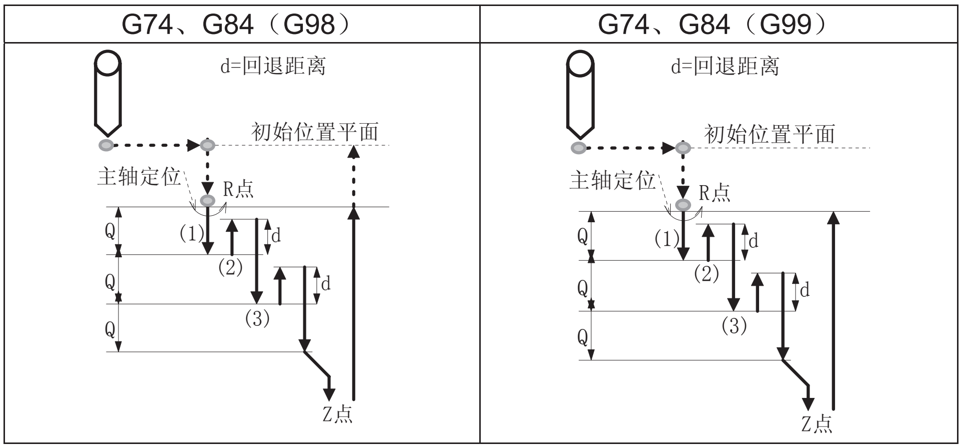
(□igh-speed□standard) peck left-handed rigid tapping: G74 X_ Y_ Z_ R_ P_ F（I）_ □_ □_ C_
(□igh-speed□standard) peck right-handed rigid tapping: G84 X_ Y_ Z_ R_ P_ F（I）_ □_ □_ C_

□ode function□When the peck tapping is performed in rigid tapping, due to chips sticking to the tool or increased cutting resistance, in such cases, the preferable tapping can be performed by the peck rigid tapping.

□ig□□speed pec□ rigid tapping□

When the RTPCP of state parameter No.025 is set to 1, the high-speed peck rigid tapping cycle is selected.

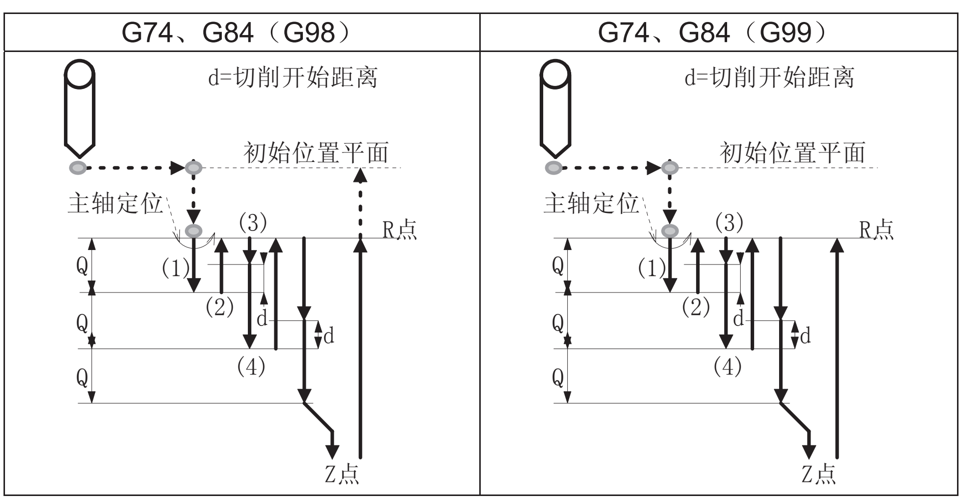
After positioning along the X- and Y-axes, rapid traverse is performed to point R, then position to the place where specifies by C. From point R, cutting is performed with depth □ (depth of cut for each cutting feed), then the tool is retracted by distance d, the retraction speed can be overridden. When point Z has been reached, the spindle is stopped, and then rotated in the reverse direction for retraction. The tool retracts to the point R, the spindle stops. If it is G98 state, rapidly move to the initial position, the Figure is shown below:



Standard pec□ rigid tapping□

When the RTPCP of state parameter No.025 is set to 1, the standard peck rigid tapping cycle is selected.

After positioning along the X- and Y-axes, rapid traverse is performed to point R, then position to the place where specifies by C. From point R, cutting is performed with depth □ (depth of cut for each cutting feed), then the tool is retracted by distance d, the retraction speed can be overridden. The position is performed from point R to a distance d from the end of the last cutting, which is where cutting is restarted, and the cutting feed is performed. When point Z has been reached, the spindle is stopped, then rotated in the reverse direction for retraction. The tool retracts to the point R, the spindle stops. If it is G98 state, rapidly move to the initial position, the Figure is shown below:



Explanations□

When tapping feed is performing, the speed override can not be adjusted; when the retraction is performed, the speed override value is set by data parameter 084, when the data parameter 084 is set to 0, the override value is fixed as 100□.

The linear acceleration or deceleration constant value in tapping feed is set by data parameter 082, the linear acceleration or deceleration constant in retraction is set by data parameter 083, if the 083 is set to 0, the acceleration or deceleration constant in retaction is then set by data parameter 082. The start speed both tapping feed and retraction are set by data parameter 081, and the retraction distance d is set by data parameter 085.

3.□1.3 □ddress Explanation

Specified content	Address	Command address explanation
□ole position data	X、Y	Specify the hole position by the absolute value or incremental
Aparture machining data	R	From the initial plane to the point distance
	Z	Depth of a hole, the distance from point R to the bottom of the hole
	P	Specify the dwell time at the bottom of the hole or at point R when a return is made. The dwell does not perform when it is not input or the value is 0.
	□	Tool infeed value of peck tapping
	□	It indicates that the consecutive maching cycle of □holes are performed on this line segment from start (the start position of block) to XY coordinate position. The continued drilling may not perform if it is not input or the value is 0.
	F	Metric thread leading, the solution range: 0.001~500mm. The alarm 201 may alarm if it is not input.
	I	The number of the thread head per□inch, the solution range is 0.06~25400 gear□inch
	C	Start angle

3.□1.□ □ec□nic Specification

- Acceleration□deceleration

  Rigid tapping adopts the acceleration or deceleration before a straight line to control.
- Override

  The override regulation is invalid for rigid tapping infeed, but the override value can be adjusted or not which is determined by data parameter.
- Dry run

  G84□G74 can be used a dry run, the dry run e□uals to the feedrate along Z axis. The override adjustment is invalid in dry run.
- Machine lock

  G84□G74 can be used a machine lock, the tapping axis and spindle axis are not moved when the machine lock is enabled.
- Resetting

  The resetting can be reset the tapping when the rigid tapping is performed, but the G74□G84 can be not be reset.
- Dwell

  The dwell is disabled.
- Working

  G84□G74 is only valid in Auto or MDI mdoe.

- Manual feed

  The rigid tapping can not used for manual feed.
- Tool length compensation

  If the tool length compensation (G43, G44 or G49) is specified in canned cycle, the offset value is added till position to the point R.
- Cutter compensation

  Cutter compensation is ignored in canned cycle.
- Axis switching

  The Z axis tapping can only be performed in rigid mode.
- S code

  If the command speed is more than the maximum speed, the alarm may occur.
- M29

  Specify an axis movement code between M29 and G84□G74 causes alarm.
- P□□

  If they are specified in non-drilling block (If they are specified in a block that does not perform drilling), they are not stored as modal data. When □0 is specified, the peck rigid tapping cycle is not performed.

  Specify them in tapping block, they are stored as modal data, when the tapping command is retracted, either □ modal (did it).
- Cancellation

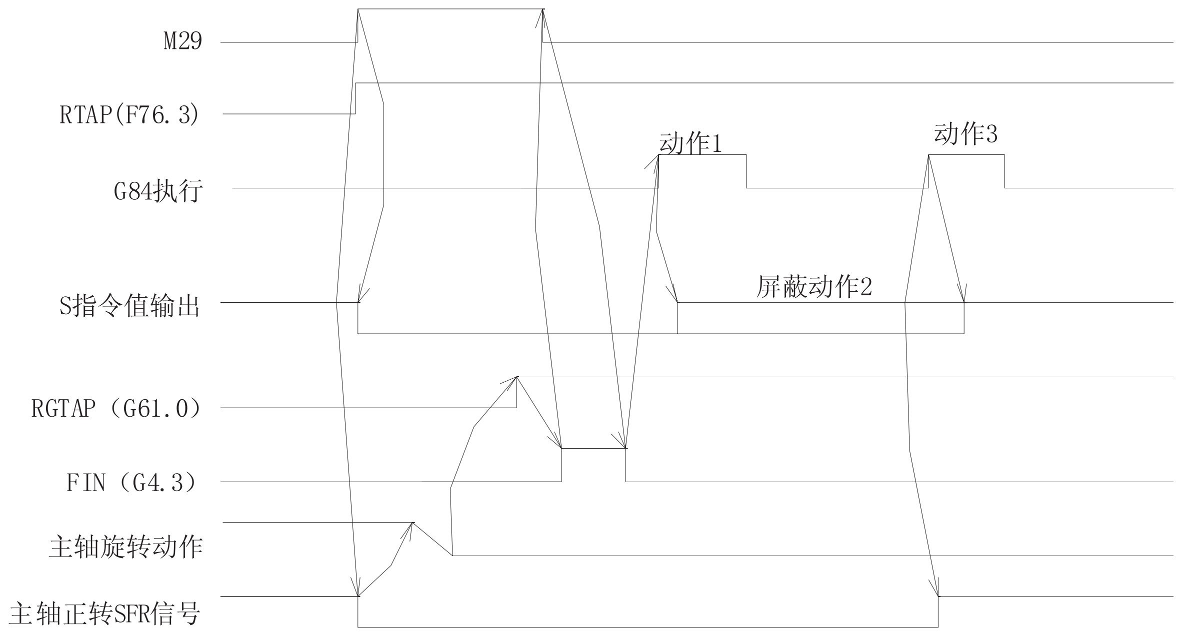
  Do not specify a group 01 G code and G84□G74 in the same block.
- A Cs contour control is used with rigid tapping at the same time.

  CS axis selects a speed mode or position mode which is determined by CON (G27.7), but, the system is rigid tapping mode, regardless of the value of CON. After the rigid tapping is cancelled, the rotation axis is either CS axis or common one which is determined by state parameter. The C axis can not be moved in manual mode when the rigid tapping is not cancelled.

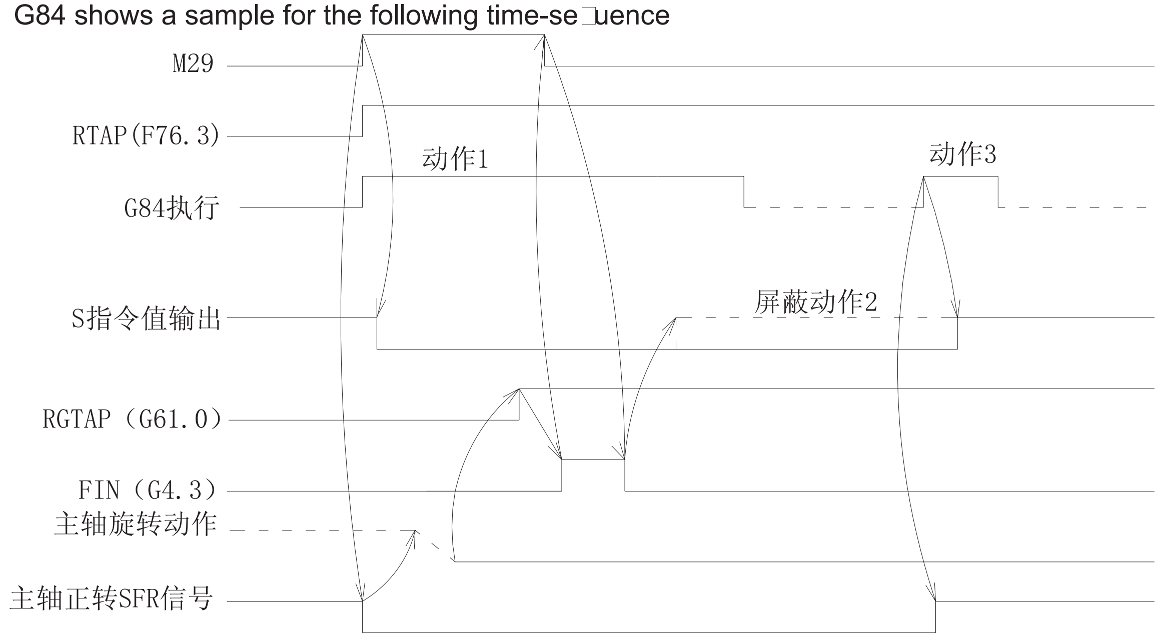
3.□1.5 Specify a □igid □apping □ode

- Specify M29 before G74□G84

  G84 shows a sample for the following time-se□uence



- Specify M29 and G74□G84 at the same block
  G84 shows a sample for the following time-se□uence



- The explanation of time se□uence

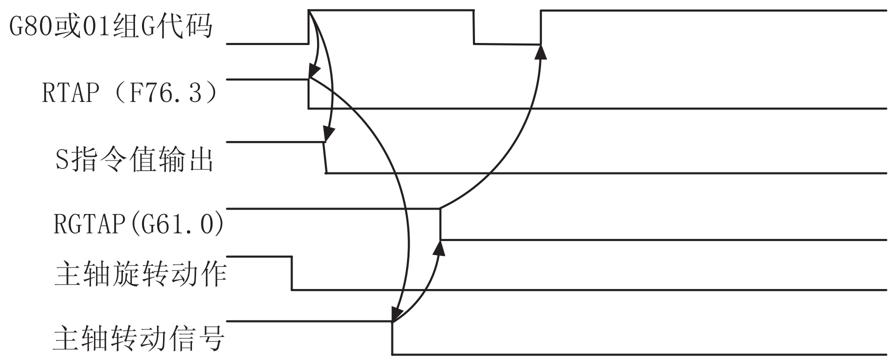
The spindle rotation operation means that the rotation axis is shifted to the position control mode (namaly, the servo spindle is needed to send a switch signal in position mode), and check the position mode arrial signal of servo spindle.

3.□1.□ □□e cancellation of rigid tapping mode

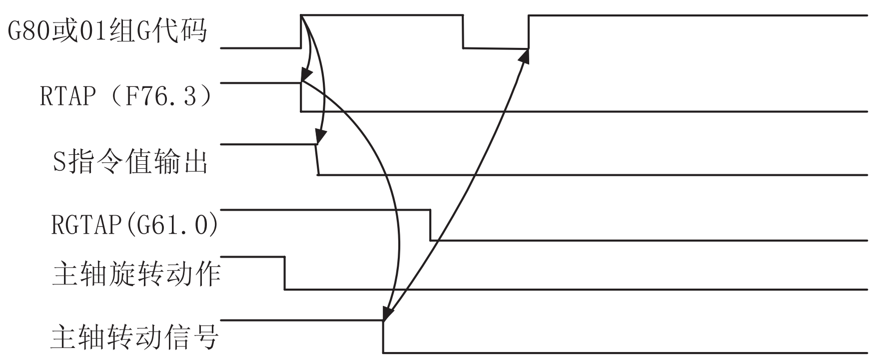
- The rigid tapping mode is canceled by G80
- Specify other canned cycles by G codes
- The other G codes of group 1.
- CNC resetting

The signal descending of F76.3 along the signal with canceling the rigid tapping of P□C, if the state RTCRG of parameter 025 is e□ual to 1, the system is then performed the next block without waiting for the rigid tapping mode signal which G61.0 is set to 0;

When the state parameter 025.2 (CRG) □0, the time se□uence is as follows:



When the state parameter 025.2 (CRG) □1, the time se□uence is as follows:



3.□1.□□and □ Signals

RGTAP (G61.0): Rigid tapping signal

When the M 29 is commanded, PMC enters the rigid tapping mode, and the signal is then set to 1 to inform the CNC

1: PMC enters the rigid tapping mode

0: PMC does not enter the rigid tapping mode

If this signal does not set to 1, after the M29 has been commanded, the alarm may occur in the block of G74□G84.

RGSPM, RGSPP (F65.1, 0) spindle turning signal

When the rigid tapping is performed, the signal is informed to the PMC whether the current spindle is CCW (positive) or CW (negative).

RGSPM: 1 spindle CW (negative)   RGSPP: 1 spindle CCW (positive)

In rigid tapping, these signals are output when the spindle is rotated. In the mode of rigid tapping, when the spindle is positioned at the hole or stoppted at the bottom of the hole or R position, these signals are not output.

In the mode of rigid tapping, when the spindle is positioned at the inter-locked stop, machine lock or Z axis ignorance states, the spindle does not regard as a stop state, in this case, these signals are output. These signals are only enabled in rigid tapping, and they are all set to 0 in the normal spindle control mode.

RTAP (F76.3): Rigid tapping process signal

This signal informs PMC which has been in the mode of rigid tapping or not. The CNC is in the mode of rigid tapping currently when the signal is set to 1.

This signal can be locked M29, P□C has been commanded the rigid tapping mode, the PMC is then treated with the correspinding logic, and this signal can be replaced the lock of M29, even so, the FIN singl of M29 is not ignored still.

3.□1.□ □larm □essage

□larm No.	Display □ontent	Explanation
218	Fail to specify the tool pitch F value in G74 or G84	Fail to specify F value
230	The spindle feed can not be performed due to the S value is 0.	S value is 0, or S code does not specify.
231	S value exceeds the maximum spindle speed allowed with rigid tapping	S value exceeds the setting value of data parameter 086
232	Other axis movement codes are specified between M29 and G74□G84.	Specify a axis movement between M29 and G74□G84__
233	G61.0 signal is abnormal in rigid tapping mode	Rigid tapping signal G61.0 is not 1 during performing in G74□G84.
234	Specify M29 repeatedly	Specify M29 or it is consecutively specified more than twice in rigid tapping.

3.□1.□ Program Example

G84 shows an example for the following program

```
O1000 (Rigid tapping example);
G0 X0 Y0 Z0;
M29 S200;
G84 X10 Y10 Z-10 R-5 P2000 F2 C20;
X20 C40
G80;
M30;
```

C□APT□R 4 CONTRO□FUNCTION of ADDITIONA□AXIS

4.1 General

The additional axis is determined by the struction design of the machine, sometimes, an additional axis is re□uired, for example, the cycle working table, rotation working table. This axis can be designed as both a linear axis and rotation axis. The basis controllable number of 980MDa is three axes, the maximum axis is 5-axis (Cs axis included). Namely, two additional axes are added based upon the original one □□ the 4th and the 5th axes, in this case, the relative functions of additional linear axis and rotation axis can be performed.

4.2 Axis Name

The names of three basis axes are always X, Y or Z. The axis name of additional axis can be set to A, B or C using data parameter No.202 and No.203.

- **Default axis name**

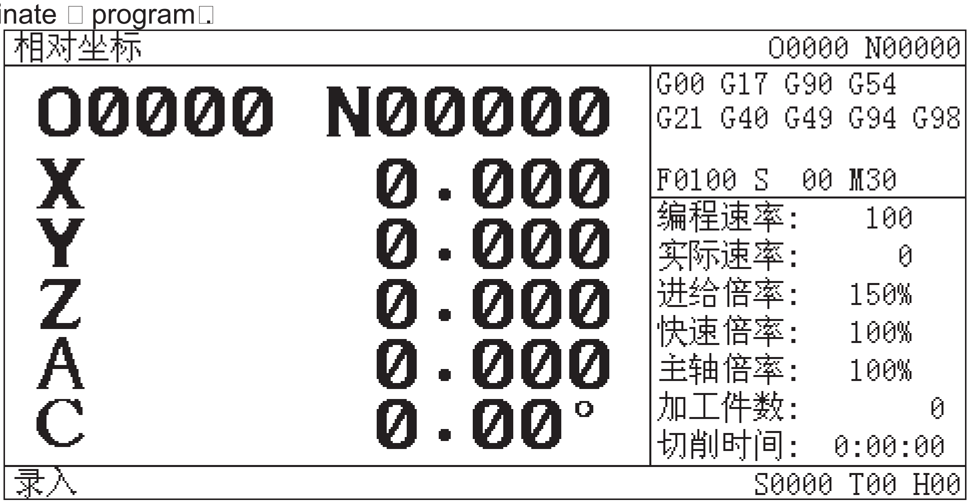
When the axis name does not set, the axis name of the 4th one is an additional axis by default; the axis name of the 5th one is C.

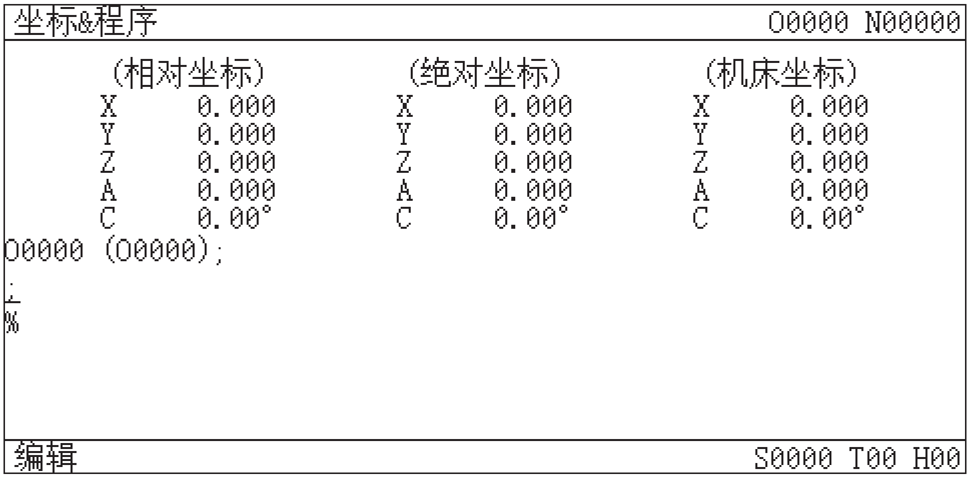
- **□epeated axis name**

When the axis name is same between the added 4th axis and the 5th axis, P□S alarm may issue.

4.3 Axis Display

When the additional axis is treated as rotation axis, the least incremental of the rotation axis is 0.01□(degree), so the 3rd digit of the decimal is displayed in unit. If it is set to a linear axis, the display is same as the basis three axes (X, Y or Z). When the 4th axis is set to a linear axis, the 5th is set to a rotation axis, the axis is displayed at the interface of □related coordinate□and □coordinate □ program□.





4.4 Axis Startup

The Bit 1 (ROSx) of data parameter No.026 and Bit0 (ROTx) of data parameter No.028 are separately set to use whether the 4th axis and the 5th axis is either the linear axis or rotation axis. The parameter settings are shown below:

ROSx	ROTx	Content
0	0	Linear axis 1. It can be switched between metric and inch; 2. All of the coordinate values are linear axis; 3. The stored pitch error compeneation is linear axis.
0	1	Rotation axis (Type A) 1. It can not be switched between metric and inch; 2. The machine coordinates are cycled based on the setting value of data parameter No.189/No.190. Whether the absolute coordinate and relative coordinate are cycled which based upon the data parameter No.027/No.029; 3. The stored pitch error compensation is rotation axis; 4. The movement amount is less than one turn when the reference position (G28, G30) is returned.
1	0	Ineffective setting (forbidden)
1	1	Rotation axis (Type B) 1. It can not be switched between metric and inch; 2. The machine coordinate is linear axis; whether the absolute coordinate and relative coordinate are cycled which based on the data parameter No.027/No.029. 3. The stored pitch error compeneation is linear axis.

Note: The start of the function of the Cs axis: the bit 5 digits (CSx) of the state parameter No.[illegible] or No.[illegible] can be set whether the function of Cs axis is enabled when the rotation axis is enabled (ROTx=1).

4.5 The Additional Axis is Linear Axis

When the additional axes (the 4th and the 5th axes) are set to linear axes, its functions are same as the basis three axes.

- **Realizable operation**

  1. Rapid traverse (Positioning): G90/91 G00 X_ Y_ Z_ A_;
  2. Cutting feed: G90/91 G01 X_ Y_ Z_ A_ F_;
  3. Skip function: G90/91 G31 X_ Y_ Z_ A_ F_;

4. Reference position return: G28/29/30 X_ Y_ Z_ A_ F_;
5. G92 coordinate setting: G92 X_ Y_ Z_ A_ ;
6. Manual/Step/MPG feed, Manual machine zero return.

Note: When there is no special explanation in the subsequent narration, the axis names of additional linear axes are expressed with [illegible].

- **Explanations**

1. When the additional linear axis rapidly moves or performs, it can be simultaneously specified with any axes of X, Y and Z. Each axis may rapidly move at its customized speed.

2. When the additonal linear axis is performed the cutting feed (G01) or used a skip function (G31), it can be simultaneously specified with any axes of X, Y and Z. in this case, the linear axis does not has an individual feedrate F but depend on each axis specified at a same time, which it is started or ended together with the specified each axis; namaly, the additional axis is shared with the basis three-axis linkage.

3. The additional linear axis can not performed a circular arc cutting (G02/03), otherwise, the P/S alarm may occur.

4. The pitch error of additional linear axis and the compensation function of inverse interval are same as the basis three-axis.

4.6 The additional axis is rotation axis

- **Input unit**

The pulse equivlance (namally, the least input unit) of 980MDa rotation axis is 0.01°(degree); the maximum vlaue of output pulse frequence is 500K.

When the selection is output based on the direction of pulse adding, it can be inputted a maximum speed n=60×f/36000=833.33 (rev./min.)

- **Rotation axis speed**

The feedrate of rotation axis is regarded the degree/min. as a unit. When the linear axis X, Y and Z is performed a linear interporlation with the rotation axis, the speed specified with F (mm/min) is the compound feedrate both X, Y and Z and the rotation axis.

Feedrate calculation: Calculate the required time when the feedrate is performed to the end; then, the feedrate unit of rotation axis is changed into degree/min..

For example: G91 G01 X20.0 C40.0 F300.0;

The unit of C axis is switched into 40mm from the 40.0 degree. The required time to the end is:

$$\frac{\sqrt{20^2+40^2}}{300} = \text{[illegible]} \ \text{(min.)}$$

The speed of C axis is:

$$\frac{40}{\text{[illegible]}} = \text{[illegible]} \quad \text{(degree/min.)}$$

Note: When there is no special explanation in the subsequent narration, the axis names of additional linear axes are expressed with [illegible].

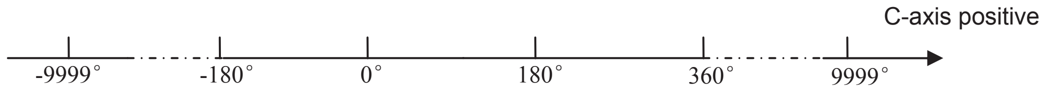
- **The cycle function of rotation axis**

The coordinate cycle function of the additional rotation axis setting is enabled, which can be avoided the coordinate value is overflowed from the rotation axis; the coordinate value will be cycled based on the setting value of data parameter No.189□No.190 (the movement amount of each axis for the rotation axis).

When the coordinate cycle function of the additional rotation axis setting is disabled, the coordinate value may change based on the linear axis, the programming command is also same to the one of the linear axis;

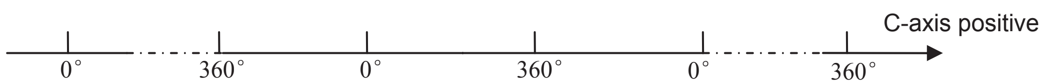
Two kinds of coordinates change are shown below:

(1) When the coordinate cycle is disabled:



The above-mentioned may occur: 1. The machine coordinate value of rotation axis (Type B)
2. The absolute coordinate value in data parameter No.027 ROAx□0 (absolute coordinate cycle function is disabled)
3. The relative coordinate value in data parameter No.027 RR□x□0 (relative coordinate cycle function is disabled)

(2) When the coordinate cycle is enabled:



The above-mentioned may occur: 1. The machine coordinate value of rotation axis (Type A)
2. The absolute coordinate value in data parameter No.027 ROAx□1 (absolute coordinate cycle function is enabled)
3. The relative coordinate value in data parameter No.027 RR□x□1 (relative coordinate cycle function is enabled)

Note 1: Refer to the Section of "Installation and connection" of the *Parameter Explanation of Chapter Three* for the parameter setting of additional rotation axis.

Note 2: When there is no special explana□ion in □□e □ub□equen□narration, the movement amount of each revolution of the additional rotation axis is expressed with 360°.

- **The pitch error compensation function of rotation axis**

When the additional axis is a linear axis or rotation axis (Type B), the pitch error compensation mode is same as the common linear axis. The pitch error compensation function is performed when the additional axis is regarded as rotation axis (Type A), refer to the following examples:

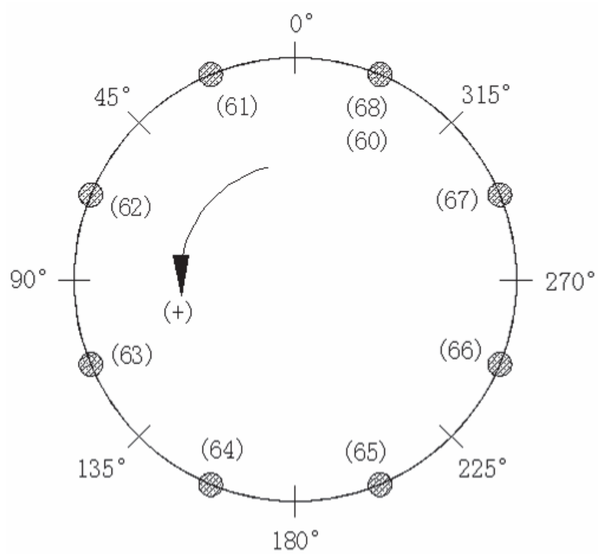
- Movement amount per revolution: 360°
- Pitch error pisition interval: 45°
- The compensation position number of reference position: 60

After the above parameters are set, the farthest compensation position number along the negative rotation axis which equals to the compensation position number of reference position;

The farthest compensation number along positive direction is shown below:

The compensation position number of reference point + (movement amount per revolution/compensation position interval) = 60 + 360/45 = 68;

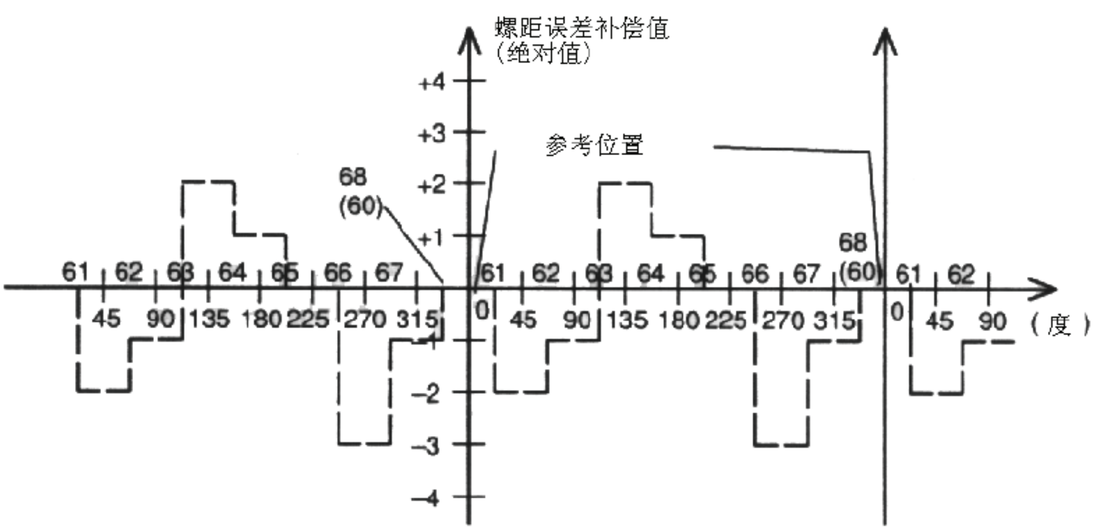
The corresponding relationships between machine coordinate and compensation position number are as follows:



The position error may occur if the total of compensation value from position 6□□68 is not 0; there is not alternative other than to set a same value at the compensation position both 60 and 68. (Because the 60 and 68 are shared a same position at the circle);

The compensation sample is shown below:

N□.	60	6□	6□	63	64	65	66	6□	68
Compensation value	□	-□	□	3	-□	-□	-3	□	□



- **The reverse interval compensation function of rotation axis**

The reverse interval compensation never changes regardless of the linear axis or rotation axis; however, the compensation unit of the rotation axis is 0.0□° (deg), and the linear axis is 0.00□(mm);

4.□The □ero return □ of rotation axis

The selection axis has four □ero return methods: □ero return method A, B, C and □. Wherein, the □ero return methods A, B and C are same as the one of the linear axis. □nly the □ is a special □ero return method for the rotation axis.

- **Setting of the □ero return method □**

The method □ is only valid to the rotation axis.

□ero return can be performed for this rotation axis using the mode □ after the 4th and the 5th axes are set to rotation axes based on the Bit6 of data parameter □o.0□□ and □o.0□□ are set to □.

□f the 4th and 5th axes are disabled or linear axes, then the Bit6 of state parameter □o.0□□ and □o.0□□ are invalid.

0	2	□		RRT□						

□□T4 = □: The □ero return mode of the 4th rotation axis is used the mode □;

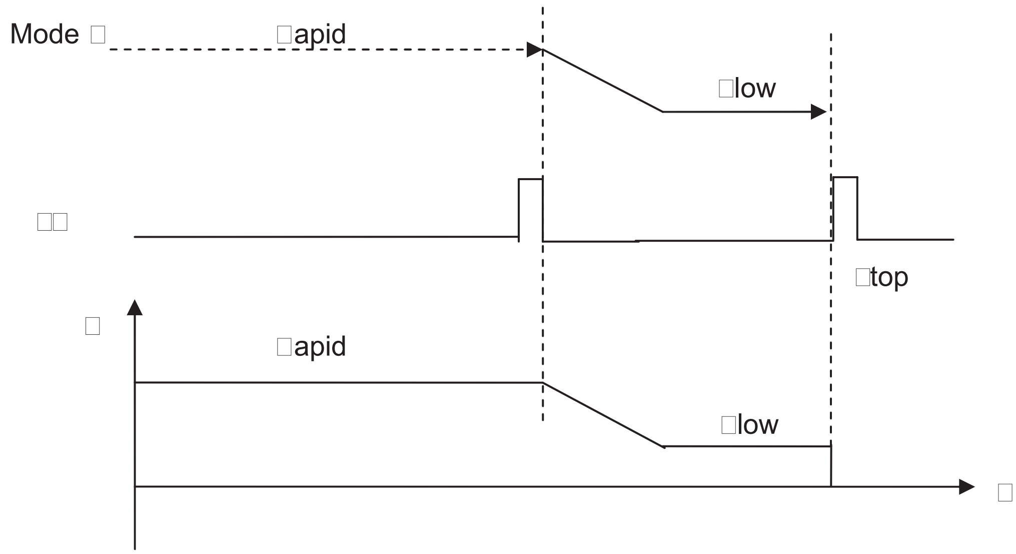
= 0: The □ero return mode of the 4th rotation axis is used the mode A, B, and C.

0	2	□		RRT□						

□□T5 = □: The □ero return mode of the 5th rotation axis is used the mode □;

= 0: The □ero return mode of the 5th rotation axis is used the mode A, B and C.

- **The time sequence and process of the □ero return mode □**



The process of □ero return

□. □elect the machine □ero return mode and press the manual positive feed □ey, the corresponding axis moves toward the □ero point at the rapid traverse rate.

□. When the one-turn signal (PC) of servo axis is carried out, the system is decelerated to the □ero return low speed, in this case, chec□ the trailing edge of PC signal.

3. The system continuously and forward operates in the □ero return low speed.

4. When the system meets one-turn signal (PC) of servo axis again, the movement stops, simultaneously, the corresponding indicator of □ero return end on operator panel goes on. The machine □ero return operation ends. □n this case, chec□ the rising edge of PC signal.

4.8 The □unction of Cs Axis

□eneral

The spindle is treated as the servo feed axis to rotate and position by the position movement command. □un speed is: degree/min., it can be interpolated together with other feed axes to machine a contour curve.

Increment s□stem: the least input increment: 0.0□deg

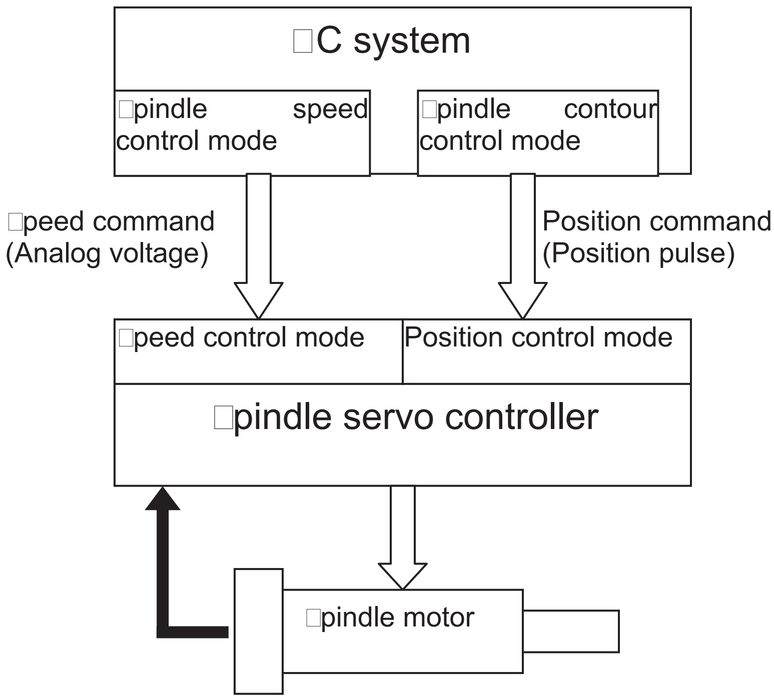
The least command increment: 0.0□deg

□xplanation: □C has two control modes for the spindle.

- □pindle speed control mode. The spindle speed can be controlled by the speed command (□amely, analog voltage).
- □pindle contour control mode (□t is also called C□ contour control). The spindle position can be controlled by the position command (□amely, position pulse).

□o, □C is required the spindle servo control unit has two control modes for the control of the spindle motor

- When □C is at the speed control mode for the control of the spindle, the spindle servo control unit can receive a speed command issued from □C to control the rotation speed of spindle motor.
- When □C is at the contour control mode for the control of the spindle, the spindle servo drive unit also can receive a position command issued from □C to control the motor operates to a specified position.



Set □s contour control axis

□n the □80M□a system, only the additional axis (the 4^{th} or the 5^{th} axis) can be set to a Cs contour control axis. But, two Cs axes can not be set at the same time. Before the Cs axis setting is valid, this axis must be set to a rotation axis. □therwise, Cs axis setting is invalid.

0	2	6		□□□	□□□	R□S□	□□□	□□□	□□□	R□S□	R□T□

□C□4 =□: The C□ axis function of the 4^{th} axis is enabled;

=0: The C□ axis function of the 4^{th} axis is disabled.

□□□4, □□T4: □et the type of the 4^{th} axis;

	□inear axis	T□pe □ rotation axis	T□pe □ rotation axis	Invalid
R□T□	0	□	□	0
R□S□	0	0	□	□

0	2	□		□□□	□□□	R□S□	□□□	□□□	□□□	R□S□	R□T□

□C□5 =□: The C□ axis function of the 5^{th} axis is enabled.

=0: The C□ axis function of the 5th axis is disabled.

□□□5, □□T5: □et the type of the 5th axis;

	□inear axis	T□pe □ rotation axis	T□pe □ rotation axis	Invalid
R□T□	0	□	□	0
R□S□	0	0	□	□

The switch between spindle speed control and □S contour control

The □C switching of spindle control mode is performed by the C□□ signal of P□C.

□n the C□ contour control mode of □C, the C□ contour control axis, as the common servo axis, can be performed manually or automatically.

- □rom spindle speed control shifts to the Cs contour control
  □et the C□□ (□0□□□□) to □, then the spindle can be set in the Cs contour control mode. □f the switch is performed during the spindle rotation, the spindle is immediately stopped and then shifts.
- □rom Cs contour control shifts to the spindle speed control
  □et the C□□ (□0□□□□) to 0, the spindle is then set in the spindle speed control mode. Confirm the spindle movement command has been ended before shifting, if the shift is performed when the spindle is being moved, the system will alarm.

The reference position return of □s contour control axis

After the spindle is shifted to the Cs contour control mode from the speed control mode, the current position is not confirmed, the spindle should be returned to the reference position.

The reference position return of Cs contour control axis is as follows:

- Manual reference position return
  After the spindle enters the Cs contour control mode, shift to the machine □ero return mode. The □ero return of Cs axis is performed opening the feed axis and the direction selection signal ＋□n (□□00) or －□n (□□0□).
- Automatic
  □pecify □□8 after the spindle enters the Cs contour control mode, and the spindle moves to the intermediate point and then return to the reference position.

□Pn (□0□4) becomes □ after the referece position return is executed.

The operation of □s contour control axis

□□anual□□utomatic□

□f the Cs contour control axis has been returned to the reference position, the operation of Cs axis is same as the common □C axis.

□n the spindle speed control, the Cs contour control axis can not be performed. □therwise, the system alarms.

□o, in the spindle speed control mode, it is not permitted the manual operation of Cs by the P□C ladder diagram.

The signal shift of spindle contour control

□□N □□02□□□□

□Type□ □ignal input

□□unction□ This signal is used for shifting between spindle speed control mode and Cs contour control mode.

When this signal is set to □, the spindle is shifted to the Cs contour control mode from speed control mode.

When this signal is set to 0, the Cs contour control mode comes bac□ to the speed control mode.

The signal shift end of spindle contour control

□S□S□□□0□□□1□

□Type□ □ignal output

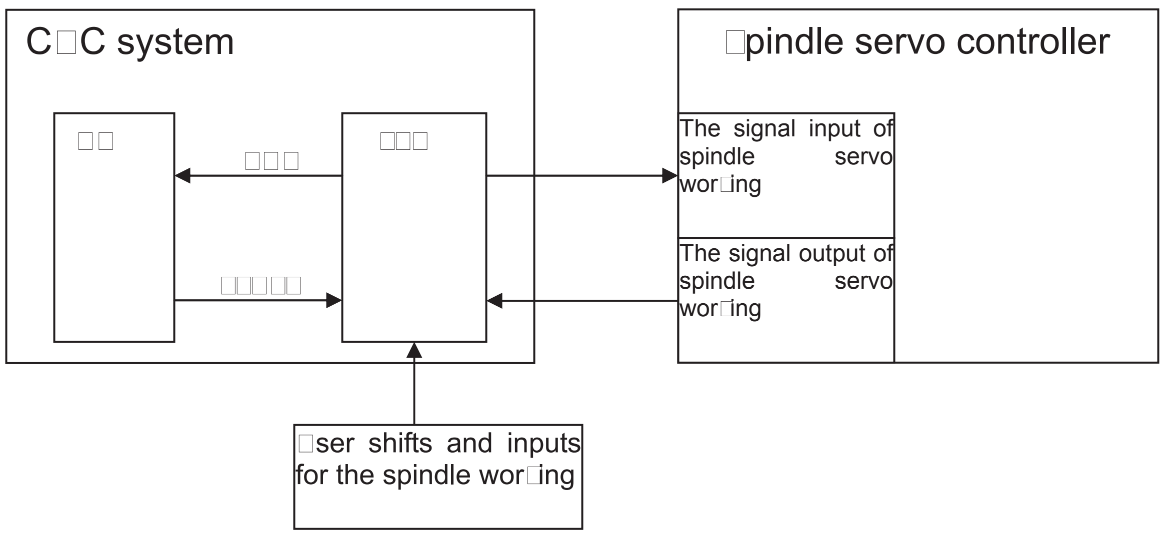
□□unction□ This signal indicates that the controlled axis has been controlled under the Cs contour.

□□utput condition□ □pindle speed control mode —□0

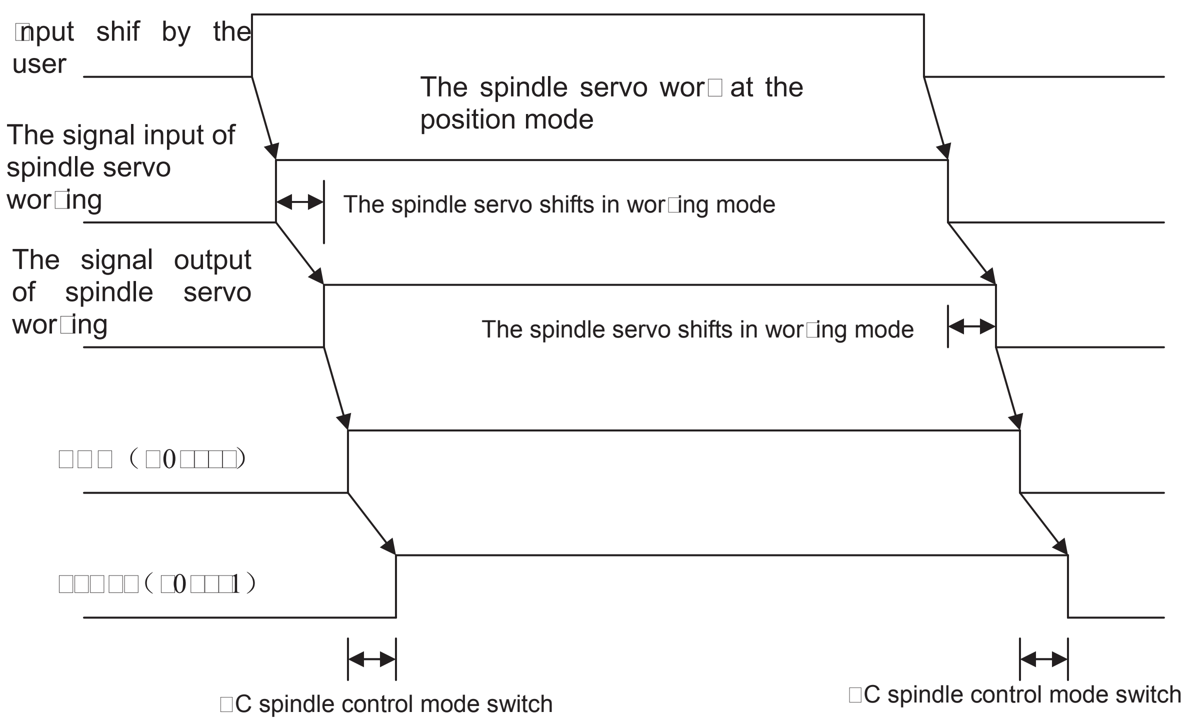
Cs contour control mode —□□

□N□ and spindle servo control unit

The signal shift relationship of the spindle wor□ing



Time sequence figure



Relative parameter

0	□	□	The start speed of acceleration/deceleration of C□ axis

Resolution range: 0～5000 (Unit:deg/min)

0	□	□	The acceleration/deceleration time constant of C□ axis

Resolution range: □0～4000 (Unit: ms)

- **The explanation of “two points same”**

Radius compensation mode is pre-read two blocks. Caculate the transit point and perform a path movement taking 3 position points (the start of the □st block, the intersection of the □st and the □nd blocks, the end of the 3rd block). In this case, “two same points” may occur in the following items:

(a) The first two points are same when starting.

(b) The last two points are same when starting.

(c) The first two points are same during the compensation.

(d) The last two points are same during the compensation.

(e) The first two points are same during the retraction.

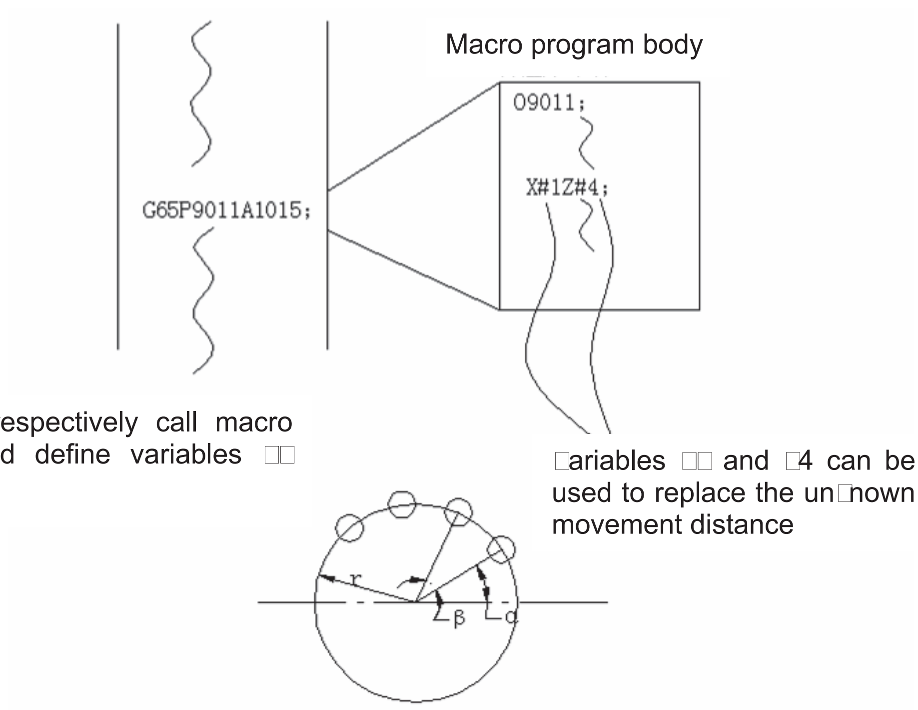
(f) The last two points are same during the retraction.

The “two same points” is regarded the point as a linear of which approximates to zero, when the “two same points” occurs, the transit point calculation can be performed based on the straight line (point) to straight line (point), straight line (point) to circular arc (point), circular arc (point) to straight line (point) and circular arc (point) to circular arc (point).

□□□□T□R □ □□□R□ □R□□R□□

□□□□80M□a provides macro programs which is similar to high level language. □ariable assignment, arithmetic operation, logical □udgment and conditional branch can be reali□ed through custom macro program. □t is in favor of the programming for special parts, lessens the complex operation and simplifies the custom program.

Custom macro programs are similar to subprograms. □owever, macro program allows variable assignment, arithmetic operation, logical □udgment and conditional branch, which ma□es it easier to program the same machining process.



□t is easy to machine the screw holes distributed in circles (shown in the figure above).

After a macro program used in circular holes is programmed and edited, it can be performed if the □C system has circular hole machining function.

By the following command, programming personnel can use circular holes function.

□65　P p □ r A a B b □ □；

p：Macro program number of circular holes

r：□adius

a：□tart angle of the hole

b：Angle of holes intervals

□：□oles number

□n this way, users can improve the □C performance on their own. Macro programs can be either provided by machine tool builder or defined by users.

5.□ Macro Call

Macro call (□65, □66) differs from subprogram call (M□8) as described below:

□. With □65 or □66, an argument (data passed to a macro) can be specified. M□8 does not have this capability.

□. When an M□8 bloc□ contains another □C command (for example, □0□ □□00.0 M□8 P), the macro program P□ is called after the command □0□ is executed. □n the other hand □65 unconditionally calls a macro P□.

3. When an M□8 bloc□ contains another □C command (for example,□0□ □□00.0 M□8 P□), the machine stops in the single bloc□ mode. □n the other hand, □65 does not stop the machine.

4. With □65 or □66, the level of local variables changes. With M□8, the level of local variables does not change.

- **Non□modal call**（□6□）

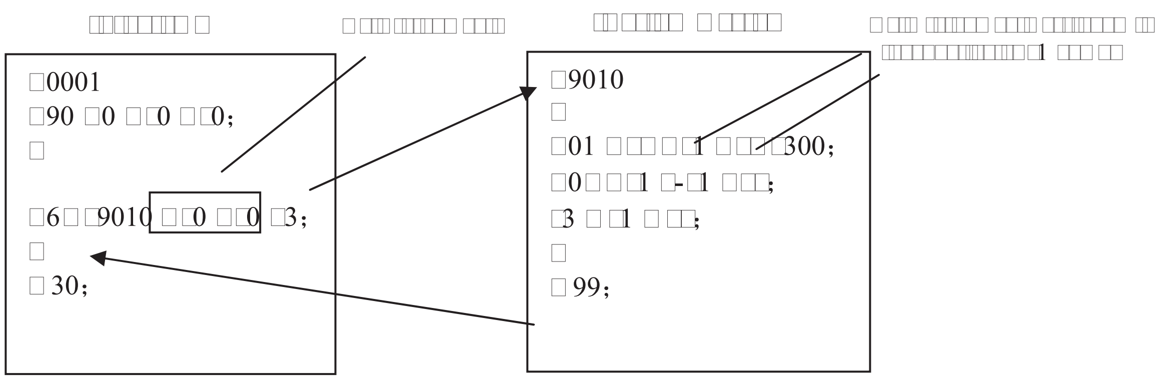
When □65 is specified, the macro program specified at address P is called. Argument (data) can be passed to the custom macro program.

□ormat：□65 P□ □□ □argument□□;

□xplanation：P □ □ number of the program to be called

□ □ □ repetition count（□by default, □to □□□□ can be specified）

□Argument□ □ □ □ata passed to the macro. □ts value is assigned to the corresponding local variables.



□rgument specification: two t□pes of argument specification are available.

Argument specification □: it uses letter other than □, □, □, □ and P once each. □n repeated specification, the last one prevails.

Argument specification □

Address	Variable number	Address	Variable number	Address	Variable number
A	#1	I	#4	T	#20
B	#2	J	#5	U	#21
C	#3	K	#6	V	#22
D	#7	M	#13	W	#23
E	#8	Q	#17	X	#24
F	#9	R	#18	Y	#25
H	#11	S	#19	Z	#26

Note： □ddresses that need not to be specified can be omitted. □ocal variables corresponding to an omitted address are set to null.

Argument specification □□: □ses A, B, C and □i, □i, □i (i is □□□0) and automatically decides the argument specification type according to the letters and the sequence. □ses A, B, C once each and uses □, □, and □ up to ten times.

□rgument specification II

Address	Variable number	Address	Variable number	Address	Variable number
A	#1	K_3	#12	J_7	#23
B	#2	I_4	#13	K_7	#24
C	#3	J_4	#14	I_8	#25
I_1	#4	K_4	#15	J_8	#26
J_1	#5	I_5	#16	K_8	#27
K_1	#6	J_5	#17	I_9	#28
I_2	#7	K_5	#18	J_9	#29
J_2	#8	I_6	#19	K_9	#30
K_2	#9	J_6	#20	I_{10}	#31
I_3	#10	K_6	#21	J_{10}	#32
J_3	#11	I_7	#22	K_{10}	#33

Note 1: Subscripts of I, □ and □ for indicating the order of argument specification are not written in the actual program.

Note 2: □rgument I, □, □ do not need to be written in orders. The□ will be identified according to the present sequence. □or example: □6□ □□010 □1 □2 □3 I1□ □1□ I6 □□ □□ □11 □12 □30□The variables are passed as follows:

I1□→□□, □1□→□□, I6→□□, □□→□□, □□→□6, □11→□□, □12→□12, □30→□11;

□ormat: □65 must be specified before any argument.

□ixture of argument specifications I and II: The □N□ internall□ identifies argument specification I and II. If a mixture of argument specification I and II is specified, the t□pe of argument specification specified later ta□e precedence.

□xample
□6□ □9001 □1□□ □□0 -3□3 □□ □□□

□variable□
#1: 1.2
#2: 2.0
#3: Null
#4: -3.3
#5: Null
#6: Null
#7: ~~4~~ 5

When both I4 and D5 arguments are commanded for variable # 7 in this example, the later, D5 is valid.

- **□odal call（□66）**

□nce □66 is issued to □ bloc□ specifying movement along axes is executed. This continues until □6□ is issued to cancel a modal call.

Note: The format, functions and argument specification of □65 are identical with that of the □65 (non-modal call). (□efer to the introduction of □65 for detailed description).

□odal call nesting: □odal calls can be nested b□ specif□ing another □66 code during

a modal call.

Explanation： □. In the specified □□□ bloc□, onl□ argument is passed, and macro modal call □ill not be executed.

□. □acro modal call can onl□ be executed in the bloc□s □ith □□□, □□□, □□□, and □□□

□. □o macro program can be called in a bloc□ □hich contains a code such as miscellaneous function that does not involve movement along an axis.

4. □□5 and □□□ should not be specified at the same time.

5. □ultiple macro programs cannot be called in □□□ bloc□.

□. □s □ith □□5, □□□ should be specified prior to arguments and □.

- **Sample program**
  - **G65 call (bolt hole circle)**

□reate a macro program for machining holes on a circle. □he radius is I□start angle is □□ holes interval is □, holes number is □□the center of the circle is □□,□□. □ommands can be specified in either the absolute or incremental mode. □o drill in the cloc□□ise direction, specif□ a negative value for □.

Format： □□5 □□□□□ □x □□ □□ □r li □a □b □h□

□：□ coordinate of center point □absolute or incremental□□#□4□

□：□ coordinate of center point □absolute or incremental□□#□5□

□：□ole depth（#□□）

□：□oordinates of an rapid approaching point（#□□）

□：□utting feedrate（#□）

I：□ircle radius（#4）

□：Drilling start angle（#□）

□：Incremental angle □cloc□□ise □hen negative value is specified□. □#□□

□：□umber of holes （#□□）

Macro call ： □□□□□

□□□ □□□ □□ □□ □□□□□

□□5 □□□□□ □□□□ □5□ □□□ □5□ □5□□ I□□□ □45 □□□ □5□

□□□□

Macro program（the called program)： □□□□□

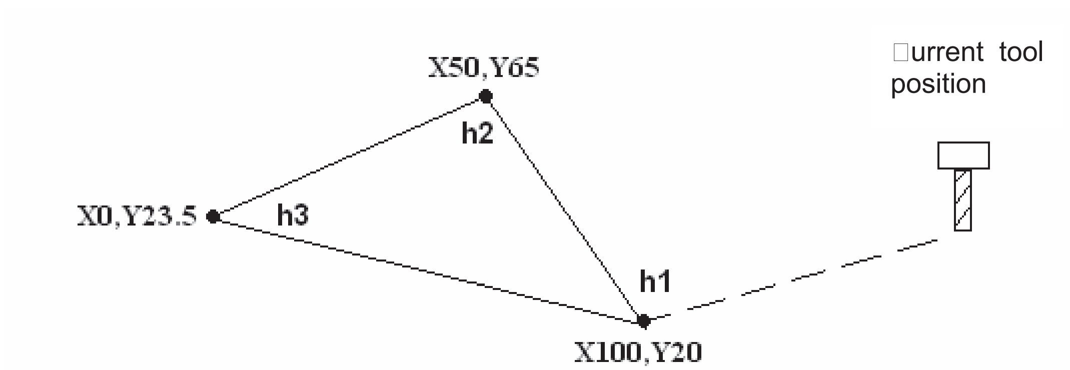
```
#□□#4□□□ □ □ □ □ □ □ □ □ □ □ □ □ .. □tores □ codes of □□ group
I□#□ □□ □□□ □□□ □□□ □ □ □ □ □ □ □ □ranches to □□ in the □□□ mode
#□4 □#5□□□□#□4□□ □ □ □ □alculates the □ coordinate of the center point
#□5 □#5□□□□#□5□□ □ □ □ □ □alculates the □ coordinate of the center point
□□W□I□□ □#□□ □□ □□D□ □□□ □ □ □ntil the number of remaining holes reaches □
#5□#□4□#4□□ □ □ □#□□□□ □ □ □ □ □ □ □ .□alculates the hole position on □ axis
#□□#□5□#4□□I□#□□□□ □ □ □ □ □ □ □ ..□alculates the hole position on □ axis
□□□ □□□ □#5 □#□ □#□□ □#□□ □#□□□ □ Drilling after moving to the target position
#□□#□□#□□□ □ □ □ □ □ □ □ □ □ □ □ □ □ □pdates the angles
#□□□#□□□□□□ □ □ □ □ □ □ □ □ □ □ □ .□ .Decrements the number of holes
□□D □□
```

□#□□□□□□ □□□□□□□□□eturns the □ codes to the original state.
□□□□

□rgumen□meaning□：#□store □ codes of □□group
#5 □ coordinate of the next hole to drill
#□□ coordinate of the next hole to drill

➢ **G66 modal call**

Sho□n a□ □ollo□ □□machine □hole□ □h□,h□,h□□



□all □ormat：□□□ □□□□□ □a □b □c□ （the argument in this example is assumed）

Macro program： □□□□□
□□□ □□7 □□□ □□ □□ □□;
□□□ □□5□ □□□; □□□position
□□□ □□□□□ □□□□ □□4□ □□□□□; □□□pass the argument, be read□for machining
□□□ □□□□ □□□; □□□position to h□, call macro program □hole machining□
□□□ □5□ □□5; □□□position to h□, call macro program □hole machining□
□□□; □□□non□movement code, does not call macro program
□□□ □□ □□□.5; □□□position to h□, call macro program □hole machining□
□□7; □□□cancel macro program modal call
□□□ □□5□ □□□; □□□positioning return
□□□□

□alled macro program：□□□□□（machining process）
□□□ □□□ □#□ □#□ □#□;
□□□;

5.□ □ariables

□n ordinar□ machining program specifies a □ code and the travel distance directl□ □ith a numeric value, for example, □□□ and □□□□.□. With a custom macro program, numerical value can be specified directl□ or using variables, for example, □#□□□ □#□□□. When variables are used, the variable value can be changed b□programs or using operation on the □DI panel.

- **□epre□entation and u□ing method□o□□ariable□**

Differ from argument □data□, variables are considered as the carrier of data, for example, #□, #□□□ □ are variables□□□□□, □□□□ □ are arguments. Data of arguments □□□□, □□□□ should be transferred to variable #□ and #□. When using or programming macro programs, numerical value can be specified directl□ □such as □□□, □□□□□or using variables □such as □#□□, □#□7□. When variables are used, the variable value can be changed b□programs or using operation on the panel.

□he address value of a macro bod□can be specified b□variables. □he variable value can be set b□the main program or be assigned the calculated value □hen executing the macro bod□. □ultiple variables can be identified b□numbers.

□□□□ariable representation

□ number sign # follo□ed b□a variable number is sho□n as follo□s□

#i □i □ □, □, □, 4 □ □ □. □or example□#5, #□□□, #□□□5

□□□. □mission of decimal point

When a variable value is defined in a program, the decimal point can be omitted. □or example□□hen defining #□□□□□, the actual value of variable #□ is □□□.□□□.

□□□. □eferencing variables

□o reference the value of a variable in a program, specif□ a □ord address follo□ed b□ the variable number. □ program □ith an expression □address□#i or □address□□#i indicates that the variable value or negative value is used as address value.

□or example□ □□#□□□□ □hen #□□□ □ □5□, it is e□uals to □□□5□.

□#□□□□ □hen #□□□ □ □, it is e□uals to □□

□4□. □eplace variable numbers □ith variables

When replace variable numbers □ith variables, #□□□□ rather than ##□□□ is used, the □ follo□ed # means the replacement. □or example□□hen #□□□ □ □□5, #□□5 □ 5□□,

□#□□□□ and □5□□ are e□ual. i.e. □#□□□□ → □##□□□, □#□□5 → □5□□

□□#□□□□ and □□5□□ are e□ual.

□ote: **Program number o□□e□uence number □ and optional bloc□ □□ip number □□cannot be □ollo□ed □ith □ariable□. For example□□#□, □#□, □#□.**

- **Variable di□pla□**

□acro variables □□□□□ □□□□□

□o.	Data	□o.	Data	□o.	Data
□□□	□ull	□□□	□□□.□□□	□□□	□ull
□□□	□□.□□5	□□□	□ull	□□7	□ull
□□□	□□□□□□□□□□	□□□	□ull	□□□	□ull
□□□	□.□□□	□□□	□ull	□□□	□ull
□□4	□.□□□	□□□	□ull	□□□	□ull
□□5	□ull	□□□	□□□□□□□□□□	□□□	□ull
□□□	□ull	□□4	□ull	□□□	□ull
□□7	□ull	□□5	□ull	□□□	□ull

□o. □□□

□DI□ □□□□□ □□□ □□□

1. □n macro variable page, □□ull□indicates the variable is null, i,e, undefined. □he mar□ □□□□□□□□□□ indicates the variable value overflo□s of the range □but the internal stored data ma□ not overflo□□.
2. □he value of common variables □#□□□□#□□□, #5□□□#□□□□can be displa□ed on macro variable page, or be assigned directl□ b□ inputting data on the page.
3. □he value of local variables □#□□#□□□and s□stem variables do not have displa□ screen. □ value of local variable or s□stem variable can be displa□ed b□ assigning the value to common variables.
4. □ariable data range□integral t□pe□□□□474□□□4□□□□474□□□47， real number t□pe：□□□[47]□□□□[□□□], □, or □□[□□□]□□□[47].

Intergra t□pe□□□474□□□4□□□□474□□□47 real number t□pe□□□□[47]□□□□[□□□], □, or □□[□□□]□□□[47].

- **□□pe□o□□ariable□**

□ariables are classified into four t□pes b□ variable number□

Variable number	□□pe o□ □ariable	Function	□ange	□emar□
#□	□ull variable	□his variable is al□a□s null. □o value can be assigned to this variable.	□□□□	
#□□#□□	□ocal variable	□ocal variable can onl□ be used □ithin a macro to hold data such as the results of operations. When the po□er is turned off, local variables are initiali□ed to null. When a macro is called, arguments are assigned to local variables.		
#□□□□#□□□	□ommon variable	□ommon variables can be shared among different macro programs.	When the po□er is turned off, variables	read□ □rite□

			are initiali□ed to null.	displa□
#5□□□#□□□			When the po□er is turned off, data is stored	
#□□□□□#□□□5	□□stem variable (□□4)	□54, □55 output	□,□ processed b□ □□□	□ead onl□
#□□□□		□tore □54, □55, read all □□ bits of a signal at one time		
#□□□□□#□□□5		□54, □55 input		□ead□□r ite
#□□□□		□tore □54, □55,□rite all □□ bits of a signal at one time		
#□□□□		□tore □5□□□5□, □rite all □□ bits of a signal at one time		
#□□□□□#□□□□	□□stem variable	□ool length compensation □ear	□□□□□.□□□□□□□□.□□□	□ead□□r ite
#□□□□□#□□□□		□ool length compensation	□□□□□.□□□□□□□□.□□□	□ead□□r ite
#□4□□□#□4□□		□utter compensation □ear	□□□□□.□□□□□□□□.□□□	□ead□□r ite
#□□□□□#□□□□		□utter compensation □ear	□□□□□.□□□□□□□□.□□□	□ead□□r ite
#□□□□□#□□□4		□utomatic operation control□ #□□□□	□, □, □, □	□ead□□r ite
		□utomatic operation control□ #□□□4	□□7	□ead□□r ite
#□□□□		□he number of machined parts	□□□□□□□□□□□	□ead□□r ite
#4□□□		□□□, □□□, □□□, □□□, □7□, □74, □□□, □□□, □□□, □□□, □□4, □□5, □□□, □□□, □□□, □□□□, □□□□, □□□□, □□□□, □□□4, □□□5, □□□4, □□□5, □□□□, □□□7, □□□□, □□□□	modal □ code group□	□ead onl□
#4□□□□#4□□□		□□7, □□□, □□□□ #4□□□	modal □ code group □	□ead onl□
		□□□, □□□□ #4□□□	modal □ code group □	□ead onl□
#4□□5□#4□□7		□□4, □□5□ #4□□5	modal □ code group 5	□ead onl□
		□□□, □□□□ #4□□□	modal □ code group □	□ead onl□
		□4□, □4□, □4□□ #4□□7	modal □ code group 7	□ead onl□
#4□□□		□4□, □44, □4□	modal □ code group □	□ead onl□
#4□□□		□□□, □□□	modal □ code group □□	□ead onl□
#4□□4		□54□□5□	modal □ code group	□ead

			□4	onl□
#4□□7		D code	□□□□	□ead onl□
#4□□□		□ code	□□□5□□□	□ead onl□
#4□□□		□ code	□□□□	□ead onl□
#4□□□□#4□□5		□ code□ #4□□□	□□□□	□ead onl□
		□e□uence number□ #4□□4	□□□□□□□	□ead onl□
		□rogram number □ #4□□5	□□□□□□	□ead onl□
#4□□□□#4□□□		□ code□ #4□□□	□□□□□□	□ead onl□
		□ code□ #4□□□	□□□□	□ead onl□
#5□□□□5□□5	□□stem variable	□□5 axes□ bloc□ end point□ □or□piece coordinate s□stem□tool compensation value not included	□□□□□.□□□□□□□□.□□□	□ead onl□
#5□□□□5□□5		□□5 axes□ current position□ machine coordinate s□stem□ tool compensation value included	□□□□□.□□□□□□□□.□□□	□ead onl□
#5□4□□5□45		□□5 axes, the current position, □or□piece coordinate s□stem contain tool compensation value	□□□□□.□□□□□□□□.□□□	□ead onl□
#5□□□□5□□5		□□5 axes, s□ip signal position□ □or□piece coordinate s□stem□tool compensation value included	□□□□□.□□□□□□□□.□□□	□ead onl□
#5□□□□5□□5		□□5 axes□ tool length compensation value□ current execution value.	□□□□□.□□□□□□□□.□□□	□ead onl□
#5□□□□5□□5		□□5 axes□external □or□piece □ero point offset value	□□□□□.□□□□□□□□.□□□	□ead□□r ite
#5□□□□5□□5		□□5 axes, □54 □or□piece □ero point offset value	□□□□□.□□□□□□□□.□□□	□ead□□r ite
#5□4□□5□45		□□5 axes, □55 □or□piece □ero point offset value	□□□□□.□□□□□□□□.□□□	□ead□□r ite
#5□□□□5□□5		□□5 axes, □5□ □or□piece □ero point offset value	□□□□□.□□□□□□□□.□□□	□ead□□r ite
#5□□□□5□□5		□□5 axes, □57 □or□piece □ero point offset value	□□□□□.□□□□□□□□.□□□	□ead□□r ite
#5□□□□5□□5		□□5 axes, □5□ □or□piece □ero point offset value	□□□□□.□□□□□□□□.□□□	□ead□□r ite
#5□□□□5□□5		□□5 axes, □5□ □or□piece □ero point offset value	□□□□□.□□□□□□□□.□□□	□ead□□r ite

5.□.□ □ull Variable□

When the variable value is undefined, the variable is null. □ariable #□ is al□a□s null, and can be read onl□.

a, referencing

□he address itself is ignored □hen an undefined variable □null variable□is □uotated.

□ **hen □□□□ □ull□□**	□ **hen □□□□**
□ □□ □□□□ □#□ e□uals to □ □□ □□□□	□ □□ □□□□ □#□ e□uals to □ □□ □□□□ □□

b, □rithmetic operation

□□ull□ e□uals to □in an□case except □hen assigned b□ □□ull□.

□ **hen □□□□ □ull □**	□ **hen □□□□**
#□□#□ （assignment） □he arithmetic operation result #□ e□uals to□ □ull□	#□□#□ □he arithmetic operation result #□ e□uals to □
#□□#□ * 5 □he arithmetic operation result #□ e□uals to □	#□□#□ * 5 □he arithmetic operation result #□ e□uals to □
#□□#□□#□ □he arithmetic operation result #□ e□uals to □	#□□#□□#□ □he arithmetic operation result #□ e□uals to □

c. □onditional expression

□□ull□ differs from □ onl□for □□ and □□.

□ **hen □□□ □ull**	□ **hen □□□□**
#□ □□ #□ ↓ □rue	#□ □□ #□ ↓ □alse
#□ □□ #□ ↓ □alse	#□ □□ #□ ↓ □alse
#□ □ □ #□ ↓ □alse	#□ □ □ #□ ↓ □alse
#□ □ □ #□ ↓ □alse	#□ □ □ #□ ↓ □alse

5.□.□ □ocal Variable□

□ocal variables are the variables internall□ defined in a program. □he□ are effective onl□ □ithin the program, i.e., it is onl□ can be used □ithin the program.

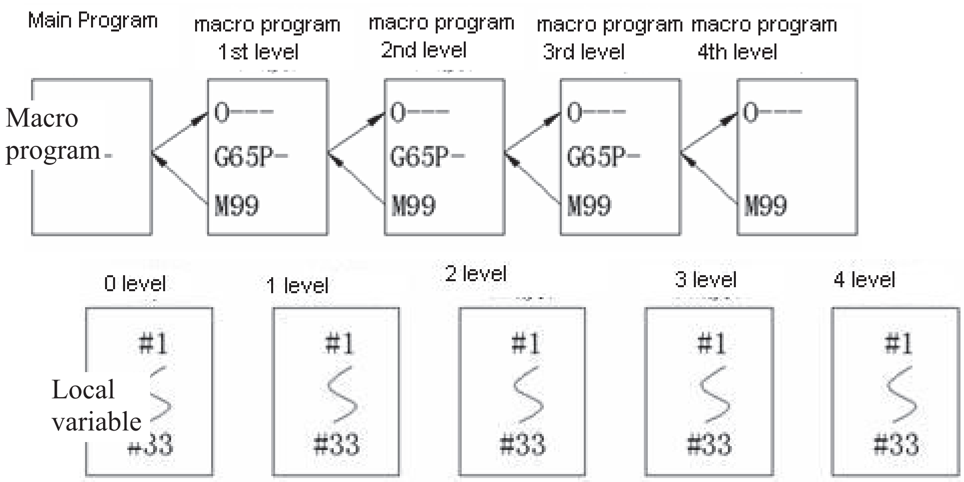
□ local variable #□ that calls macro programs at a certain moment is different from the #□ at another moment. □□o matter the macro programs are identical or not□. □herefore, □hen macro program □ is called from macro program □, li□e nesting, the local variables used in macro □ □ill not be misused in macro □, and □ill not disable the value in macro □.

Usually, the local variables are used to accept the value passed from argument. Please refer to Argument Specification for the relationship between arguments and addresses. Pay attention that, the initial state of local variable is Null, before the local variable is defined (assigned).

- **Custom macro program nesting and local variable**

When calling a macro program, its nesting level increases by one, and correspondingly, the level of local variable increases by one as well.

The relationship between macro program call and local variable is shown as follows:



- **Explanation:**

1. #1~#33 local variables (0 level) are provided in the main program.

2. When a macro program (1 level) is called by G65, the local variable (0 level) is stored, and local variables #1~#33 of the new macro program is prepared. The argument replacement is possible (the same as ③).

3. Each time a macro program (2, 3, 4 levels) are called, local variables (1, 2,3 levels) in each group are stored, and new local variables (2,3,4, levels) are prepared.

4. When M99 (return from macro programs) is commanded, the local variables (0, 1, 2, 3 levels) stored in ②, ③ are recovered in the state as they are stored.

5.2.3 Common Variable

Common variable is the global variable defined within the system. It can be used in any program. That is to say, #100 used in a macro program is the same as the one used in another macro program. Therefore, the arithmetic operation result of common variable #100 in a program can be used in another program.

In the system, there is no special regulation for using common variables. #100~#199 is the variable group without power-off memory function; #500~#999 is the variable group with power-off memory function, i.e. data are stored after power-off.

5.2.□ □□□□em Variable□

□ystem variables are used to read and write C□C internal data, such as tool length compensation value, tool nose radius compensation value. □ome system variables can only be read. □ystem variables are the basis of automatic control and general-purpose machining program development.

- **□n□er□a□e □i□nal** The macro variable corresponding to interface signal is the e□change signal between □□C and custom macro program.

Variable □o.	□□n□□ion
#1000~#1015	□ 1□-bit signal can be sent from the □□C to a custom macro. □sed to read signal bit by bit.
#1032	□ 1□-bit signal can be sent from the □□C to a custom macro. □sed to read al 1□ bits of a signal at one time.
#1100~#1115	□ 1□-bit signal can be sent from the □□C to a custom macro. □sed to read and write signal bit by bit.
#1132	□ 1□-bit signal can be sent from the □□C to a custom macro. □sed to read and write all 1□ bits of a signal at one time.
#1133	□ 32-bit signal can be sent from the □□C to a custom macro. □sed to read all 32 bits of a signal at one time.

□o□e□□lea□e re□er □o □□e *GSK980TD PLC User Manual* □or □□e rela□ion□□i□□ be□□een □ariable□ an□ □□□ □i□nal□.

- **□ool □om□en□a□ion □al□e** tool compensation value can be read and written

Com□en□a□ion □o.	□ool len□□□ □om□en□a□ion		C□□□er □om□en□a□ion	
	□eome□ri□ (□)	□ ear (□)	□eome□ri□ (□)	□ ear (□)
01	#2201	#2001	#2□01	#2401
02	#2202	#2002	#2□02	#2402
03	#2203	#2003	#2□03	#2403
□ □ .				
31	#2231	#2031	#2□31	#2431
32	#2232	#2032	#2□32	#2432

- **Automatic operation control**   The control state of automatic operation can be changed

Variable No.	Variable value	Single block	Completion of an auxiliary function
#3003	0	Enabled	To be awaited
	1	Disabled	To be awaited
	2	Enabled	Not to be awaited
	3	Disabled	Not to be awaited

Note 1: When the power is turned on, the value of this variable is 0.
Note 2: When single block stop is enabled (SBK=1), in this case, #3003 can change the execution of single block stop.
Note 3: When single block stop is disabled (SBK=0), in this case, single block stop operation is not performed even if the single block switch is set to ON.
Note 4: When awaiting for the completion of auxiliary function (M, S and T functions) is not specified, program execution proceeds to the next block before completion of auxiliary functions. Also distribution completion signal DEN is not output.

Variable No.	Variable value	Feed hold	Feedrate override	Exact stop
#3004	0	Enabled	Enabled	Enabled
	1	Disabled	Enabled	Enabled
	2	Enabled	Disabled	Enabled
	3	Disabled	Disabled	Enabled
	4	Enabled	Enabled	Disabled
	5	Disabled	Enabled	Disabled
	6	Enabled	Disabled	Disabled
	7	Disabled	Disabled	Disabled

Note 1: When the power is turned on, the value of this variable is 0.
Note 2: When feed hold is disabled, if the feed hold button is held down, the machine stops in the single block stop mode. However, single block stop operation is not performed when the single block mode is disabled with variable #3003.
Note 3: When the feed hold is disabled, if the feed hold button is pressed then released, the machine does not stop; program execution continues and the machine stops at the first block where feed hold is enabled. The feed hold lamp is ON.
Note 4: When feedrate override is disabled, an override of 100% is always applied regardless of the setting of the feedrate override.
Note 5: When exact stop check is disabled, no exact stop check is

ma□e e□en in blo□□□ in□l□□in□ □o□e □□i□□ □o no□ □er□orm □□□in□.

- **□□mber o□ma□□ine□ □ar□□** The number of machined parts can be read and written.

Variable □o.	□□n□ion
#3901	□umber of machined parts

- **□o□al in□orma□ion**

  Modal information specified in bloc□s up to the immediately preceding bloc□can be read.

Variable □o.	□□n□ion
#4001	□roup 1 （□00, □01, □02, □03, □□3, □□4, □□0, □□1, □□2, □□3, □□4, □□5, □□□, □□□, □□9, □110, □111, □112, □113, □114, □115, □134, □135, □13□, □13□, □13□, □139）
#4002	□roup 2（□1□, □1□, □19）
#4003	□roup 3（□90, □91）
#4005	□roup 5（□94, □95）
#400□	□roup □（□20, □21）
#400□	□roup □（□40, □41, □42）
#400□	□roup □（□43, □44, □49）
#4010	□roup 10（□9□, □99）
#4014	□roup 14（□54, □55, □5□, □5□, □5□, □59）
#410□	□ code
#4109	□ code
#4111	□ code
#4113	M code
#4114	□loc□se□uence number
#4115	□rogram name
#4119	□ code
#4120	T code

- **C□rren□□o□i□ion** □osition information can be read.

Variable □o.	□□n□ion	□ea□ □□rin□ mo□emen□
#5001~#5005	Wor□piece coordinate system bloc□ end point (tool compensation value not included)	Enabled
#5021~#5025	Machine coordinate system current position(tool compensation value	□isabled

	included)	
#5041~#5045	Workpiece coordinate system current position (tool compensation value included)	Disabled
#5061~#5065	Workpiece coordinate system skip signal position (tool compensation value included)	Enabled
#5081~#5085	Tool length compensation value	Disabled

Note 1: The first digit (from 1 to 5) represents an axis number.

Note 2: The tool length compensation value currently used for execution rather than the immediately preceding tool compensation value is held in variables #5081~#5085.

- **Workpiece coordinate system compensation value**

Workpiece coordinate system compensation value can be read and written.

Variable No.	Function
#5201~#5205	The first to the fifth axes external workpiece zero point offset value
#5221~#5225	The first to the fifth axes G54 workpiece zero point offset value
#5241~#5245	The first to the fifth axes G55 workpiece zero point offset value
#5261~#5265	The first to the fifth axes G56 workpiece zero point offset value
#5281~#5285	The first to the fifth axes G57 workpiece zero point offset value
#5301~#5305	The first to the fifth axes G58 workpiece zero point offset value
#5321~#5325	The first to the fifth axes G59 workpiece zero point offset value

5.3 Arithmetic and Logic Operation

- Macro programs in both traditional G65 format and statement format are compatible with GSK980MDa.

  Users can alternatively select one of them for programming. This makes programming more convenient and flexible.

- Please strictly observe the formats and specifications in the following "Arithmetic and Logic Operation" table.

Arithmetic and Logic Operation

Function	Statement format	Traditional G65 format	Remark
Definition, assignment	#i = #j	G65 H01 P#i Q#j	
Sum Subtraction	#i = #j + #k #i = #j - #k	G65 H02 P#i Q#j R#k G65 H03 P#i Q#j R#k	Logic operation is performed on binary

Multiplication □ivision	#i □#□□#□ #i □#□□#□	□□5 □4 □#i □#□□#□ □□5 □5 □#i □#□□#□	numbers bit by bit.
□□ □□□ □□□	#i □#□□□ #□ #i □#□□□□ #□ #i □#□□□□ #□	□□5 □11 □#i □#□□#□ □□5 □12 □#i □#□□#□ □□5 □13 □#i □#□□#□	
□□uare root □bsolute value □ounding off □ounding up □ounding down □ature logarithm E□ponential function	#I □□□□T ［#□］ #I □□□□ ［#□］ #I □□□□□□ ［#□］ #I □□□□ ［#□］ #I □□I□ ［#□］ #I □□□ ［#□］ #I □E□□ ［#□］	□□5 □21 □#i □#□ □□5 □22 □#i □#□ □□5 □23 □#i □#□ □□5 □24□#i □#□ □□5 □25 □#i □#□ □□5 □2□ □#i □#□ □□5 □2□ □#i □#□	
□ine □rcsine Cosine □rccosine Tangent □rctangent	#i □□I□ ［#□］ #i □□□I□ ［#□］□［#□］ #i □C□□ ［#□］ #i □□C□□ ［#□］ #i □T□□ ［#□］ #i □□T□□ ［#□］□［#□］	□□5 □31 □#i □#□ □□5 □32 □#i □#□ □□5 □33 □#i □#□ □□5 □34 □#i □#□ □□5 □35 □#i □#□ □□5 □3□ □#i □#□□#□	□n angle is specified in degree. 90 degrees and 30 minutes is represented as 90.5 degree.
Conversion from □C□ to □I□ Conversion from □I□ to □C□	#i □□I□ ［#□］ #i □□C□ ［#□］	□□5 □41 □#i □#□ □□5 □42 □#i □#□	□sed for the signal e□change to and from □□C.
□nconditional branch E□uals to branch □ot e□uals to branch □reater than branch □maller than branch □reater than or e□uals to branch □maller than or e□uals to branch	□□T□ #i I□ (#i E□ #□) □□T□ #□ I□ (#i □E #□) □□T□ #□ I□ (#i □T #□) □□T□ #□ I□ (#i □T #□) □□T□ #□ I□ (#i □E #□) □□T□ #□ I□ (#i □E #□) □□T□ #□	□□5 □□0 □#i □#□□#□ □□5 □□1 □#i □#□□#□ □□5 □□2 □#i □#□□#□ □□5 □□3 □#i □#□□#□ □□5 □□4 □#i □#□□#□ □□5 □□5 □#i □#□□#□ □□5 □□□ □#i □#□□#□	□lease note that #□ is the s□ip signal in macro statement and □#i is the s□ip signal in traditional □□5□ format.
□ser alarm	□one	□□5 □99 □#i	0≤P≤100

5.3.1 Tranditional Format

If traditional G65 H format is used for programming, only limited operations and jump command can be specified by it. The currently used H operation needs at most 3 operands, so the corresponding operation can be completed when the needed variables (or constants) are obtained in a block.

- **General format**

  □65 Hm P#i Q#j R#k ；

  m: 01～99 means operation command or jump command function

  #i: the name of variable that stored the operation result

  #j: operand 1; it can be constant.

#k: operand □; it can be constant.

□eaning: #i □#j ○ #k

└────── □perational sign, designated by Hm

(□□ample) G65 Hm P#100 □#101 □#10□□ □ #100 □#101 ○ #10□ ;

G65 Hm □#100 P#101 □15 □ □ #101 □15 ○ #100 ;

G65 Hm □#100 □□100 P#10□□ □ #10□ □#100 ○ □100 ;

□ote 1□G65 H □□o□ld □e □ommanded □rior to o□eration or j□m□ □ommand.

□ote □□□□en P □ode i□ □ommanded in G65 □lo□k□G65 P mean□ ma□ro □ro□ram □all. H mean□ ar□□ment. □o o□eration or j□m□ □ommand i□ □erformed.

□ote 3□□t mo□t □ de□imal n□m□er□ of t□e □on□tant de□imal □art □an □e o□tained for ro□ndin□. 3 di□it n□m□er□ □an □e di□□la□ed in t□e □indo□.

● **□ode f□n□tion e□□lanation**

(1) □ariable value assignment, #I ＝ #□

<u>G65 H□1 P#□ Q#□;</u>

(e□ample) G65 H01 P#101 □1□5; (#101 ＝ 1□5)

G65 H01 P#101 □#110; (#101 ＝ #110)

G65 H01 P#101 □□#10□; (#101 ＝ □#10□)

(□) □ddition operation #I ＝ #□ ＋ #□

<u>G65 H□□ P#□ Q#□ R#□;</u>

(e□ample) G65 H0□ P#101 □#10□ □15; (#101 ＝ #10□ ＋ 15)

G65 H0□ P#101 □#110 □#10□; (#101 ＝ #110 ＋ #10□)

(3) □ubtraction operation #I ＝ #□ － #□

<u>G65 H□3 P#□ Q#□ R#□;</u>

(e□ample) G65 H03 P#101 □#10□ □#103; (#101 ＝ #10□ － #103)

(□) □ultiplication operation #I ＝ #□□#□

<u>G65 H□□ P#□ Q#□ R#□;</u>

(e□ample) G65 H0□ P#101 □#10□ □#103; (#101 ＝ #10□□#103)

(5) □ivision operation #I ＝ #□□#□

<u>G65 H□5 P#□ Q#□ R#□;</u>

(e□ample) G65 H05 P#101 □#10□ □#103; (#101 ＝ #10□□#103)

□ote□T□e di□i□or #k □annot □e □□ot□er□i□e an alarm o□□□r□.

(6) □□ operation #I ＝ #□ □□ #□

<u>G65 H11 P#□ Q#□ R#□;</u>

(e□ample) G65 H11 P#101 □#10□ □#103; (#101 ＝ #10□ □□ #103)

(□) □□□ operation #I ＝ #□ □□□ #□

<u>G65 H1□ P#□ Q#□ R#□;</u>

(e□ample) G65 H1□ P#101 □#10□ □#103; (#101 ＝ #10□ □□□ #103)

(□) □□□ operation #I ＝ #□ □□□ #□

G65 H13 P#□ Q#□ R#□；

(e□ample) G65 H13 P#101 □#10□ □#103； (#101 ＝ #10□ □□□ #103)

(9) □□uare root □□= √□□

G65 H□1 P#□ Q#□；

(e□ample) G65 H□1 P#101 □#10□； (#101 ＝ √□1□2)

□ote□t□e radi□and #□ □annot □e ne□ati□e□ot□er□i□e□an alarm o□□□r□.

(10) □bsolute value #I ＝ ∣#□∣

G65 H□□ P#□ Q#□；

(e□ample) G65 H□□ P#101 □□10□； (#101 ＝ ∣□10□∣ #101□10□)

(11) □ounding off #I ＝ □□□□□□#□□（□□□□□ off the first decimal）

G65 H□3 P#□ Q#□；

(e□ample) G65 H□3 P#101 □1.□359； (#101 ＝ 1.□359 #101□1)

(1□) □ounding up #I ＝ □□P□#□□

G65 H□□ P#□ Q#□；

(13) □ounding down #I ＝ □I□ □#□□

G65 H□5 P#□ Q#□；

□ith □□□, when the absolute value of the integer produced by an operation on a number is greater than the absolute value of the original number, such an operation is referred to as rounding up to an integer. □onversely, when the absolute value of the integer produced by an operation on a number is less than the absolute value of the original number, such an operation is referred to as rounding down to an integer. □e particular careful when handling negative numbers.

(□□ample) suppose that #1□1.□,#□□ □1.□

□hen #3□□□P□#1□is e□ecuted, □.0 is assigned to #3

□hen #3□□I□□#1□is e□ecuted, 1.0 is assigned to #3

□hen #3□□□P□#□□is e□ecuted, □□.0 is assigned to #3

□hen #3□□I□□#□□is e□ecuted, □1.0 is assigned to #3

(1□) □atural logarithm #I ＝ □□ □#□□

G65 H□6 P#□ Q#□；

(e□ample) G65 H□6 P#101 □#10□；（#101 ＝ □□□#10□□）

□ote□□ □en t□e antilo□arit□m #j i□ □ero or □maller□ot□er□i□e□an alarm i□ i□□□ed.

(15) □□ponential function #I ＝ □□P□#□□

G65 H□□ P#□ Q#□；

(e□ample) G65 H□□ P#101 □#10□；（#101 ＝ □□P □#10□□）

(16) Sine #I = SIN[#J] (unit: deg)

G65 H31 P#I Q#J;

(example) G65 H31 P#101 Q#103; (#101=SIN[#103])

(17) Arcsine #I = ASIN[#J]

G65 H32 P#I Q#J;

(example) G65 H32 P#101 Q#103; (#101=ASIN[#103])

Note 1: When the NAT bit of parameter No.015 is set to 0, the output range is 270°~90°;
When the NAT bit of parameter No.015 is set to 1, the output range is -90°~90°.
Note 2: ASIN operand cannot exceed the range -1~1, otherwise an alarm is issued.

(18) Arccosine #I = COS[#J] (unit: deg)

G65 H33 P#I Q#J;

(example) G65 H33 P#101 Q#103; (#101=COS[#103])

(19) Arccosine #I = ACOS[#J]

G65 H34 P#I Q#J;

(example) G65 H34 P#101 Q#103; (#101=ACOS[#103])

Note 1: ACOS operand cannot exceed the range -1~1, otherwise an alarm is issued.

(20) Tangent #I = TAN[#J] (deg)

G65 H35 P#I Q#J;

(example) G65 H35 P#101 Q#103; (#101=TAN[#103])

Note: #J cannot be equal to Kπ+π/2（K=0, ±1, ±2, ±3 …）, otherwise the result is wrong.

(21) Arctangent #I = ATAN [#J] / [#K] (unit: deg)

G65 H36 P#I Q#J R#K;

(example) G65 H36 P#101 Q#103 R3; (#101=ATAN [#103] /[3])

Note 1: When the NAT bit of parameter No.015 is set to 0, the output range is 0° ~ 360°
When the NAT bit of parameter No.015 is set to 1, the output range is -180° ~ 180°

(22) Conversion from BCD to BIN #I = BIN[#J]

G65 H41 P#I Q#J;

(example) G65 H41 P#101 Q#102; (#101 = BIN[#102])

(23) Conversion from BIN to BCD #I = BCD[#J]

G65 H42 P#I Q#J;

(example) G65 H42 P#101 Q#102; (#101 = BCD[#102])

(24) Unconditional branch

G65 H80 Pn; **Pn: sequence number**

(example) G65 H80 P120; (Go to N120 block)

(25) Equal to conditional branch

G65 H81 Q#I R#J Pn; **Pn: sequence number, can be variable**

(example) G65 H81 Q#101 R#102 P1000;

When #101 equals to #102, branch to N1000 block, or execut in order.

(26) Not equal to conditional branch

G65 H82 Q#I R#J Pn; **Pn: sequence number, can be variable**

(example) G65 H82 #101 #102 C1000;

When #101 does not equal to #102, branch to N1000 block, or execut in order.

(27) Greater than conditional branch

G65 H83 Q#I R#J Pn; **Pn: sequence number, variable**

(example) G65 H83 Q#101 R#102 P1000;

When #101 is greater than #102, branch to N1000 block; when #101≤#102, execut in order.

(28) Smaller than conditional branch

G65 H84 Q#I R#J Pn; **Pn: sequence number, variable**

(example) G65 H84 Q#101 R#102 P1000;

When #101 is smaller than #102, branch to N1000 block, or execut in order.

(29) Greater than or equals to conditional branch

G65 H85 Q#I R#J Pn; **Pn: sequence number, variable**

(example) G65 H85 Q#101 R#102 P1000;

When #101 is greater than or equals to #102, branch to N1000 block, or execut in order.

(30) Smaller than or equals to conditional branch

G65 H86 Q#I R#J Pn; **Pn: sequence number, variable**

(example) G65 H86 Q#101 R#102 P1000;

When #101 is smaller than or equals to #102, branch to N1000 block, or execut in order.

(31) P/S alarm issued

G65 H□□ Pn; **Pn: sequence number, variable (alarm No.=n +600)**

(example) G65 H□□ P15;

P/S custom alarm 615 is issued.

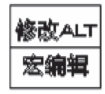
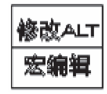
5.3.2 Macro Statement

The operations listed in Arithmetic and Logic Operation table can be executed in program. The expressions right to the operator contain constants and (or) variables that consisting of functions and operators. The variables #j and #k in the expression can be assigned as constants. The left variable (the first variable) can be assigned by expression. The macro statement is more intuitive, convienent and flexible. It can perform compound operation and multinesting. Sometimes, a macro statement is equal to several tranditional G65H macro programs.

- **General format**

  Please refer the statement format in the Arithmetic and Logic Operation table for editing macro statement.

- **□acro program e□iting**

  In program editing mode or □ID mode□b□pressing   ke□□ macro editing state can be s□itched or inserted□

□ifferences of two states	Automatic space	Processing of letter □	Input of special signs
Insert state	□hen editing□spaces are automaticall□ added to identif□the □ords□	Press □ to s□itch□cop□□delete programs	□pecial signs cannot be input
□acro editing state	space are not automaticall□added	Input as a letter □□□	□pecial signs can be input

- **□□planations**

  1□Angular unit

  The angular units of function □IN□C□□□A□IN□AC□□□TAN and ATAN are degree□□or example□□0□30 ´ means 90.5 degree.

  2, ARCSIN # i=ASIN[#j]

  i. the solution ranges are as indicated below

  when the NAT bit of parameter No.015 is set to 0: 270°~ 90°

  when the NAT bit of parameter No.015 is set to 1: -90°~ 90°

  ii. when the #j is beyond the range of -1 to 1, P/S alarm is issued.

  iii. a constant can be used instead of the #j variable.

  3, ARCCOS # i =ACOS[#j]

  i. the solution ranges from 180°~ 0°

  ii. when the #j is beyond the range of -1 to 1, P/S alarm is issued.

  iii. a constant can be used instead of the #j variable.

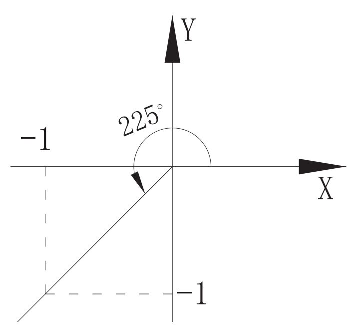
  4, ARCTAN #i=ATAN[#j]/[#k]

  Specify the lengths of two sides and separate them by a slash /.

  The solution ranges are as follows:

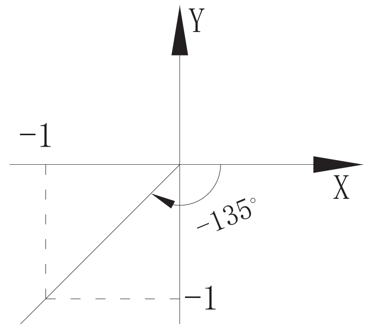
  When the NAT bit of parameter No.015 is set to 0: 0°~ 360°

  [Example] when #1=ATAN[-1]/[-1] is specified, #1=225°



  When the NAT bit of parameter No.015 is set to 1: -180°~ 180°

  [Example] when #1=ATAN[-1]/[-1] is specified, #1=-135°



ii. A constant can be used instead of the # j variable.

5. Natural logarithm #i=□N[#j]
   i. Note that the relative error may be greater than 10^{-8}.
   ii. When the antilogarithm #j is □ero or smaller, P/S alarm is issued.
   iii . A constant can be used instead of the #j variable.

6. Exponential function #i=E□P[#j]
   i. Note that the relative error may be greater than 10^{-8} .
   ii. When the result of the operation exceeds 3.65□10^{47} □j is about 110□, an overflow occurs and P/S alarm is issued.
   iii. A constant can be used instead of the # j variable.

7, RO□N□ function

When the RO□N□ function is included in an arithmetic or logic operation command, I□ statement, or W□I□E statement, the RO□N□ function rounds off at the first decimal place.

□□□□ □□□

When #1=RO□N□[#2] is executed where #2=1.2345 the value of variable #1 is 1.0.

When the RO□N□ function is used in NC statement address, the RO□N□ function rounds off the specified value according to the least input increment of the address.

8. Rounding up and down to an integer

With CNC, when the absolute value of the integer produced by an operation on a number is greater than the absolute value of the original number, such an operation is referred to as rounding up to an integer. Conversely, when the absolute value of the integer produced by an operation on a number is less than the absolute value of the original number, such an operation is referred to as rounding down to an integer. □e particular careful when handling negative numbers.

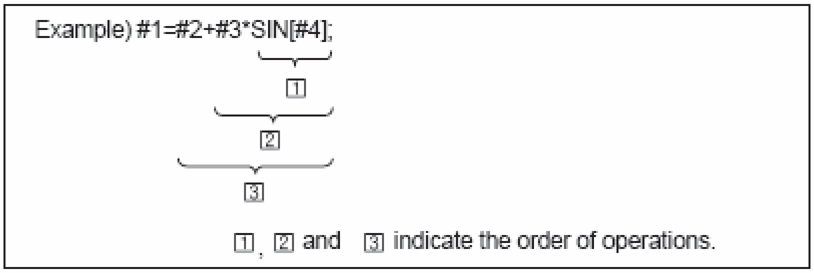
□□□□ □□□

Suppose that #1=1.2, #2= -1.2
When #3=□□P[#1] is executed, 2.0 is assigned to #3.
When #3=□I□[#1] is executed, 1.0 is assigned to #3.
When #3=□□P[#2] is executed, -2.0 is assigned to #3.
When #3=□I□[#2] is executed, -1.0 is assigned to #3.

□□□ □□□□□ □□□ □□□□□□□□

1. □unction
2. Operation such as multiplication and division（□, /, AN□）

3. Operation such as addition and subtraction (+, -, OR, XOR)



□□□□ □□□□□□□ □□□□

Brackets are used to change the order of operations. Brackets can be used to multinesting.

Note that the square bracket [,] is used to enclose an expression, the round bracket (,) is used in comments. When the priority is not defined, it is advised to use square bracket to enclose.

5.4 Branch and Repetition

In a program, the flow of control can be changed using the GOTO statement and IF statement. Three types of branch and repetition operations are used:

1. GOTO statement (unconditional branch)
2. IF statement (conditional branch: IF… THEN…)
3. WHILE statement (repetition WHILE…)

□□□□ □□□□□□□□□ □□□□ □□ □□ □□□□ □□□

Go to the block with sequence number n. when a sequence number out the range of 1~99999 is specified, an alarm is raised. A sequence number can also be specified using an expression.

□□□□ □□□GOTO n; n: sequence number (1~99999)

□□□□ □□: GOTO 1; GOTO #101;

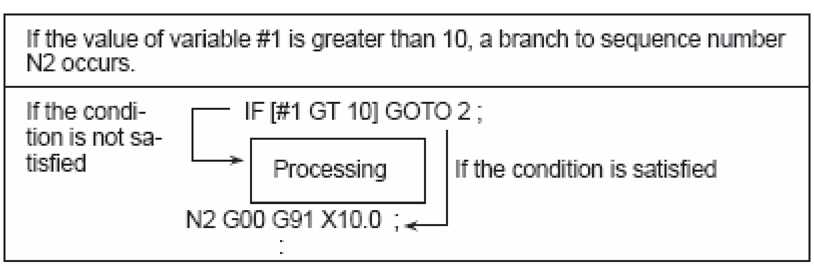
□□□□ □□□□□□□□□ □□□□ □□ □□□□ □□□

Specify a conditional expression after IF.

□□□□ □□□ □□□IF [conditional expression] GOTO n;

If the specified conditional expression is satisfied, a branch to sequence number n occurs. If the specified condition is not satisfied, the next block is executed.

□□□□ □□□



□□□□ □□□ □□: IF [conditional expression] THEN macro statement;

If the specified conditional expression is satisfied, a predetermined macro statement is executed. Only a single macro statement is executed.

IF[#1 EQ #2] THEN #3=0;

If the value of #1 and #2 are the same, 0 is assigned to #3; if not, no execution will be performed.

In 980[illegible]a, operators in the following table are used to compare two values to determine whether they are equal or one value is smaller or greater than the other value.

EQ or ==	Equal to (=)
NE or <>	Not equal to (≠)
GT or >	Greater than (>)
GE or >=	Greater than or equal to (≥)
LT or <	Less than (<)
LE or <=	Less than or equal to (≤)

IF [3<>2] GOTO 2; it means if 3 is not equal to 2, branch to N2 block

IF [#101>=7.22] THEN #101=SIN30; it means, if #101 is greater than 7.22, the expression after THEN is executed, i.e., assign Sin 30°to #101.

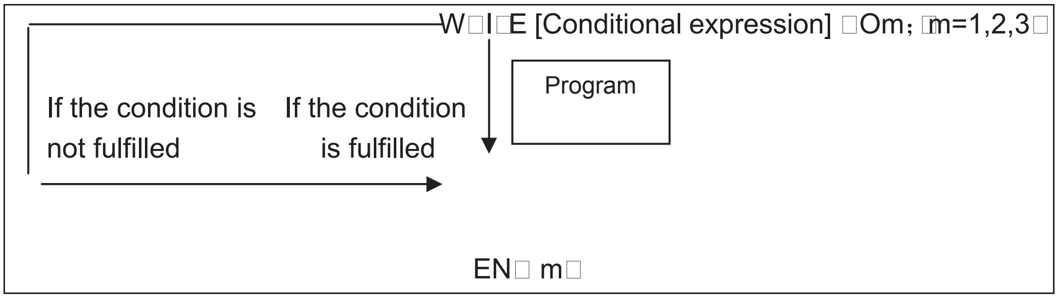
The sample program below finds the sum of number 1 to 10.

```
O9500
#101=0                      Initial value of the variable to hold the sum
#102=1                      initial value of the variable as an addend
N1 IF[#102 GT 10]GOTO 2 ;   Branch to N2 when the addend is greater than 10
#101= #101+#102 ;           calculation to find the sum
#102= #102+1 ;              Next addend
GOTO 1 ;                    Branch to N1
N2 M30 ;                    End of program; Sum of number 1 to 10
```

□□□ □□□□□□□ (□ □□□ □□□□ □□□)

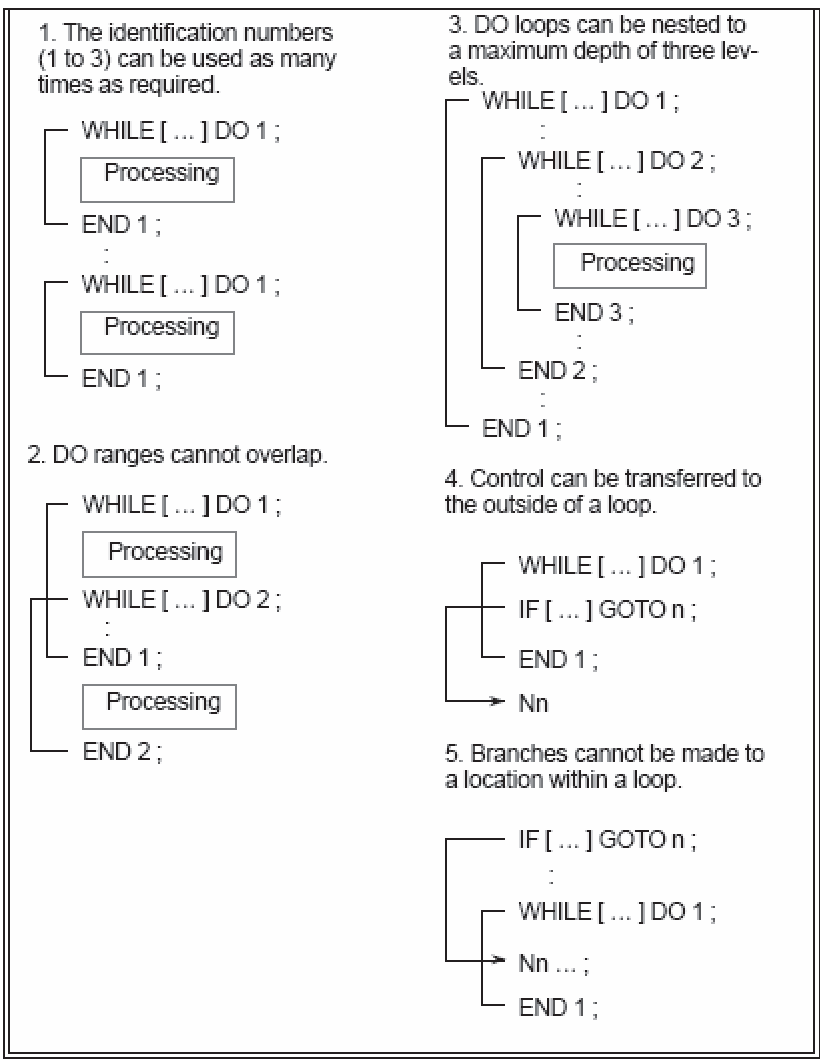
Specify a conditional expression after WHILE. While the specified condition is satisfied, the program from DO to END is executed. If not, program execution proceeds to the block after END.

□□□□ □□:



□□□□□□□□□: While the specified condition is fulfilled, the program from DO to END after WHILE is executed. If the specified condition is not fulfilled, program execution proceeds to the block after END. The same format as the IF statement applies. A number after DO and a number after END are identification numbers for specifying the range of execution. The number 1, 2, and 3 can be used. When a number other than 1, 2, and 3 is used, P/S alarm occurs.

□□□□□□: The identification number (1 to 3) in a DO, END loop can be used as many times as desired. Note, however, when a program includes crossing repetition loops (overlapped DO ranges), P/S alarm occurs.



5.5 Macro Statement and NC statement

The following blocks are referred to as macro statements:

- Blocks containing arithmetic or logic operation (=).
- Blocks containing a controlling statement (such as GOTO, DO, END).
- Blocks containing a macro call command. (such as G65, G66)

Blocks other than macro statements are referred to as NC statement.

[illegible]

Custom macro program are similar to subprogram. They can be edited, registered and used in the same way as subprogram. M98 can call a custom macro program, but cannot pass arguments.

Usually, the macro program is provided by tool builders, but it can also be programmed by customers. It is not necessary for the customers to remermber all related commands in macro programs besides codes that call macro programs.

[illegible]

- [illegible]

In cutter compensation C mode (G41, G42), in order to calculate the transmission point, NC prereads the next block. The processing way is not the same as general NC statement.

When a macro statement is executed as a single block, it is the block that does not involve movement. And, [illegible] (strictly speaking, such block involves 0 distance of movement).

- Jump (GOTO, DO, END)

  In cutter compensation C mode, when jump command (GOTO, DO, END) is specified, P/S alarm occurs.

- When the move command adopts variables

In cutter compensation C, when the move command (such as G01, X#101) adopts variables, P/S alarm occurs. Because cutter compensation C mode is block preread mode, the end point of the next block is essential for calculating the current transmission point position. Specifying X#101 (an unknown data) does not enable a correct calculation of the current transmission point.

- [illegible]

In MDI mode, macro programs can be specified, but macro program call cannot be executed.

- [illegible]

A "/" appearing in the middle of an (expression) (enclosed in brackets [ ] on the right-hand side of an arithmetic expression) is regarded as a division operator, it is not regarded as the specified for an optional block skip code.

- [illegible]

A reset operation clears any called states of custom macro programs and subprograms, and cursor returns to the first block of the main program.

□□□□□□□□ □□□□□□□□ □□□ □□□□□□□□□□

6.1 Application for Cutter Radius Compensation

□□□□ □□□□

Generally, the parts machining process is programmed according to parts drawing in one point on a tool. As for the tool used actually, because of the processing or other requirement, the tool is not an ideal point, but an arc only. The position offset exists between actual cutting point and ideal point when the cutting feed is performed. It may cause over cut or undercut, so the part accuracy will be affected. So, the cutter radius compensation can be used to improve the part accuracy in machining.

The path of part figure can be shifted by a cutter radius, which this method is called B type tool compensation, this is a simply method but the movement path of next block can be processed only after a block is performed, so the phenomenon as over cutting will be generated at the intersection point of two blocks.

In order to settle the above issues and eliminate the error, the Tool compensation C should be setup. When a block is read in, the tool compensation C is not performed immediately but the next block is read in again. Corresponding movement path is calculated according to the point of intersection of two blocks (conjunction vector). The tool compensation C performs more accurate compensation in figure because two blocks are read for processing in advance. See the Fig. 6-1

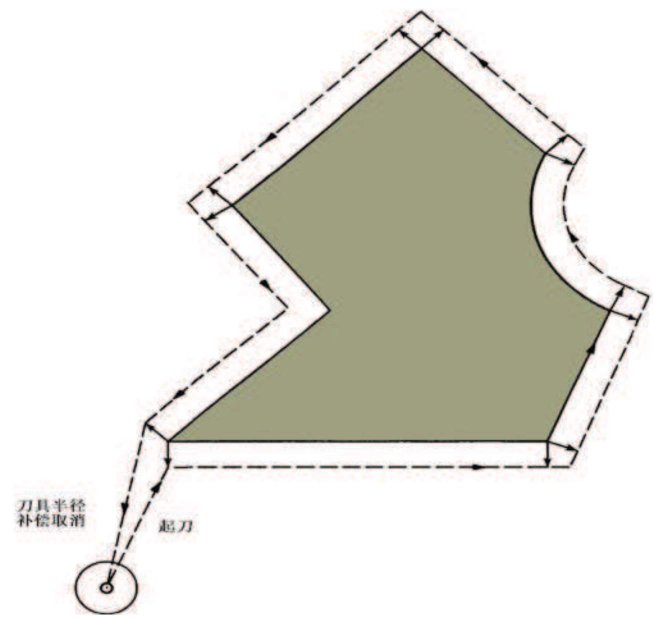


Fig.6-1 C type cutter radius compensation

□□□□ □□□ □□□□□□□□ □□□□ □□□□□□

The radius value of each tool should be set before tool compensation C is applied. Tool radius compensation value is set in the OFFSET page (table 6-1), this page contains tool geometric radius and tool radius wear. There into, D is the tool compensation value, when the bit 1 of bit parameter No.003 is 1, the D is compensation value input by diameter. If the bit 1 of bit parameter No.003 is 0, the D is compensation value input by radius. The following explanations are all indicated in radius compensation value if not especially pointed out.

Table 6-1 Display page for CNC cutter radius compensation value

□□□	□□□□ □□□□(□)	□ □□□□□ (□)	□□□□ □□□□(□)	□ □□□□□ (□)
001	20.020	0.030	5.000	0.020
002	10.020	0.123	0.500	0.030
□	□	□	□	□

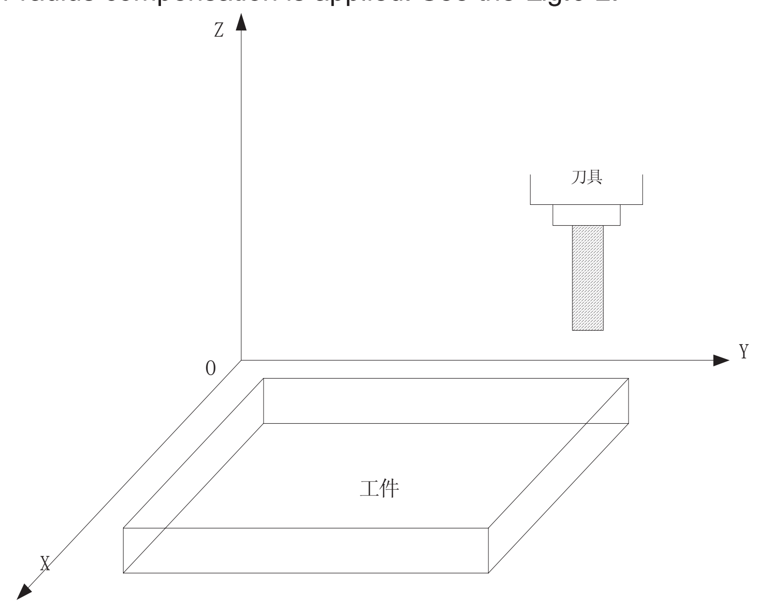
□□□□ □□□ □ □□□ □□□□ □□

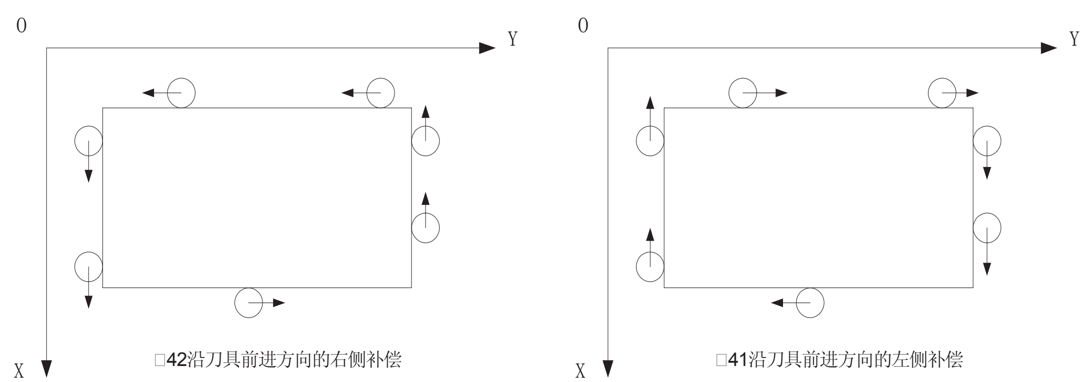
$$\begin{Bmatrix} G17 \\ □□□ \\ □□□ \end{Bmatrix} \begin{Bmatrix} □□□ \\ □□□ \\ □□□ \end{Bmatrix} \begin{Bmatrix} □□□ \\ □□□ \end{Bmatrix} \quad □□ \quad □\underline{□} \quad □\underline{□} \quad □\underline{□} \;;$$

□□□ □ □□□□	□□□□□□□□□□	□□□ □□□□
□17	Offset plane selection command □□□ plane□	See the □ig.6-2
□18	Offset plane selection command □□□ plane□	
□19	Offset plane selection command □□□ plane□	
□40	Cutter radius compensation cancellation	
□41	Cutter radius compensation left along advancing direction	
□42	Cutter radius compensation right along advancing direction	

□□□□ □□□ □□□□□□□□□ □□□□□□□

Tool compensation direction is determined according to the relative position of tool with work piece, when the cutter radius compensation is applied. See the □ig.6-2.





□42沿刀具前进方向的右侧补偿 □41沿刀具前进方向的左侧补偿

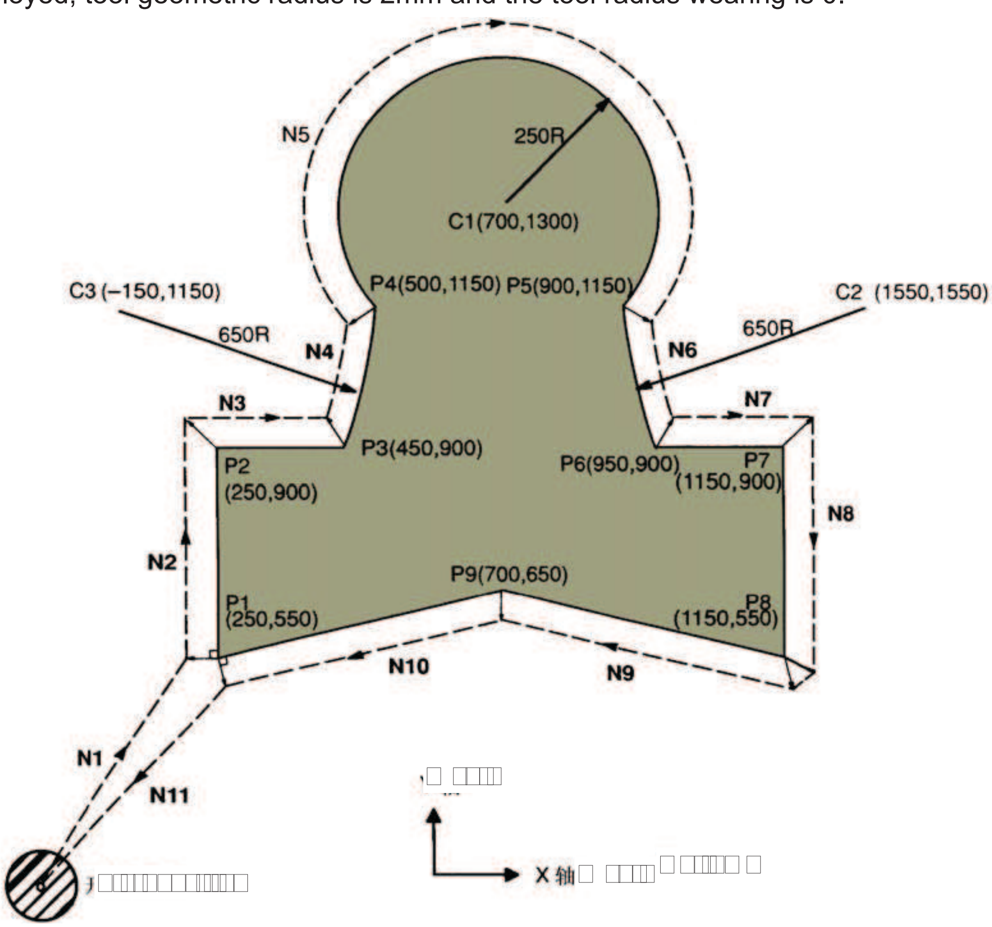
□□□ □□□□□

- In initial status CNC is in cutter radius compensation cancellation mode. CNC sets cutter radius compensation offset mode when the □41 or □42 command is executed. At the beginning of the compensation, the CNC reads two blocks in advance, the next block is stored in the cutter radius compensation buffer memory when a block is performed. When in Single mode, two blocks are read, after the end point of the 1st block is performed, it is stopped. Two blocks are read in advance in successive performance. So, there are a block being performed and two blocks behind it in CNC.
- Neither setup nor cancellation of the Tool compensation C can be performed in the □□I mode.
- The cutter radius compensation value can not be a negative, normally, the wearing value is negative □negative value indicates for wearing□
- Instead of □02 or □03, the setting or cancellation of cutter radius compensation can be commanded only by using □00 or □01, or the alarm occurs.
- CNC will cancel Tool compensation C mode when you press RESET key.
- Corresponding offset should be specified while the □40, □41 or □42 is specified in the block, or the alarm occurs.
- When cutter radius compensation is employed in main program and subprogram, the CNC should cancel compensation mode before calling or exiting sub-program □namely, before □98 or □99 is performed□, or the alarm occurs.

Cancel the compensation mode temporarily when □54-59, □28-31 and canned cycle command are executed. Restore the cutter radius compensation mode when the above commands are finished.

□□□ □□□□ □□ □□□□□□□□□□

The parts are machined in the coordinate system in □ig. 6-3. The tool compensation number □07 is employed, tool geometric radius is 2mm and the tool radius wearing is 0.



Perform tool setting in the mode of offset cancellation, after finishing the tool setting, and set the tool radius □ in O□□SET page.

Table.4-2

NO.	□eometric□□ □	Wearing□□□	□eometric□□□	Wearing□□□
01	□	□	□	□
□	□	□	□	□
07	□	□	2.000	0.000
08	□	□	□	□
□	□	□	□	□
32	□	□	□	□

Programs:

N0 □92 □0 □0 □0□ Tool are positioned at start position □0, □0 and □0 when the absolute coordinate system is specified

N1 □90 □17 □00 □41 □07 □250.0 □550.0□ Start-up cutter, the tool is shifted to the tool path by the distance specified in □07, geometric radius of □07 is set to 2.0mm, tool wearing 0, then the tool radius is 2mm.

N2 □01 □900.0 □150□ Specifies machining from P1 to P2

N3 □450.0□ Specifies machining from P2 to P3

N4 □03 □500.0 □1150.0 R650.0□ Specifies machining from P3 to P4

N5 □02 □900.0 R-250.0□ Specifies machining from P4 to P5

N6 □03 □950.0 □900.0 R650.0□ Specifies machining from P5 to P6

N7 □01 □1150.0□ Specifies machining from P6 to P7

N8 □550.0□ Specifies machining from P7 to P8

N9 □700.0 □650.0□ Specifies machining from P8 to P9

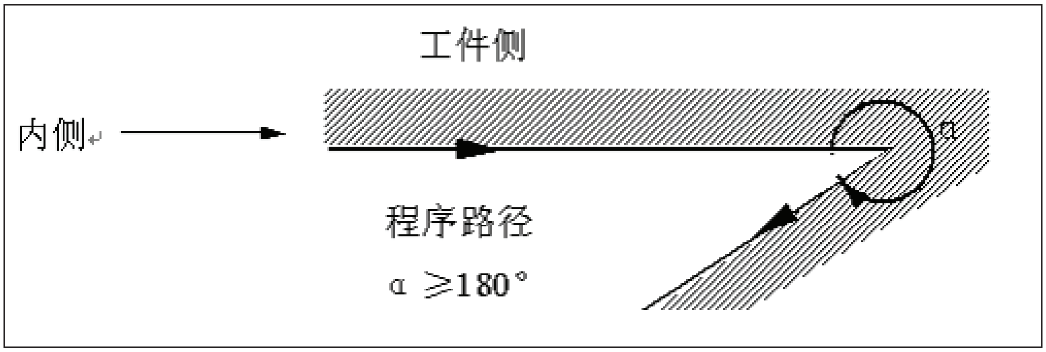
N10 □250.0 □550.0□ Specifies machining from P9 to P1

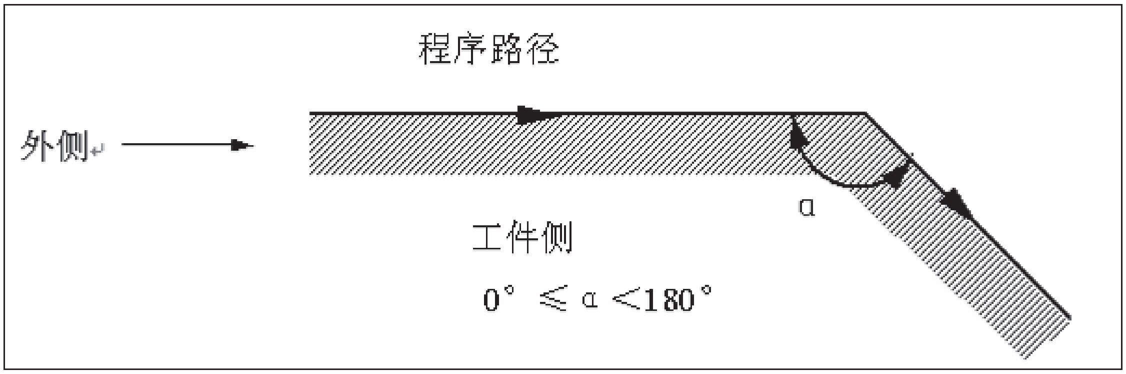
N11 □00 □40 □0 □0□ Cancels the offset mode, the tool is returned to the start position □□0, □0□

6.2 Offset Path Explanation for Cutter Radius Compensation

□□□□ □□□□□□□□□ □□□□□□□□□□□ □□□□□□□□□□

□Inner side□and □outer side□will be employed in the following explanations. When an angle of intersection created by tool paths specified by move commands for two blocks is over or e□ual to 180°, it is referred to as □inner side□. When the angle is between 0° and 180°, it is referred to as □outer side□.





There are 3 steps should be performed for cutter radius compensation: establishment, performing and cancellation.

The tool movement performed from offset cancellation mode to □41 or □42 command establishment is called tool compensation establishment □also called start-up□

(α≥□□□□)

1）□inear to linear

2）□inear to circular

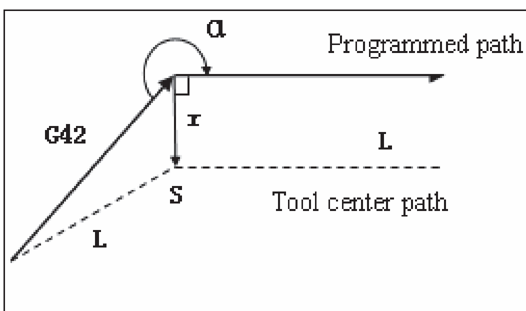


Fig.6-4a Linear to linear (start-up from inner side)

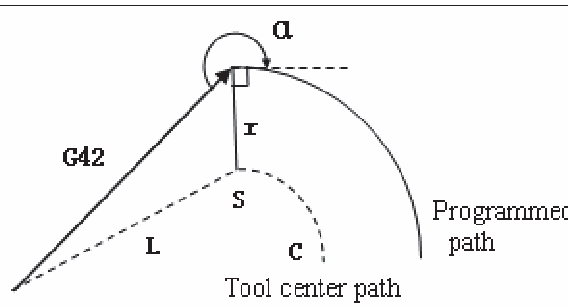


Fig.6-4b Linear to circular (start-up from inner side)

[illegible] (□□□□>α≥□□□)

1) □inear to linear　　　　2□□inear to linear

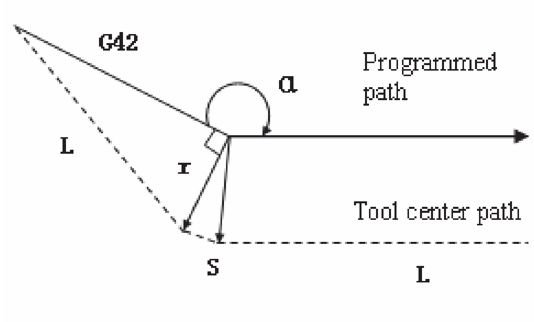


Fig.6-5a Linear to linear (start-up outside)

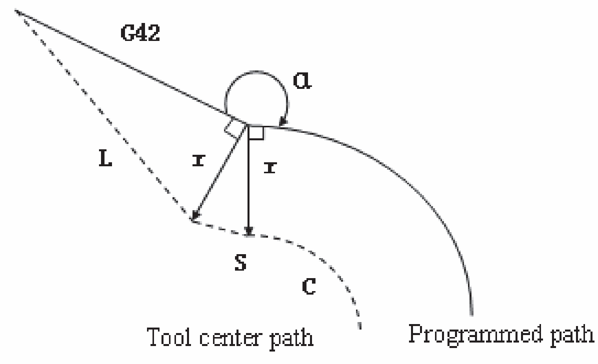


Fig.6-5b Linear to circular (Start-up outside)

[illegible] (α<□□□)

1) □inear to □inear　　　　2) □inear to circular

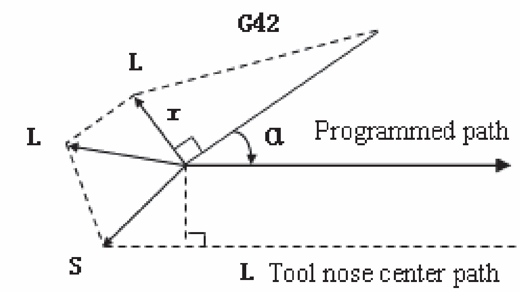


Fig.6-6a Linear to linear (start-up from outer side)

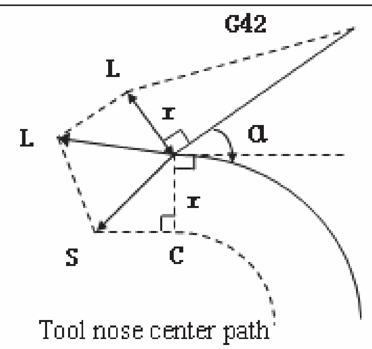


Fig.6-6b Linear to circular (start-up from outer side)

[illegible] (α≦□□)

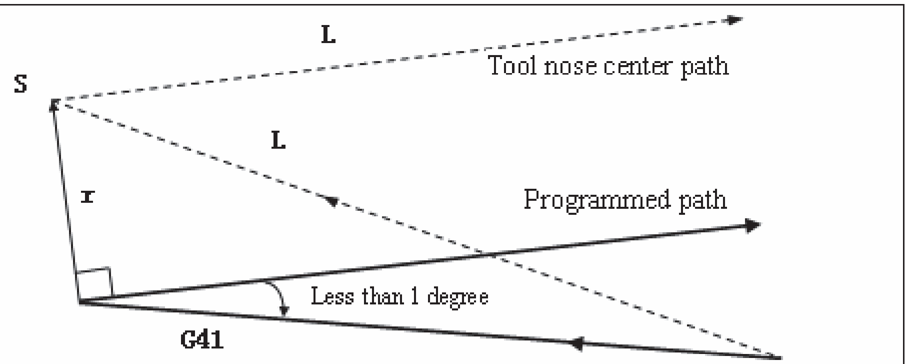


Fig.6-7 Linear to linear (the corner is less than 1 degree, start-up from outer side)

6.2.3 Tool movement in offset mode

The mode after setting the cutter radius compensation and before canceling the cutter radius compensation is called offset mode.

- **Offset path of invariable compensation direction in compensation mode**

1) Linear to linear　　　　2) Linear to circular

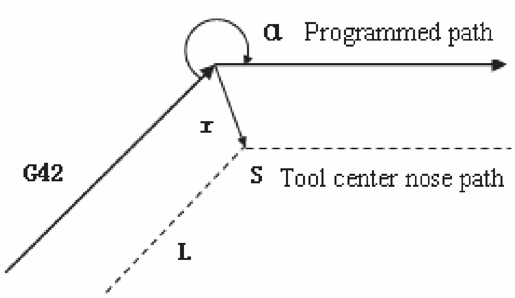


Fig.6-8a Linear to linear (inside movement)

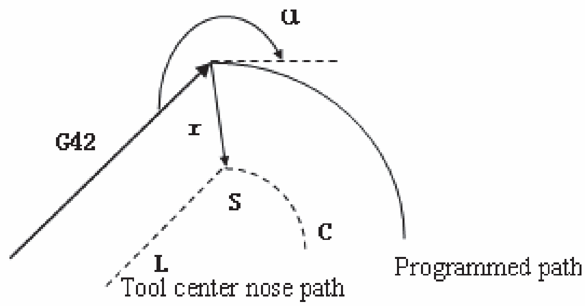


Fig.6-8b Linear to circular (inside movement)

3□□ircular to linear **□□□ircular to circular**

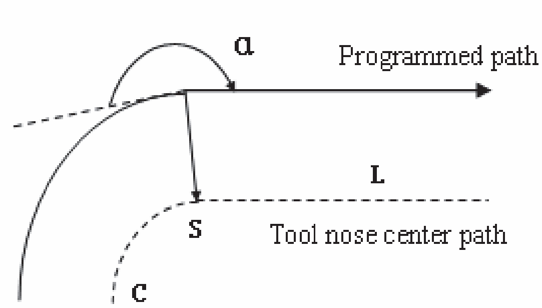


Fig.6-8c Circular to linear (inside movement)

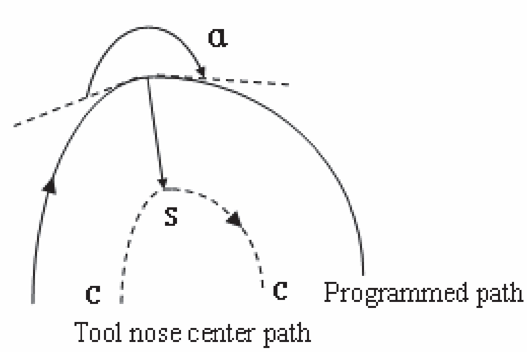


Fig.6-8d Circular to circular (inside movement)

□）□nner side machining less than 1 degree and compensation □ector amplification

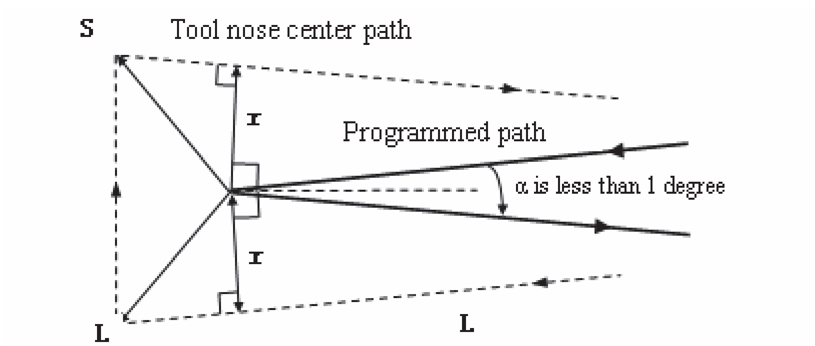


Fig.6-8e Linear to linear (corner is less than 1 degree, inside movement)

（b） □ove alon□ the outer of abtuse an□le □orner（□□□□>α≥□□□）

1）Linear to linear 2）Linear to circular

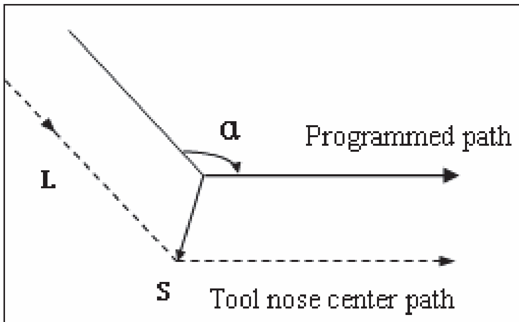


Fig.6-9a Linear to linear
(obtuse angle, outside movement)

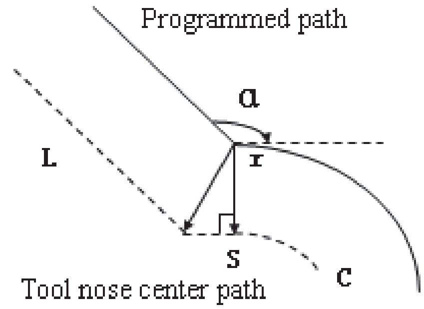


Fig.6-9b Linear to circular
(obtuse, outside movement)

□) Linear to linear

□) □ircular to circular

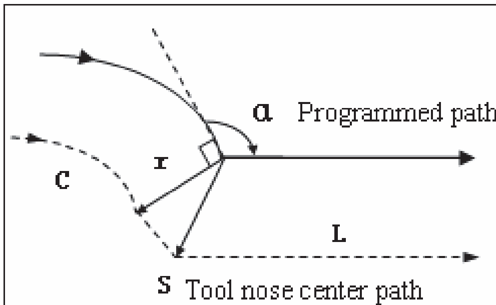


Fig.6-9c Circular to linear
(obtuse angle, outside movement)

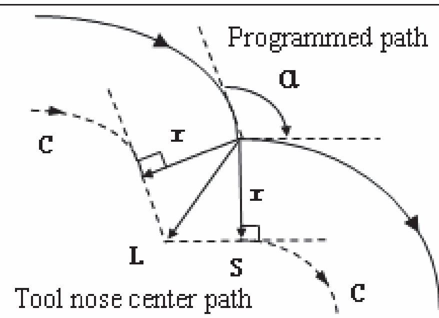


Fig.6-9d Circular to circular
(obtuse angle, outside movement)

(c) □ove alon□ the o□ter of ac□te an□le corner (α< □□□)

1) Linear to linear

2) Linear to circular

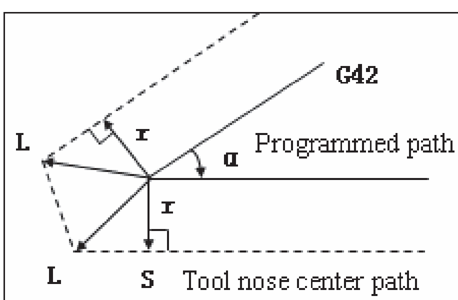


Fig.6-10a Linear to linear
(acute, movement outside)

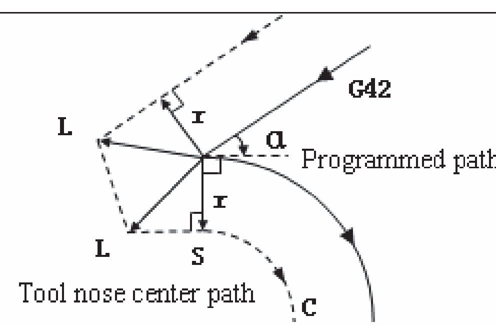


Fig.6-10b Linear to circular
(acute, movement outside)

□) □ircular to linear

□□ircular to circular

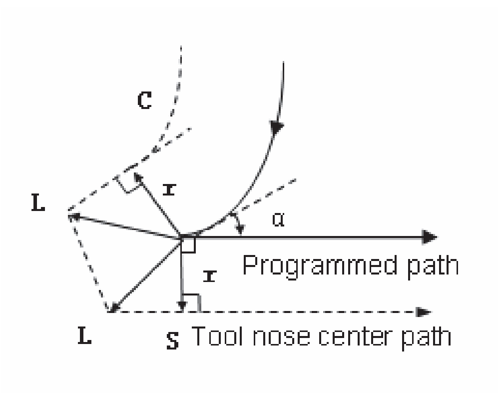


Fig.6-10c Circular to linear (acute, movement outside)

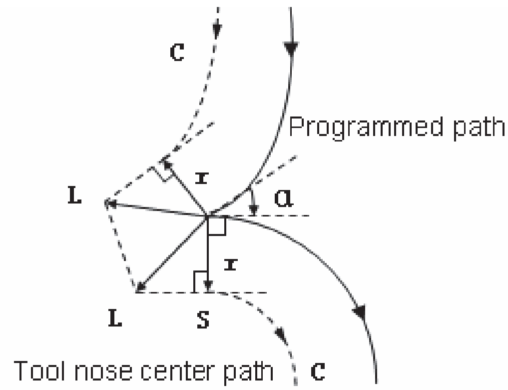


Fig.6-10d Circular to circular (acute, movement outside)

□d □□ hen it is e□ceptional

1□There is no intersection

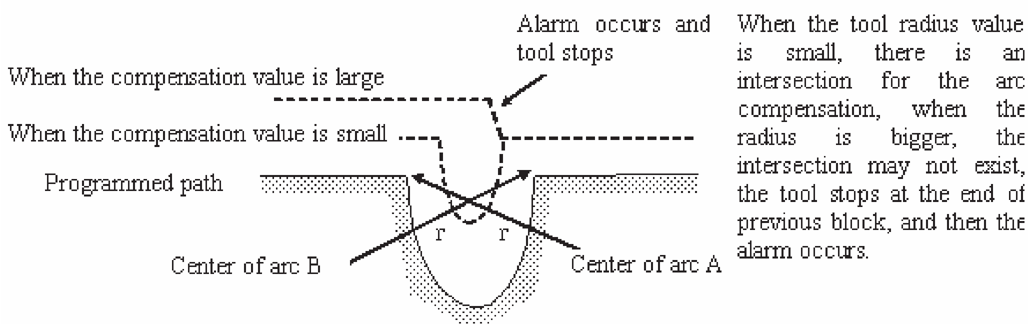


Fig.6-11 Exceptional ------There is no intersection after the path offset

2）The arc center is consistent to the start point or end point

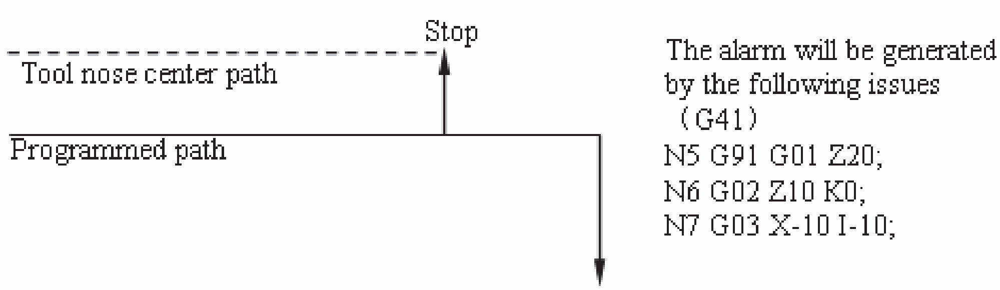


Fig.6-12 Center of arc is consistent to the start point or end point

- **Offset path □ith the compensation direction chan□ed in compensation mode**

.

The compensation direction can be changed in special occasion□but it cannot be changed at the beginning and the follo□ing bloc□. There are no inner side and outer side for the full compensation.

1）Linear to linear

2）Linear to □ircular

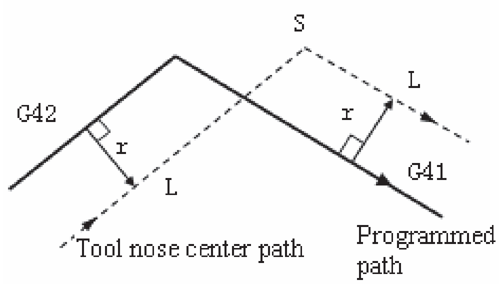


Fig.6-13a Linear to linear (compensation direction changed)

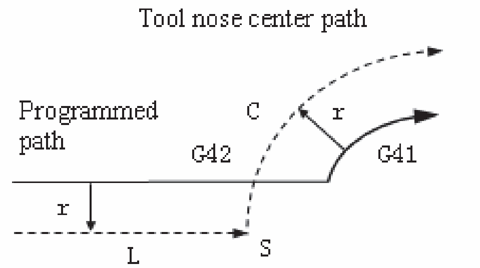
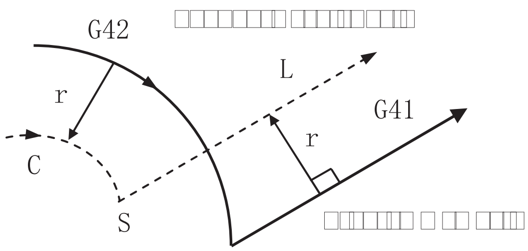
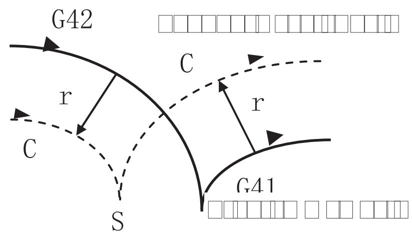


Fig.6-13b Linear to circular (compensation direction changed)

□）□ircular to linear



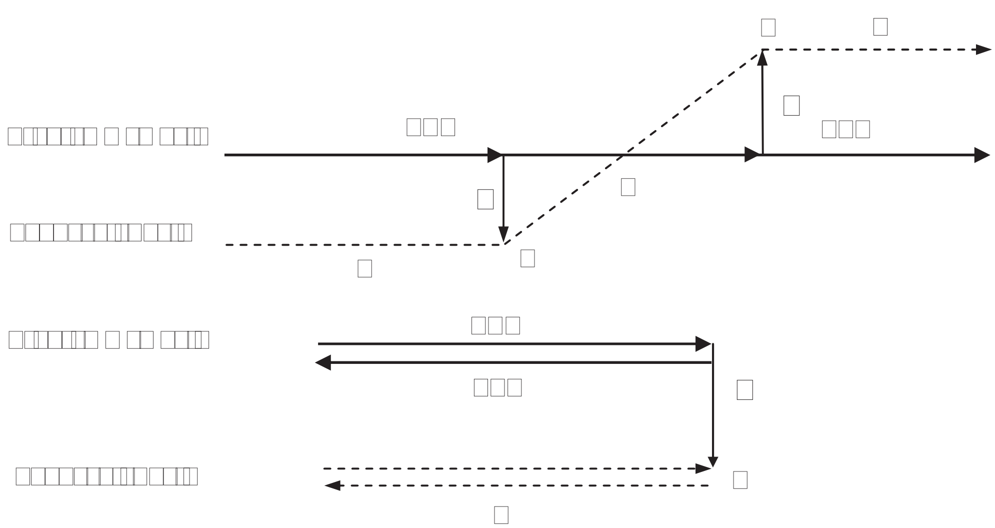
□）□ircular to □ircular



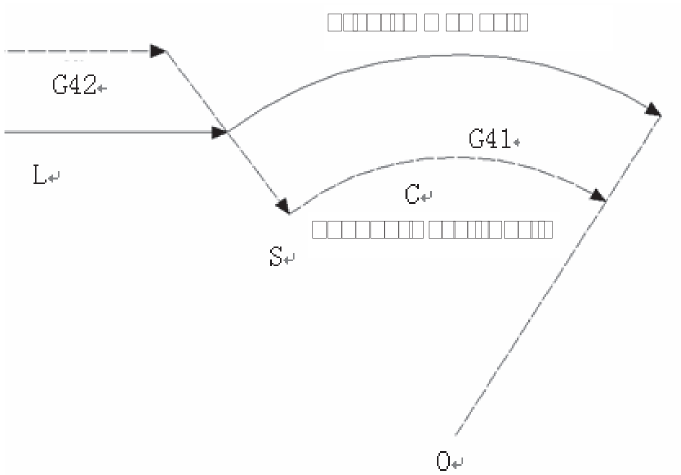
□）□hen there is no intersection if the compensation is normall□performed

□hen changing the offset direction from bloc□□to bloc□□using □□1 and □□2□if the intersection of the offset path is not re□uired□create the □ector □ertical to bloc□□at the start point of bloc□□.

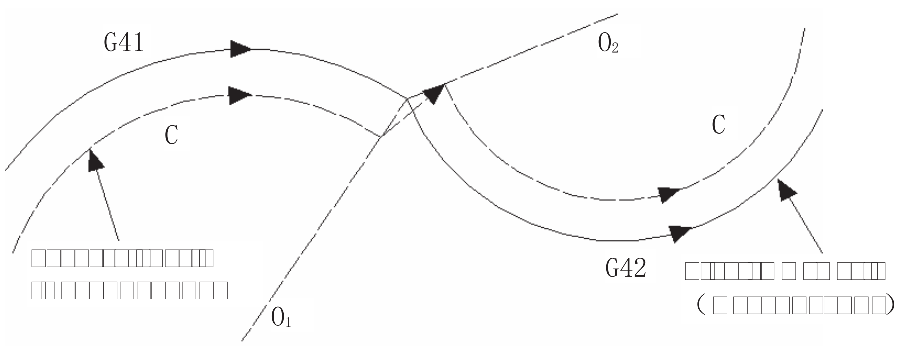
i □Linear to linear



ii□Linear to circular



iii□□ircular to circular



6.2.□ Tool operation in offset cancellation mode

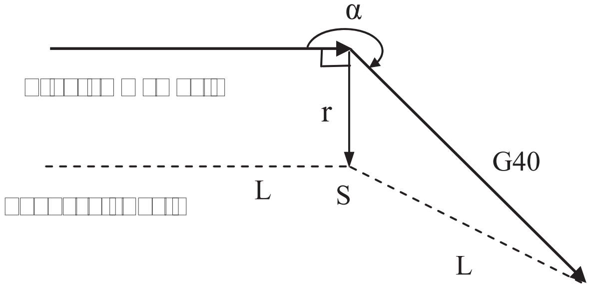
□hen the □□□ command is emplo□ed in bloc□ in compensation mode□the □□□ enters the compensation cancellation mode. This is called compensation cancellation.

The circular arc command □□□2 and □□□□ can not be emplo□ed □hen the cutter radius compensation □ is cancelled. □f the□ are commanded□alarm is generated and the operation is stopped

□t controls and performs this bloc□ and the bloc□s in the cutter radius compensation buffer memor□ in the compensation cancellation mode. □f the single bloc□ s□itch is turned on□it stops after e□ecuting a bloc□. The ne□t bloc□ is e□ecuted instead of reading it □hen the start □e□ is pressed again

□a□Tool movement alon□ an inner side of a corner （α≥□□□□）

1）Linear to linear　　　2）□ircular to linear



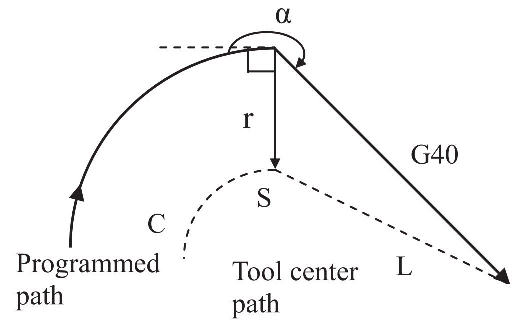
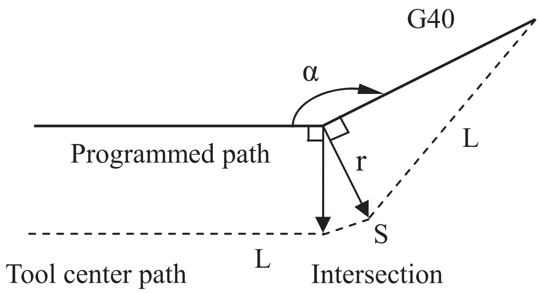


Fig.6-15b Circular to linear
(inner side, offset cancellation)

(b) Tool movement along the outside of a corner at an obtuse angle （180°>α≥90°）

1）Linear to linear

2）Circular to linear



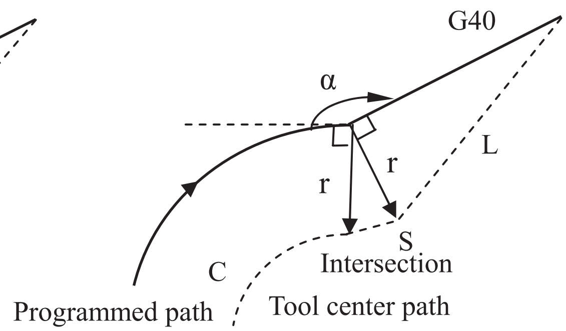


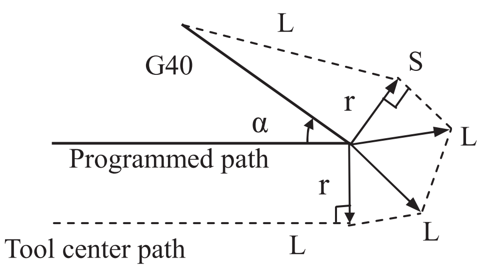
Fig.6-16a Circular to linear
(obtuse, outside, offset cancellation)

Fig.6-16b Circular to linear
(obtuse, outside, offset cancellation)

(c) Tool movement along the outside of a corner at an acute angle （90°>α≥1°）

1）Linear to linear

2）Circular to linear



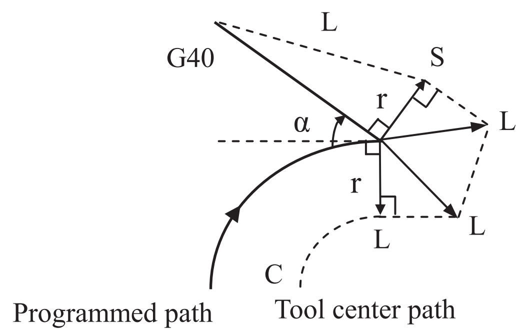
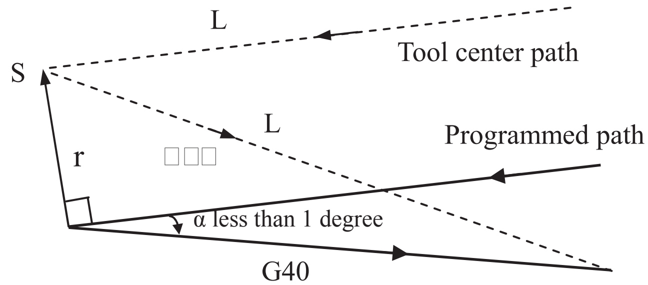


Fig.6-17aLinear to linear
(acute angle, outside, offset cancellation)

Fig.6-17b Linear to linear
(acute angle, outside, offset cancellation)

(d) Tool movement along the corner outside at an acute angle less than 1 degree (linear to linear（α<1°）



□□□□□ □□□□ □ □□□ □□ □□□□ □□□ □□ □□□ □ degree, outside, offset cancellation)

6.2.5 Interference check

Tool over cutting is called “interference”. The interference check function can check tool over cutting in advance. This interference check is performed even if the over cutting does not occur. However, all interference can not be checked by this function.

(1) Conditions for the interference

1) The direction of the tool path is different from that of the programmed path. (90 degrees to 270 degrees between these paths)

2) In addition to the condition above, the angle between the start point and end point of the tool center path is quite different from that between the start point and end point of the programmed path in circular machining (more than 180 degrees).

Example: Linear machining

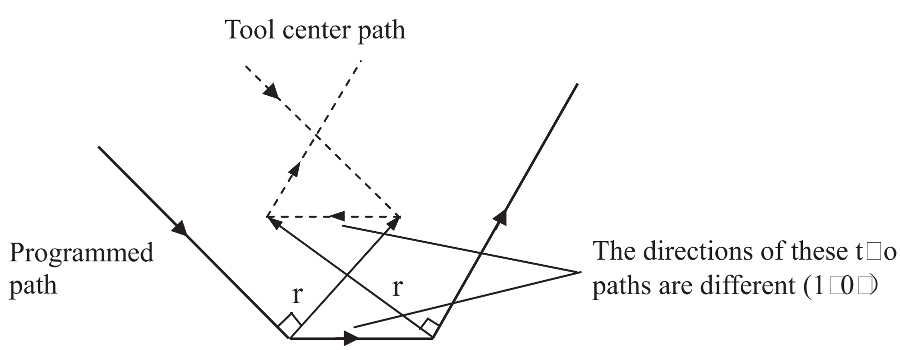


Fig.6-1□a □achining interference (1)

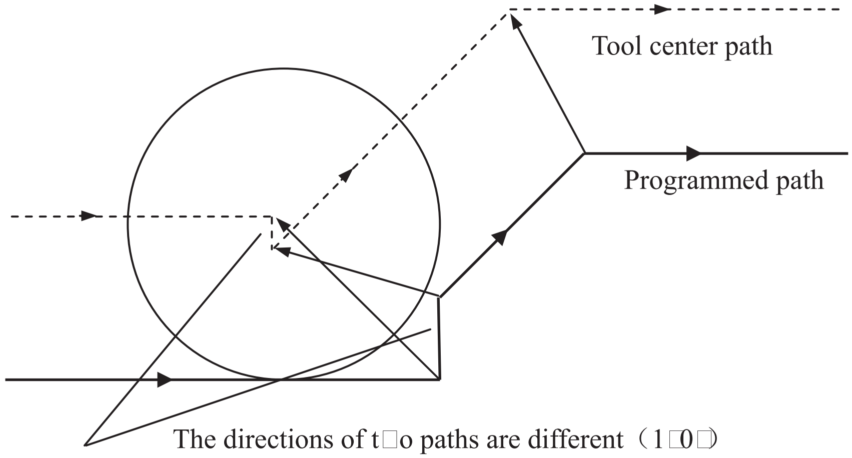


Fig.6-1□b □achining interference (□)

(□) □t□□r□ i□ no int□r□□r□nc□ actuall□□□ut it i□ tr□at□□ a□ int□r□□r□nc□□

1) The groove depth less than the compensation value

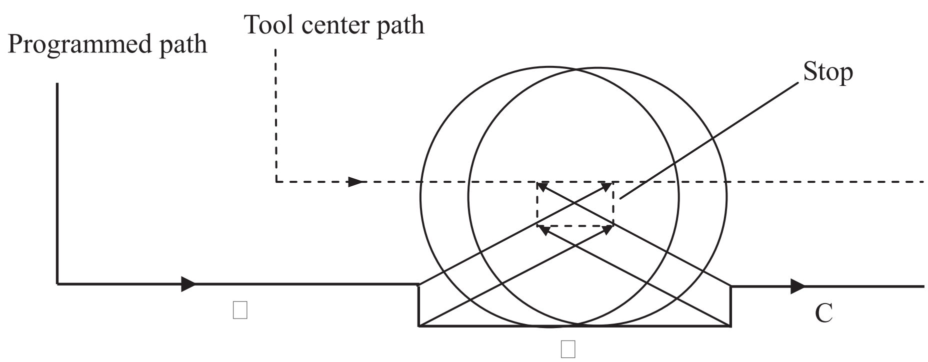


Fig.6-□0 □□ceptional case (1) treated as interference

There is no interference actually, but program direction in block □ is opposite to the cutter radius compensation path. The cutter stops, and the alarm occurs.

2) The groove depth less than compensation value

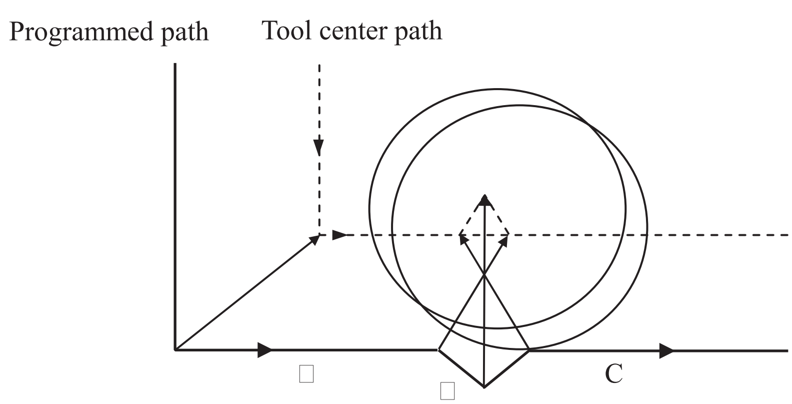


Fig.6-□1 □□ceptional case (□) treated as interference

There is no interference actually, but program direction in block □ is opposite to the cutter radius compensation path. The cutter stops, and the alarm occurs.

6.2.6 Command of compensation □ector cancel temporaril□

If the following commands G92, G28, G29, coordinate command selection G□□□G□9 and canned cycle are specified in compensation mode, the compensation vector is temporarily cancelled and then automatically restored after these commands are e□ecuted. Now, the temporary compensation vector cancellation is different to the compensation cancellation mode, tool is moved to the specified point by compensation vector cancellation from the intersection. □nd the tool moves to the intersection directly when the compensation mode restores.

- **Coordinate s□stem setting command □□2 and coordinate s□stem selection command □5□□□5□**

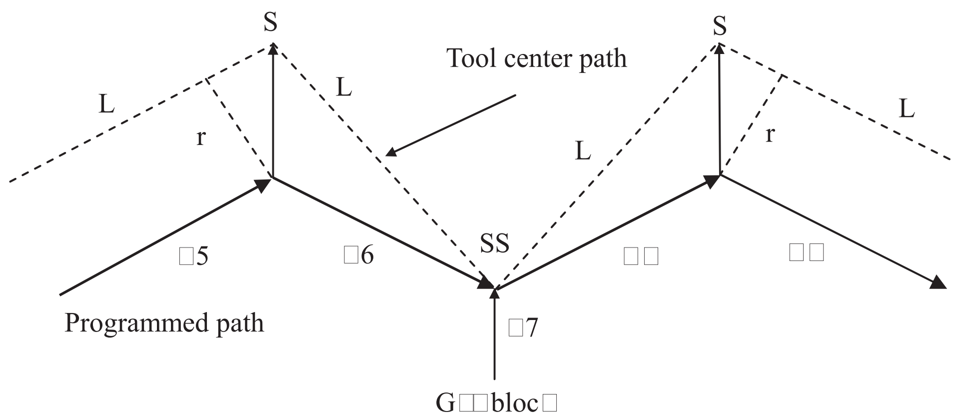
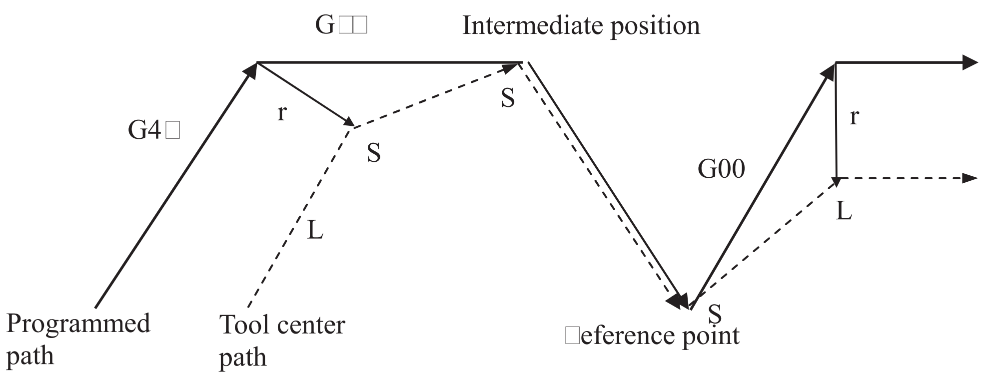


Fig.6-□□ Temporar□compensation □ector b□G□□

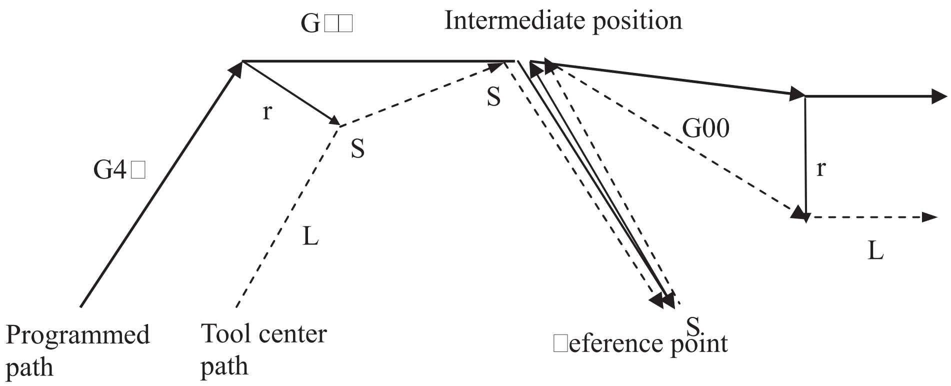
□ote: □□ is indicated as the point stopped for t□ice in □ingle □lock mode.

- **□utomatic return to the reference point □2□**

If G28 is specified in compensation mode, the compensation will be cancelled at an intermediate position. The compensation mode is automatically restored after the reference point is returned.



□ig.□2□ Temporarily cancel compensation vector by G28



□ig. □2□ G29 temporarily cancel compensation vector

- **Canned c□cle**

If the canned cycle command is specified in compensation mode, the compensation will be temporarily cancelled in the canned cycle operation 1. The compensation mode is automatically restored after the canned cycle is terminated.

6.2.□ Exceptional case

- **□hen the inner corner machining is less than tool radius**

□hen the inner corner machining is less than tool radius, the inner offset of a tool will cause over cut. The tool stops and alarm occurs after moving at the beginning or at the corner in previous block. □ut if the switch of “Single block” is □N, the tool will be stopped at the end of the previous block.

- **□hen a groo□e less than the tool diameter is machined**

□hen the tool center moves opposite to the direction of programmed path, the over cutting will be generated by the cutter radius compensation. Tool stops and alarm appears after moving at the beginning of previous block or at the corner.

- **□hen a step less than the tool radius is machined**

□hen a program contains a step which is an arc and less than tool radius, tool center path may form a opposite movement direction to the programmed path. So the first vector is ignored and it moves to the end of the second vector along a straight line. The program will be stopped for Single block mode, the cycle continues if it is not single block mode. The compensation will be e□ecuted correctly and no alarm will be generated if the step is a straight line. (□ut the uncut part is reserved.)

- **□hen the su□□program is contained in □ code**

CNC should be in compensation cancellation mode before calling the sub□program (namely, before the G98 is performed). □ffset can be applied after entering the sub□program, but the compensation cancellation should be applied before returning to the main□program (before M99), or the alarm occurs.

- **□hen compensation □alue is changed**

(a) Usually, the compensation value is changed when the tool change is performed in compensation cancellation mode. If the compensation value is changed in compensation mode, the new one is ineffective which is effective till the program is e□ecuted again.

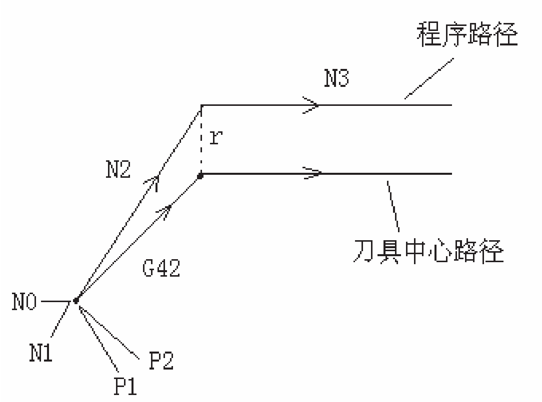
(b) If different compensation values are commanded in different blocks of a program, different compensation value will be compensated to the corresponding block. □ut if it is an arc, the alarm will be generated. □or details, refer to the following e□planation.

(c) about “arc data error in C type cutter radius compensation”.

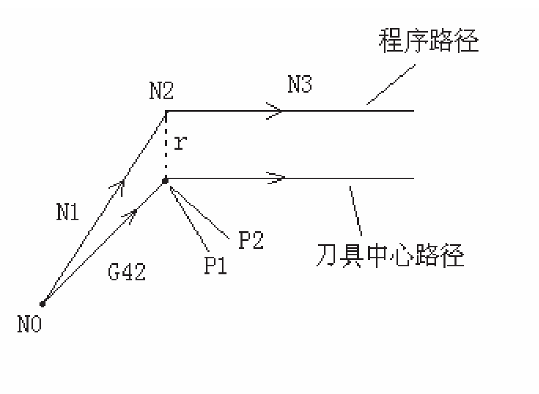
- **□hen the end point for the programming arc is not on the arc**

□hen the end point for the programming arc is not on the arc, the tool stops and the alarm information shows “end point is not on the arc”.

Two same points in the starting is shown an e□ample□



N0 G90 G00 □□□0 □□□0
N1 G91 G1 G□1 □0 □0 D1 □800 □ without moving
N2 G90 □0 □0
N□ □□0



The above□mentioned program may occur the “two same points” when starting, and the compensation may not perform. The transit point □1 between N0 and N1 and the transit point □2 between N1 and N2 are shared a same point.

N0 G90 G00 □□□0 □□□0
N1 G1 G□1 □0 □0 D1 □800
N2 G91 □0 □0 □ without moving
N□ □□0

The “last two same points” may occur when starting at the last program, in the case of the compensation has been performed. The section without moving which is regarded as the movement appro□imates to the □ero, so it is necessary to maintain the compensation amount. The transit point between N1 and N2 is □1, and the transit point between N2 and N□ is □2, □1 and □2 are shared a same point.

In the same way, in the compensation mode, if the “two same points” may occur, the compensation value will be maintained□in the retraction mode, the similar start mode is divided into “the previous two same points” and “the last two same points”

- **□he alarm and corresponding explanation of □Circular arc data error in cutter compensation C□**

(a) □he example of this alarm ma□ occur in a circle

□orgram e□ample：N0 G90 G00 □□□0 □□□0 □□0

N1 G01 G□2 □0 □0 D1 □800

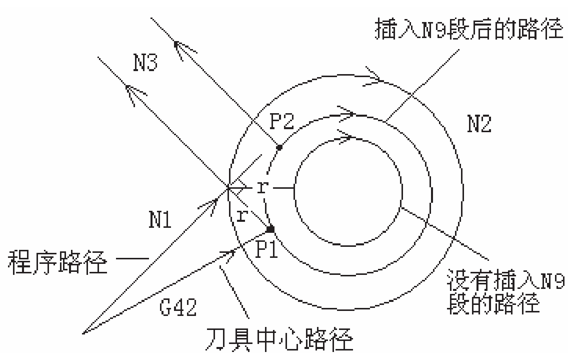
N2 G02 I□0

N□ G91 G01 □□□0 □□□0

程序路径：□rogrammed path

刀具中心路径：Tool center path

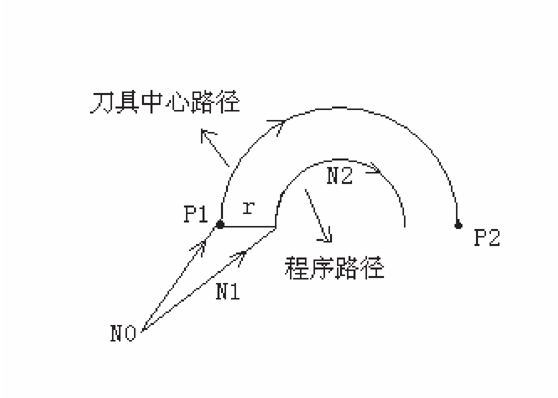
The transit point between straight line N1 and circular arc N2 is □1, the transit point between circular N2 and straight line N□ is □2, and the compensation radius is r, in this case, the circular after tool compensation is more than □□0□.



□fter a block (N9 G91 G0 □0 □0) (without moving) is inserted between N1 and N2 in the above□mentioned program, the “circular data error in cutter compensation C” may alarm.

□ecause the point after N9 inserted which is equal to the one of N1, namely, they are regarded as “two same points”. The transit point □1 is performed treating the “two same points”, the position of □1 is obviously differ from the above one which does not insert the N9 block. So the cut circular arc path by this transit pont is absolutely differing from the path to be machined, so the alamr is then generated□“circular arc data error in cutter compensation C”

(□) □he example for a non□circle ma□ occur:



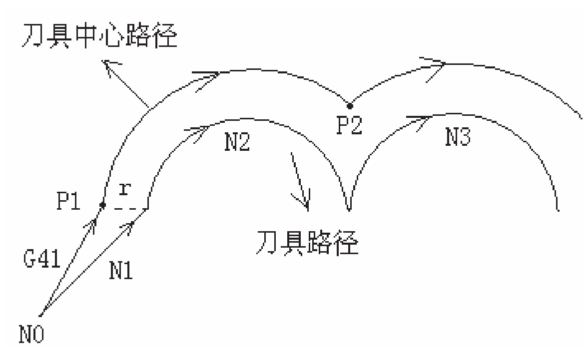
□rogram e□ample□N0 G90 G00 □□□0 □□□0 □□0
N1 G01 G□1 □0 □0 D1 □800
N2 G02 □□0 □2□

The □1 and □2 are the transit point of tool compensation as the left figure shown, wherein the "r" is compensation radius. This is a normal treatment mode for the straight line to circular arc.

The alarm may occur in terms of the following program
N0 G90 G00 □0 □0 □0
N1 G01 G□1 □0 □0 D1 □800 □ without moving, originally start
N2 G02 □□0 □2□

□ecause the N1 block does not a movement, namaly, it equals to the "two same points". The transit points □1 and □2 are performed based on the treatment of two same points (The path of two same points), so the circular arc path cut by this transit point obviously differs from the actual path to be machined, in this case, the "circular arc data error in cutter compensation C" may alarm.

(c) In the calculation of arc cutter compensation C□ this alarm ma□ issue if the compensation radius □ is modified.



□rogram e□ample□N0 G90 G00 □□□0 □□□0 □2□
N1 G01 G□1 □0 □0 D1 □800
N2 G02 □□0 □2□
N□ G02 □100 □2□

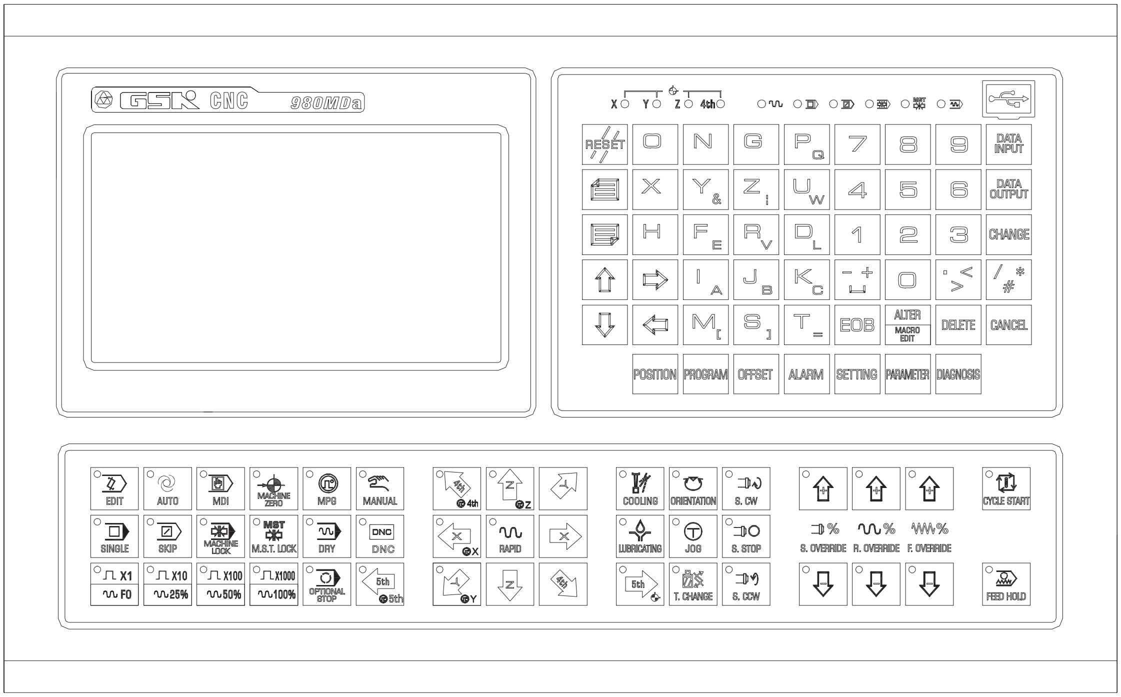
The left figure is shown the programmed path and the tool center path.

If the compensation radius D is changed in N□, for e□ample, the D2 is speicified in N□ block (the value of D2 is not equal to the one of D1), in this case, it is similar as (b), an alarm of the "circular arc data error in cutter compensation C" may occur.

VOLUME Ⅱ OPERATION

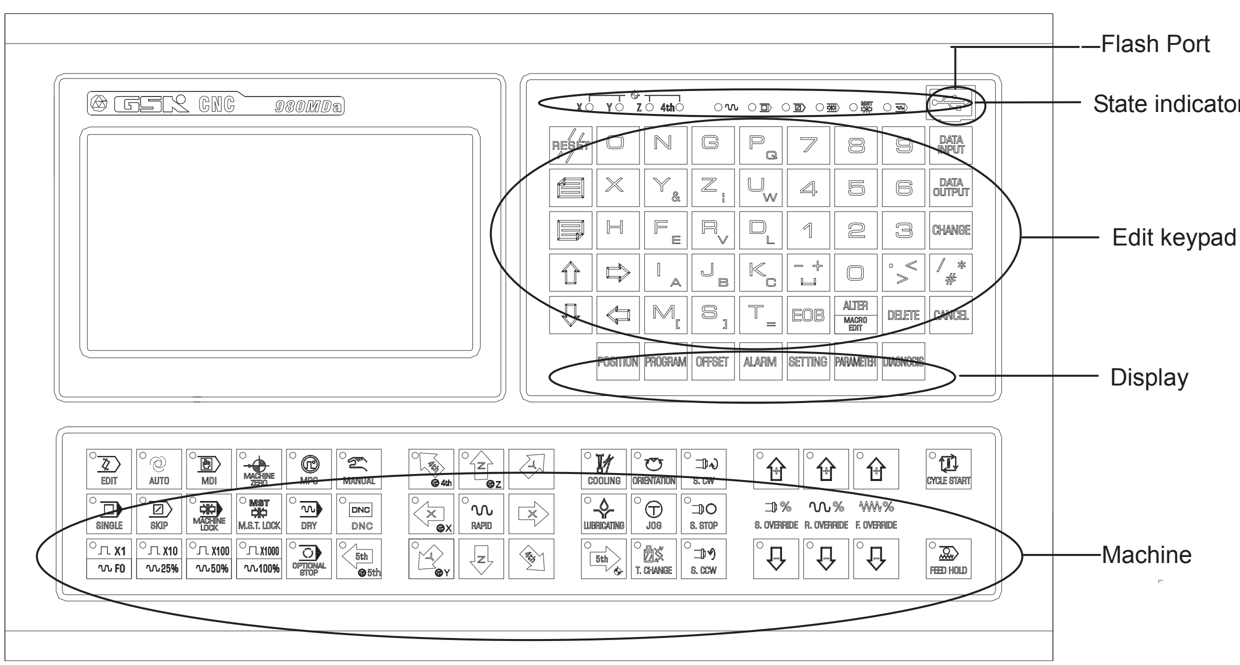
CHAPTER1 OPERATION MODE AND DISPLAY

This GSK980MDa system employs an aluminum alloy solid operator panel, which exterior is as follows.

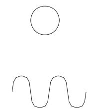
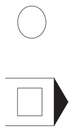
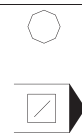
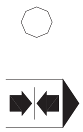
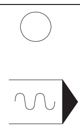


1.1 Panel Division

This GSK980MDa adopts an integrated panel, which division is as follows:

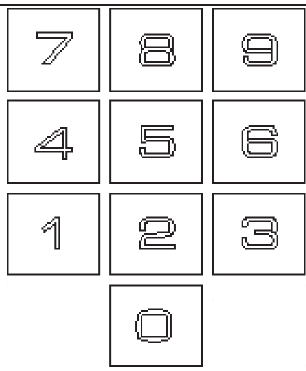
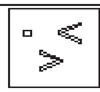
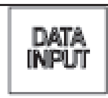
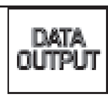
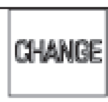
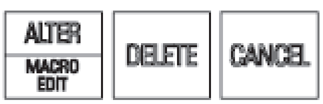
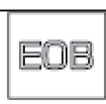
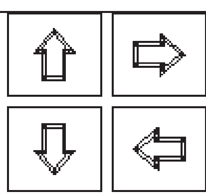
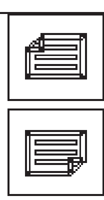


1.1.1 State indication

	machine zero return finish indicator		Rapid indicator
	Single block indicator		Block Skip indicator
	Machine Lock indicator		MST Lock indicator
	Dry Run indicator		

1.1.2 Edit keypad

Key	Name	Function
RESET	RESET key	For CNC reset, feed, output stop etc.
O N G P/Q X Y/& Z/! U/W H F/E R/V D/L	Address key	Address input
I/A J/B K/C M/[S/] T/=		Double address key, switching between two sides by pressing repeatedly
- + ␣ / * #	Sign key	Double address key, switching between two characters by pressing repeatedly

Key	Name	Function
	Numerical key	For digit input
	Decimal point	For decimal point input
	Input key	For confirmation of parameters, offset values input
	Output key	For start communication output
	Change key	For switching of message, display
	Edit key	For insertion, alteration, deletion of programs, words in editing(is a compound key, switching between two functions by pressing repeatedly)
	EOB key	For block end sign input
	Cursor moving keys	For cursor moving control
	Page key	Page switching in a same interface

1.1.3 Menu display

Menu key	Remark
	**To enter position interface. There are RELATIVE POS, ABSOLUTE POS, INTEGRATED POS, POS&PRG pages in this interface.**

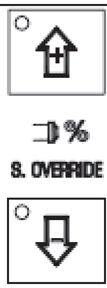
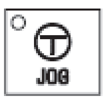
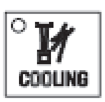
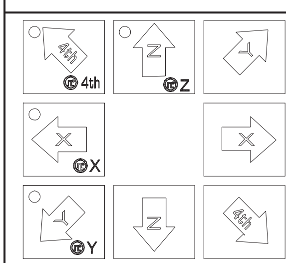
PROGRAM	**To enter program interface. There are PRG CONTENT, PRG STATE, PRG LIST, PRG PREVIEW,4 pages in this interface.**
OFFSET	**To enter TOOL OFFSET interface. There are TOOL OFFSET, MARRO variables and Tool Life Management (modifying Bit0 of state parameter №002). OFFSET interface displays offset values; MARRO for CNC macro variables.**
ALARM	**To enter alarm interface. There are CNC, PLC ALARM and ALARM Log pages in this interface.**
SETTING	**To enter Setting interface. There are SWITCH, PASSWORD SETTING, DATE &TIME, SETTING （G54～G59）, GRAGH SET and TRACK pages in this interface.**
PARAMETER	**To enter BIT PARAMETER, DATA PARAMETER, PITCH COMP interfaces (switching between each interface by pressing repeatedly).**
DIAGNOSIS	**To enter DIAGNOSIS interface.There are CNC DIAGNOSIS, PLC STATE, PLC VALUE, VERSION MESSAGE interfaces (switching between each interfaces by pressing the key repeatedly). CNC DIAGNOSIS, PLC STATE, PLC VALUE interfaces display CNC internal signal state, PLC addresses, data state message; the VERSION MESSAGE interface displays CNC software, hardware and PLC version No.**

1.1.4 Machine panel

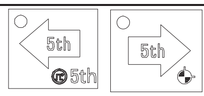
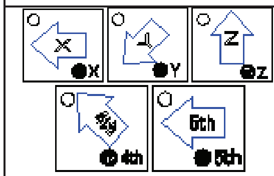
The keys function in GSK980MDa machine panel is defined by PLC program (ladder), see their function significance in the machine builder’s manual.

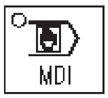
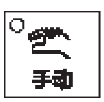
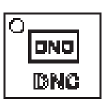
The functions of the machine panel keys defined by standard PLC program are as follows:

Key	Name	Function explanation	Function mode
FEED HOLD	Feed Hold key	Dwell commanded by program, MDI	Auto mode, DNC, MDI mode
CYCLE START	Cycle Start key	Cycle start commanded by program, MDI	Auto mode, DNC, MDI mode
F. OVERRIDE	Feedrate Override keys	For adjustment of the feedrate	Auto mode, DNC, MDI mode, Edit mode, Machine zero mode, MPG mode, Single Step mode, MANUAL mode

Key	Name	Function explanation	Function mode
	Rapid override keys	For adjustment of rapid traverse	Auto mode, DNC, MDI mode, Machine zero mode, MANUAL mode
	Spindle override keys	For spindle speed adjustment (spindle analog control valid)	Auto mode, DNC, MDI mode,edit mode, Machine zero mode, MPG mode, Step mode, MANUAL mode
	JOG key	For spindle Jog ON/OFF	Machine zero mode, MPG mode, Single Step mode, MANUAL mode,
	Lubricating key	For machine lubrication ON/OFF	Machine zero mode, MPGmode, Single Step mode,MANUAL mode,
	Cooling key	For coolant ON/OFF	Auto mode, MDI mode,Edit mode, Machine zero mode, MPG mode Step mode, MANUAL mode
	Spindle control keys	Spindle CCW Spindle stop Spindle CW	Machine zero mode, MPGmode, Single Step mode,MANUAL mode,
	Rapid traverse key	For rapid traverse /feedrate switching	Auto mode, DNC,MDI mode, Machine zero mode, MANUAL mode,
	Manual feed key	For positive/negative moving of X, Y, Z axis in Manual, Step mode	Machine zero mode, Step mode, MANUAL mode,

Volume II Operation

Key	Name	Function explanation	Function mode
			
	Handwheel axis selection key	For X, Y, Z axis selection in MPG mode	MPG mode
	MPG/Step increment and Rapid override selection key	Move amount per handwheel scale 0.001/0.01/0.1 mm Move amount per step 0.001/0.01/0.1 mm	Auto mode, MDI mode, Machine zero mode, MPG mode, Step mode,MANUAL mode,
	Single Block key	For switching of block/blocks execution, Single block lamp lights up if Single mode is valid	Auto mode, DNC, MDI mode
	Block Skip key	For skipping of block headed with"/"sign, if its switch is set for ON, the Block Skip indicator lights up	Auto mode, DNC, MDI mode
	Machine Lock key	If the machine is locked, its lamp lights up, and X, Z axis output is invalid.	Auto mode, DNC, MDI mode, Edit mode, Machine zero mode, MPG mode, Step mode, MANUAL mode,
	M.S.T. Lock key	If the miscellaneous function is locked, its lamp lights up and M, S, T function output is invalid.	Auto mode, DNC, MDI mode
	Dry Run key	If dry run is valid, the Dry run lamp lights up. Dry run for program/MDI blocks command	Auto mode, DNC, MDI mode

Key	Name	Function explanation	Function mode
	Edit mode key	To enter Edit mode	Auto mode, DNC, MDI mode, Machine zero mode, MPG mode, Step mode, MANUAL mode
	Auto mode key	To enter Auto mode	MDI mode, DNC, Edit mode, Machine zero mode, MPG mode, Step mode, MANUAL mode,
	MDI mode key	To enter MDI mode	Auto mode, DNC, Edit mode, Machine zero mode, MPG mode, Step mode, MANUAL mode,
	Machine zero mode key	To enter Machine zero mode	Auto mode, DNC, Edit mode, Machine zero mode, MPG mode, Step mode, MANUAL mode,
	Step/MPG mode key	To enter Step or MPG mode (one mode is selected by parameter)	Auto mode, DNC, Edit mode, Machine zero mode, MPG mode, Step mode, MANUAL mode,
	Manual mode key	To enter Manual mode	Auto mode, DNC, Edit mode, Machine zero mode, MPG mode, Step mode, MANUAL mode,===========
	DNC mode key	To enter DNC mode	To enter DNC mode by pressing this key in Auto mode

1.2 Summary of Operation Mode

There are 7 modes that include Edit, Auto, DNC， MDI, Machine zero, Step/MPG, Manual, modes in this GSK980MDa.

- **Edit mode**

  In this mode, the operation of part program setting-up, deletion and modification can be performed.

- **Auto mode**

  In this mode, the program is executed automatically.

- **MDI mode**

In this mode, the operation of parameter input, command blocks input and execution can be performed.

- **Machine zero mode**

In this mode, the operation of X, Y, Z, 4th, 5th axis machine zero return can be performed separately.

- **MPG / Step mode**

In the Step/MPG feed mode, the moving is performed by an increment selected by CNC system.

- **Manual mode**

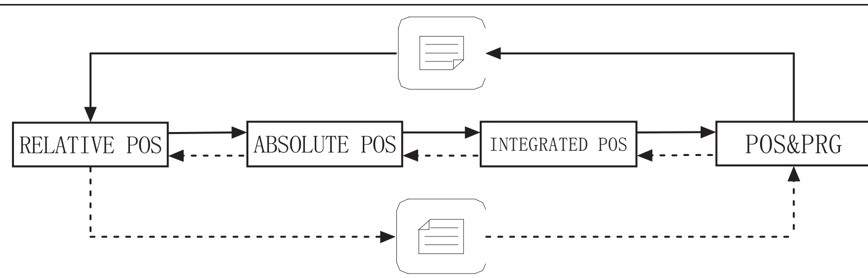
In this mode, the operation of Manual feed, Manual Rapid, feedrate override adjustment, Rapid override adjustment and spindle ON/OFF, cooling ON/OFF, Lubrication ON/OFF, spindle jog, manual tool change can be performed.

- **DNC mode**

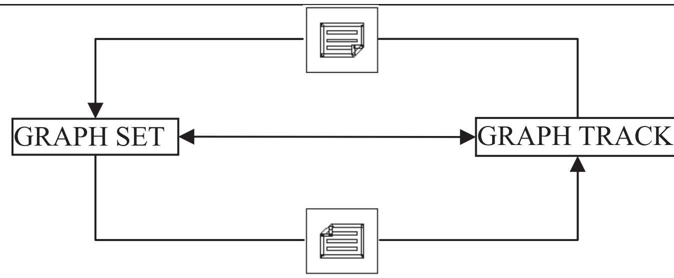
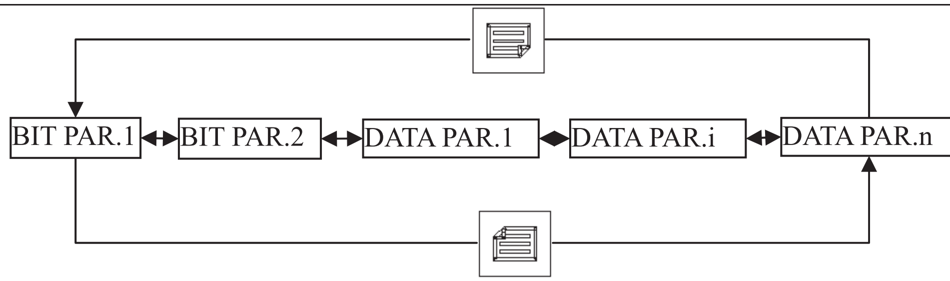
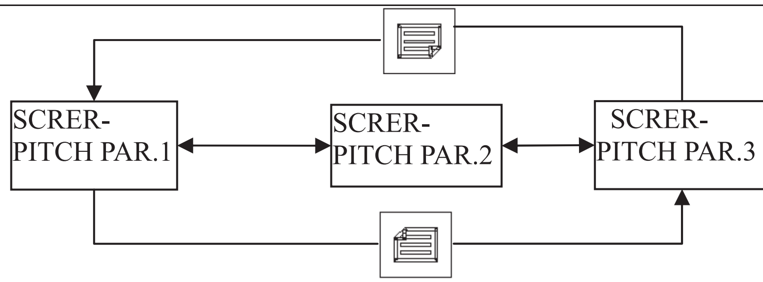
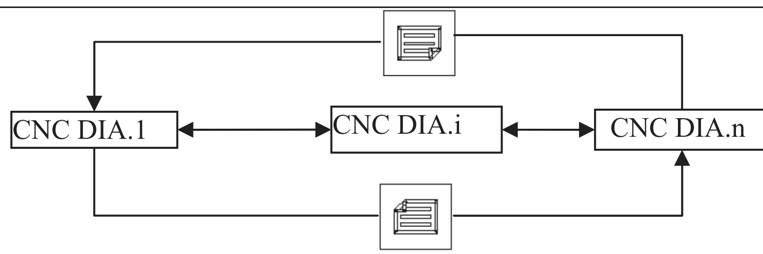
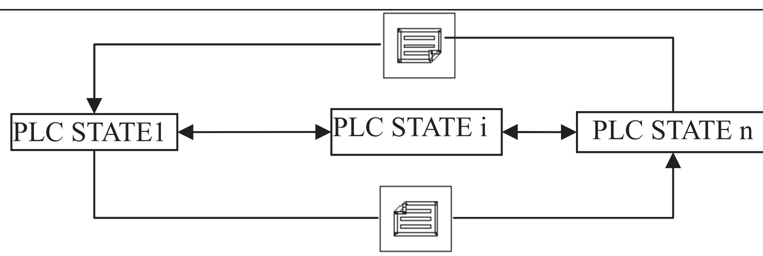
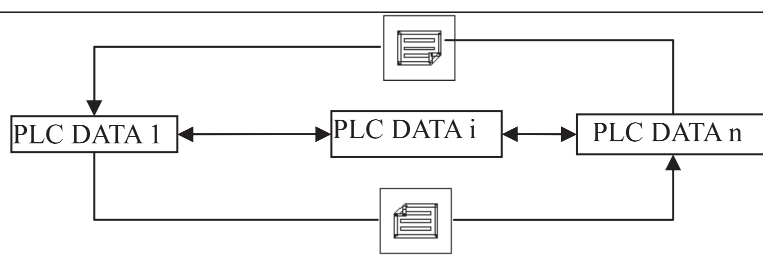
In this mode, the program is run by DNC mode.

1.3 Display Interface

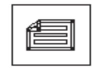
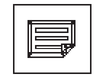
There are 7 interfaces for GSK980MDa such as Position, Program etc., and there are multiple pages in each interface. Each interface (page) is separated from the operation mode. See the following figures for the display menu, display interface and page layers:

Menu key	Display interface	Display page
POSITION	Position interface	
PROGRAM	Pro. content	PRG CONTENT
	Pro. state	PRG STATE
	Pro.preview	PRG PREVIEW
	Program list	PRG LIST

Menu key	Display interface	Display page
OFFSET	TOOL OFFSET interface	Tool Offset 1 ↔ Tool Offset i ↔ Tool Offset 5
	MACRO interface	MACRO 1 ↔ MACRO i ↔ MACRO 4
	Tool life interface	Tool Life 1 ↔ Tool Life i ↔ Tool Life n
ALARM	CNC alarm	CNC ALARM
	PLC alarm/warn	PLC ALARM/WARN
	Alarm log	ALARM LOG
SETTING	Setting interface	SWITCH SETTING ↔ Time &DATE ↔ AUTH.OPERATION
	G54 setting	SET (G54~G59)

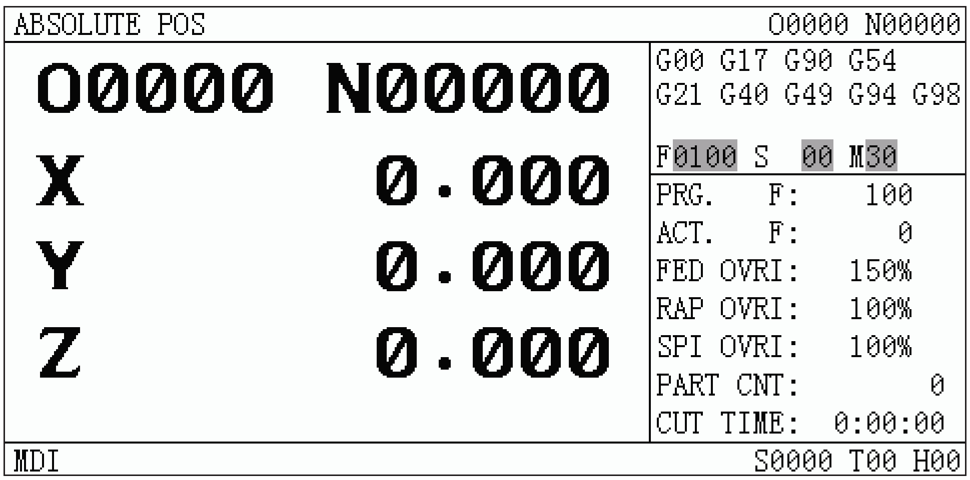
Menu key	Display interface	Display page
	Graph interface	
PARAMETER	Bit parameter Data parameter	
	Pitch parameter	
DIAGNOSIS	CNC diagnosis	
	PLC state	
	PLC data	
	Version message	VERSION MESSAGE

1.3.1 Position interface

Press  to enter Position interface, which has four interfaces such as ABSOLUTE POS, RELATIVE POS, INTEGRATED POS and POS&PRG, and they can be viewed by  or  key.

1) ABSOLUTE POS display interface

The X, Y, Z coordinates displayed are the absolute position of the tool in current workpiece coordinate system, as CNC power on, these coordinates are held on and the workpiece coordinate system is specified by G92.



PRG. F: a rate specified by F code in program

Note: It displays "PRG. F" in Auto, MDI mode; "MAN. F" in Machine zero, Manual mode;"HNDL INC"in MPG mode; "STEP INC"in Step mode.

ACT. F: Actual speed after feedrate override calculated.

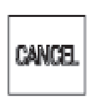
FED OVRI: An override that is selected by feedrate override switch.

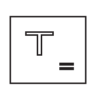
SPI OVRI: Adjust the spindle rotational speed by altering spindle override.

PART CNT: Part number plus 1 when M30 (or M99 in the main program) is executed

CUT TIME: Time counting starts if Auto run starts, time units are hour, minute and second

The parts counting and the cut time are memorized at power-down and the clearing ways for them are as follows:

PART CNT clearing: press  key then press  key.

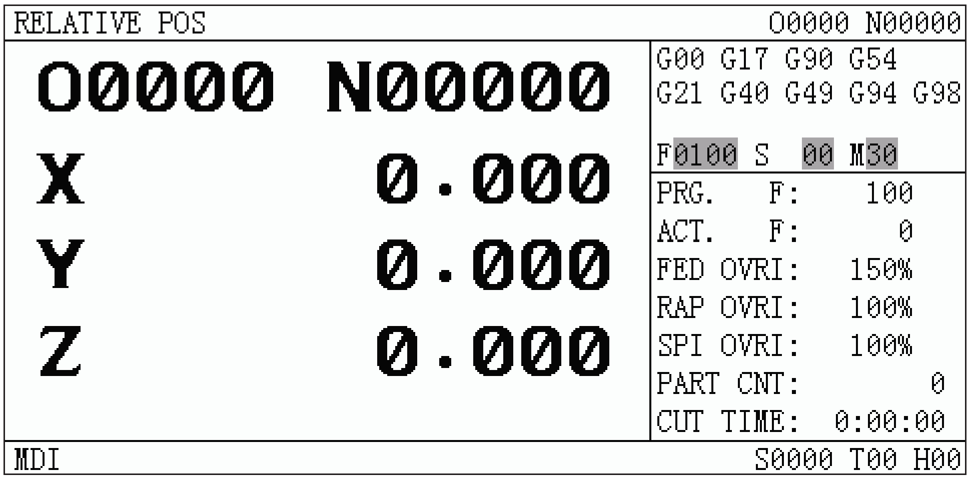
CUT TIME clearing: press  key then press  key.

S0000： Feedback spindle speed of spindle encoder, and spindle encoder must be fixed to display actual spindle speed.

T01： Current tool No. and tool offset No.

2) RELATIVE POS display page

The X, Y, Z axis coordinates displayed are the current position relative to the relative reference point, and they are held on at CNC power on. They can be cleared at any time. If X, Y, Z axis relative coordinates are cleared, the current position will be the relative reference point. When CNC parameter No.005 Bit1=1, as the absolute coordinates are set by G92 code, X, Y, Z axis relative coordinates are identical with the set absolute coordinates.



The clearing steps of X, Y, Z axis relative coordinates:

In RELATIVE POS page, press and hold X key till the "X"in the page blinks, press CANCEL key to clear X coordinate;

In RELATIVE POS page, press and hold Y key till the"Y"in the page blinks, press CANCEL key to clear Y coordinate;

In RELATIVE POS page, press and hold Z key till the "Z"in the page blinks, press CANCEL key to clear Z coordinate;

The method for X, Y, Z axis relative coordinates divided by 2:

In RELATIVE POS page, press and hold X key till the "X"in the page blinks, press / key, X coordinate will be divided by 2;

In RELATIVE POS page, press and hold Y key till the "Y"in the page blinks, press / key, Y coordinate will be divided by 2;

In RELATIVE POS page, press and hold Z key till the "Z"in the page blinks, press / key, Z coordinate will be divided by 2;

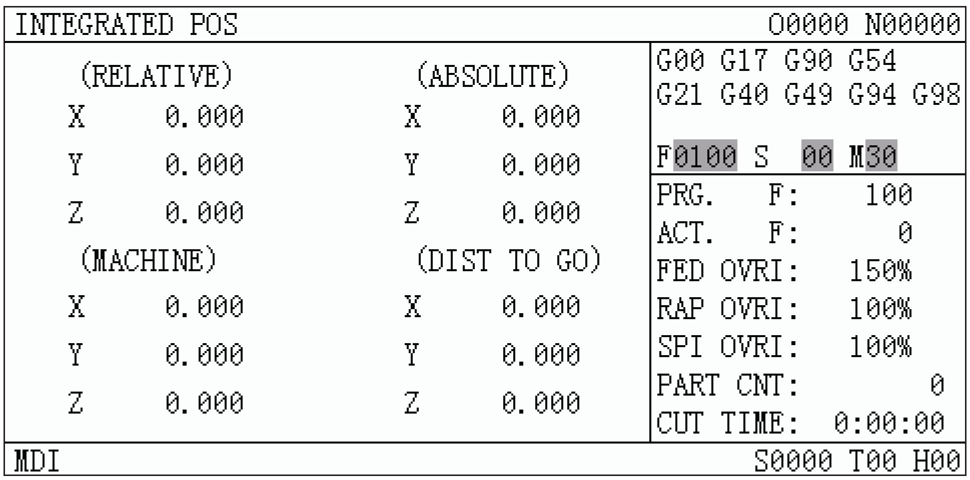
3) INTEGRATED POS display page

In INTEGRATED POS page, the RELATIVE, ABSOLUTE, MACHINE coordinate, DIST TO GO (only in Auto and MDI mode) are displayed together.

The displayed value of MACHINE coordinate is the current position in the machine coordinate system which is set up according to the machine zero.

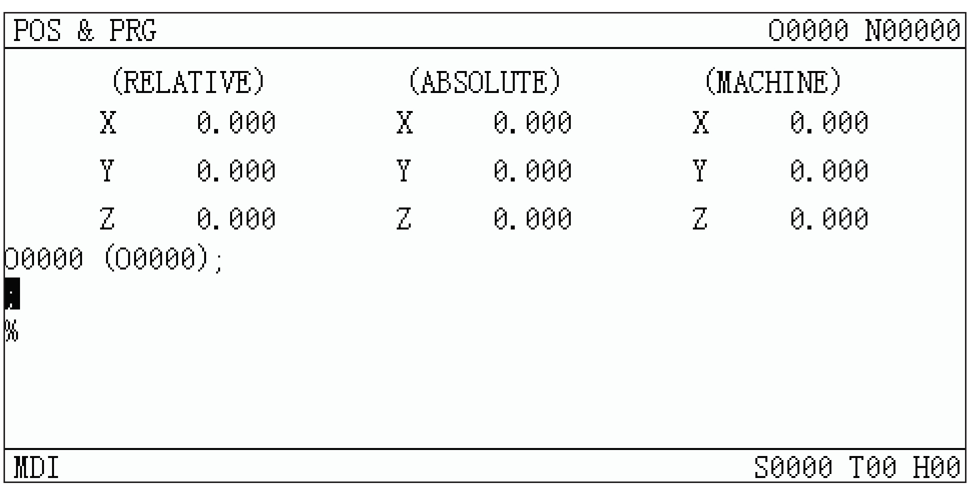
DIST TO GO is the difference between the target position of block or MDI and the current position.

The display page is as follows:



4) POS&PRG display page

In this page, it displays ABSOLUTE, RELATIVE of the current position (ABSOLUTE, DIST TO GO of current position will be displayed if BIT0 of bit parameter No.180 is set to 1) and 5 blocks of current program together. During the program execution, the blocks displayed are refreshed dynamically and the cursor is located in the block being executed.



1.3.2 Program interface

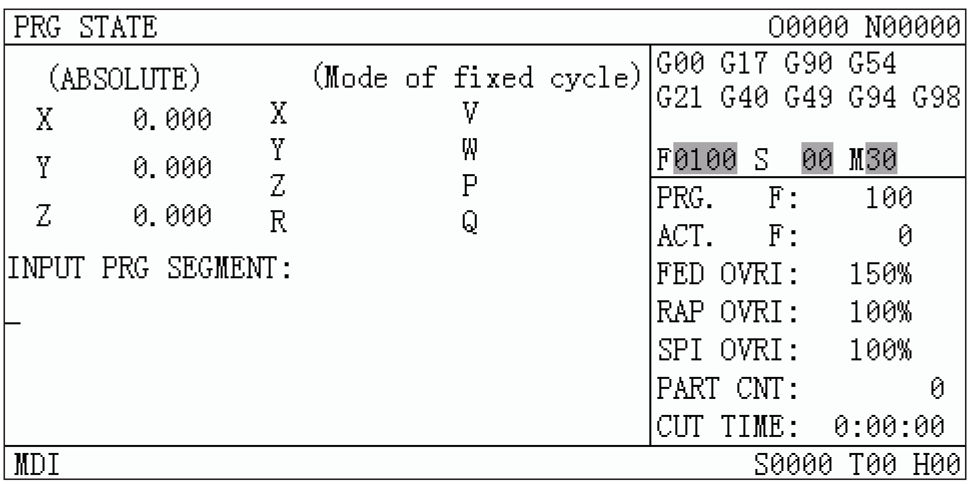
1) PROGRAM CONTENT page

is a compound key.Press


key once to enter the program content interface，and all blocks will be displayed by pressing and keys in MDI mode.

2) PROGRAM STATE page

Press key to enter program state interface in program content interface. Current G,M,S,T,F commands and related commands are displayed in program state interface and a single block （MDI）can be executed in this interface.



3) PROGRAM PREVIEW page

In program content interface，press key to enter program preview page. In this page, all part programs are listed. To make it easier for user to select a program, the system displays 5 blocks before the program with cursor at the bottom of the page. User can press EOB directly to select a program and process automatically, or press DEL key to delete the program in this page. It displays the following contents :

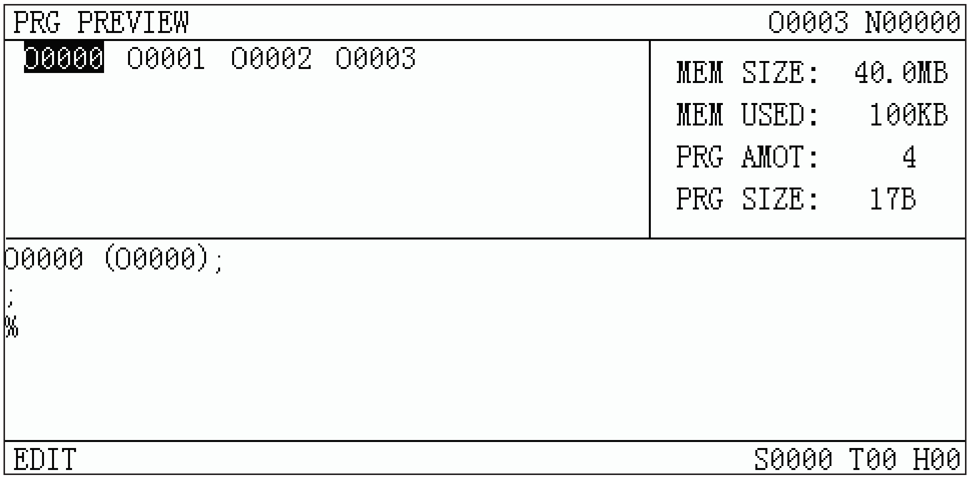
(a) Memory capacity: Display the maximum capacity of CNC memory unit.

(b) Used capacity：The space occupied by the saved programs

(c) Program NO.：Display the total number of programs in the CNC (including subprograms)

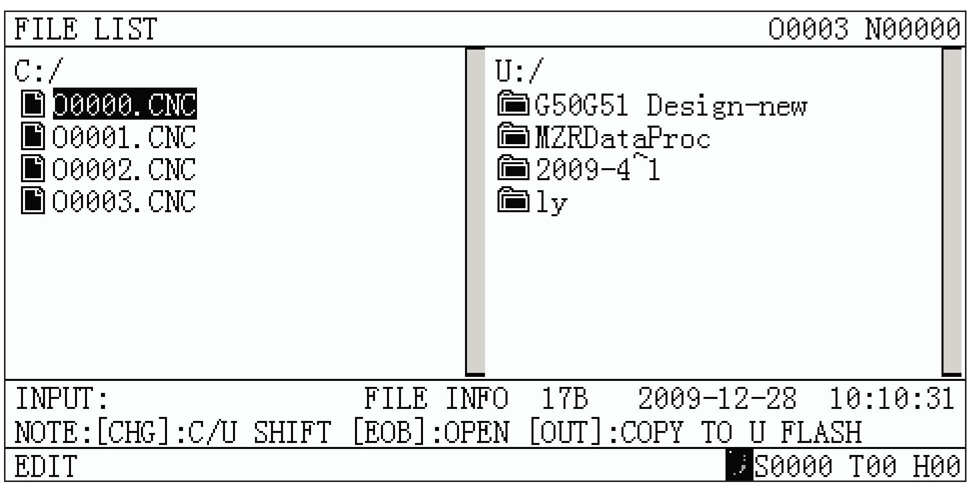
(d) Size of the program：The size of the program which the cursor is in, unit: byte (B)

(e) Program list：Display numbers of saved programs (arranged by name).

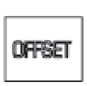


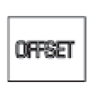
4) FILE LIST page

GSK980MDa supports USB interface, CNC→USB and USB→CNC mutual transmission operation are provided in this interface. In this page, it is easy to see the file list and file of CNC and USB (when USB is connected). At the same time, opening, duplication and deletion can be done here.



1.3.3 Tool offset, macro variable and tool life management interface

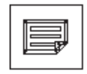


is a compound key, press


key once in other page to enter the TOOL OFFSET page, press

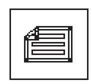
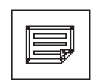

key again to enter the MACRO interface.

1． OFFSET interface

There are 4 tool offset pages in this interface, and 32 offset numbers（No.001～No.032）available for user, which can be shown as the following figure by pressing  or  keys.

TOOL OFFSET					O0003 N00000
NO.	Geo(H)	Wear(H)	Geo(D)	Wear(D)	(RELATIVE)
01	0.000	0.000	0.000	0.000	X 0.000
02	0.000	0.000	0.000	0.000	Y 0.000
03	0.000	0.000	0.000	0.000	Z 0.000
04	0.000	0.000	0.000	0.000	(ABSOLUTE)
05	0.000	0.000	0.000	0.000	X 0.000
06	0.000	0.000	0.000	0.000	Y 0.000
07	0.000	0.000	0.000	0.000	Z 0.000
08	0.000	0.000	0.000	0.000	
NO. 001					
EDIT					S0000 T00 H00

2． MACRRO interface

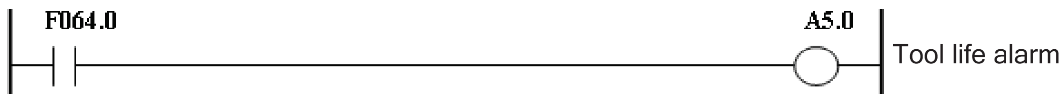
There are 25 pages in this interface, which can be shown by pressing  or  keys. In Macro page there are 600 （No.100～No.199 and No.500～No.999）macro variables which can be specified by macro command or set by keypad. Please refer to “macro, chapter 5, program” for related information.

MACRO					O0003 N00000
NO.	DATA	NO.	DATA	NO.	DATA
_100	Null	108	Null	116	Null
101	Null	109	Null	117	Null
102	Null	110	Null	118	Null
103	Null	111	Null	119	Null
104	Null	112	Null	120	Null
105	Null	113	Null	121	Null
106	Null	114	Null	122	Null
107	Null	115	Null	123	Null
NO. 100					
EDIT					S0000 T00 H00

3. Tool life management

Note: The tool change signal TLCH：F064#0 should be added for PLC when using this function.

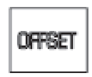
Ladder example:

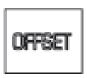


- **Using of tool life management function**

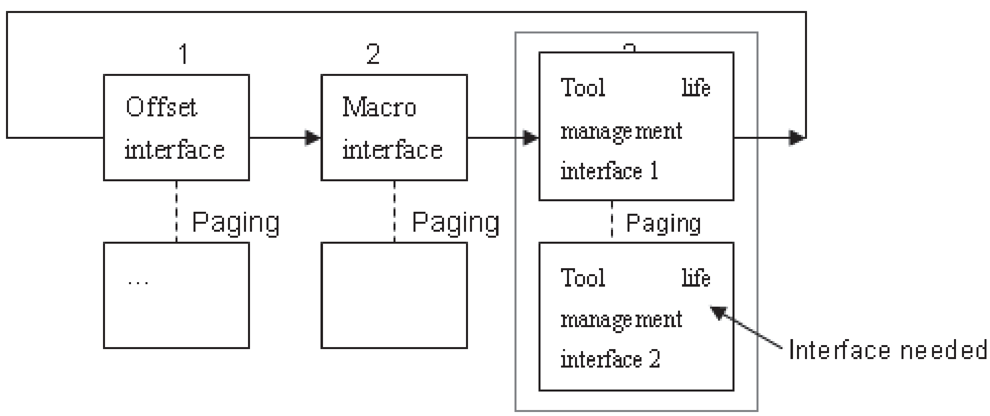
Parameter（No.002#0）is used as the symbol for tool life management function (0—unused，1—used); if the function is not used, the relevant tool life management page is not shown.

- **Tool life management display interface**

The tool life management is controlled by 

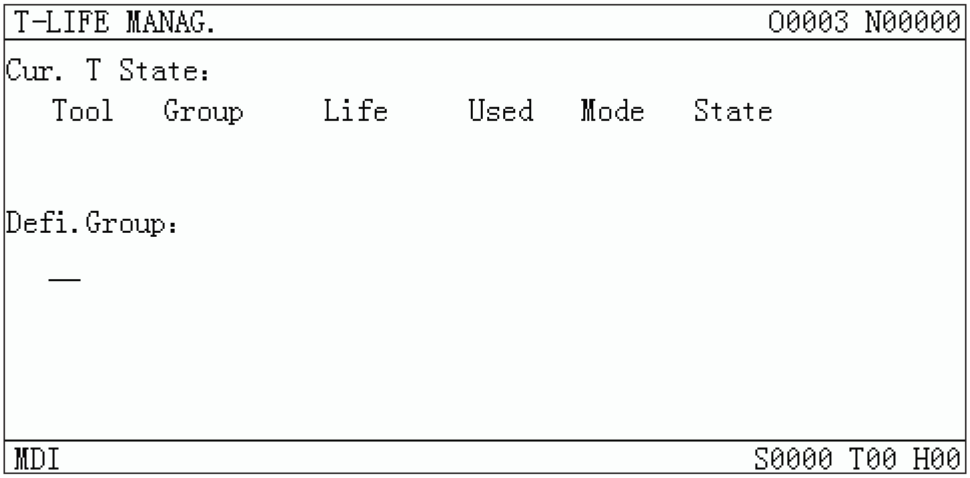
key, which is displayed in the third sub-interface, and it is composed by 2 pages (paging by page keys). Interface is shown by pressing 

key repeatedly



Tool life management display (the 1st page)

The 1st page for tool life management interface displays the life data of the current tool and the tool group list that has been defined. This page is mainly used for monitoring the tool life data by group units. The data monitoring of each tool in a group, group number setting and tool life management data are displayed in the following page.



ⅰ. Display explanation

<Current Tool State>: It displays the life data of the current tool which is being used.

Mode: It displays the counting unit of life data. (0: minute/1: times)

State: It displays the tool status. (0—Unused，1—Using，2—Over，3—Skip)

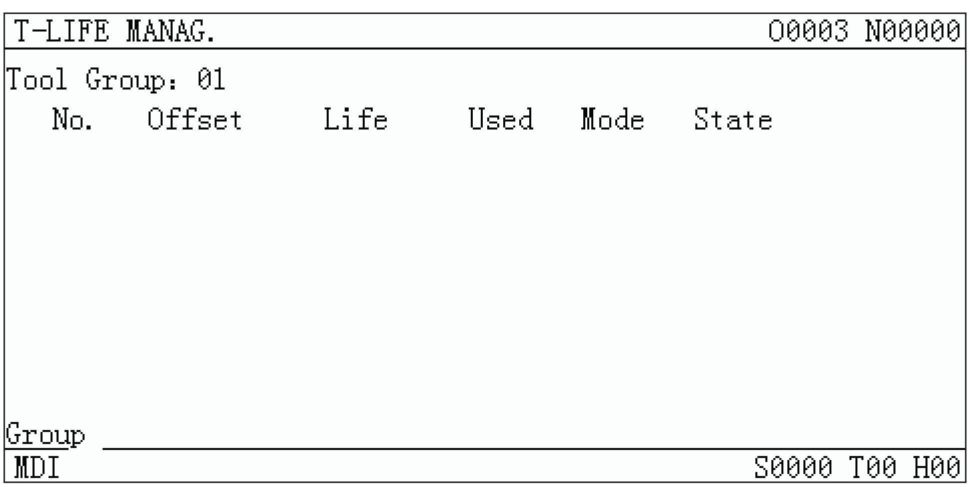
< Defined Group No. >： It only displays the group numbers which have been defined, and the undefined are not shown. The group number with the backlight means that all the tool life in that group has expired.

ⅱ. Deletion of all defined data

In this page, press CANCEL + G keys, it may delete all the data which have been defined (including group number, group tool numbers and life values, etc.)

Tool life management interface (the 2nd page)

The 2nd page is used to set and display the life data of a group which are displayed by order 1～8.



There are 3 display types for tool group selection:

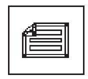
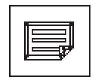
i. Directly input the group number in the “Tool Group P”of the 2nd page, it displays the tool life data. If the group does not exist, the number input will be taken as a new group number.The new group number: 05, and the 1st tool will be defined by system automatically:

ii. Move the cursor to select the group number in the "Defined Group No."of the 1st page, and it displays the group content as turning to the 2nd page.

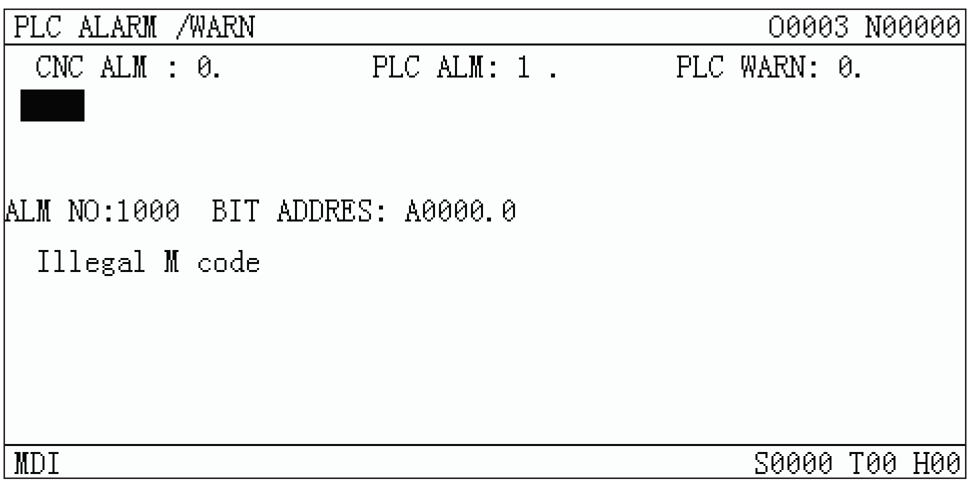
iii. As the current group number content is displayed in the 2nd page, it continues to display the following group number content by turning to the next page.

1.3.4 Alarm interface

Press 

key to enter Alarm interface, there are CNC ALARM, PLC ALARM, ALARM LOG pages in this interface, which can be viewed by  or  key.

1) PLC ALARM: It displays the numbers of CNC alarm, PLC alarm and the current PLC alarm No., as well as PLC warning and warning No.. It may display 24 PLC alarm or warning No. together. The details for the respective alarm No. can be viewed by moving the cursor. The page is as follows：



Page as the cursor locates at the alarm No.1000

2) CNC ALARM: It displays the numbers of CNC alarm, PLC alarm and the current CNC alarm No.. It can display 24 CNC alarm No. together. The details for the respective alarm No. can be viewed by moving the cursor. The page is as follows:

```
CNC ALARM                                          O0003 N00000
 CNC ALM : 2.          PLC ALM: 0 .        PLC WARN: 0.
[redacted]  431


CTR WARN:432
Y axis driver is not ready







MDI                 ALM                          S0000 T00 H00
```

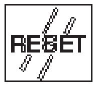
Page as the cursor locates at the alarm No.432

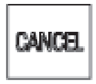
3) WARN LOG: Press ALARM key to enter Alarm interface, then press it again to enter the WARN LOG page, which records the latest alarm message including alarm date, alarm time, alarm No. and alarm content. 200 pieces warn log messages can be viewed by or key. See the following figure:

```
WARN LOG                        PAGE:1            O0003 N00000
2009/12/28  10:52:54   432#  1  O0003.CNC    N0000
Y axis driver is not ready
2009/12/28  10:52:54   431#  1  O0003.CNC    N0000
X axis driver is not ready
2009/12/28  10:52:38   431#  1  O0003.CNC    N0000
X axis driver is not ready






MDI                                              S0000 T00 H00
```

① **Sequence of warn log:** the latest alarm log message is shown on the forefront of the 1st page, and the others queue in sequence. If the alarm log messages areover 200, the last one will be cleared.

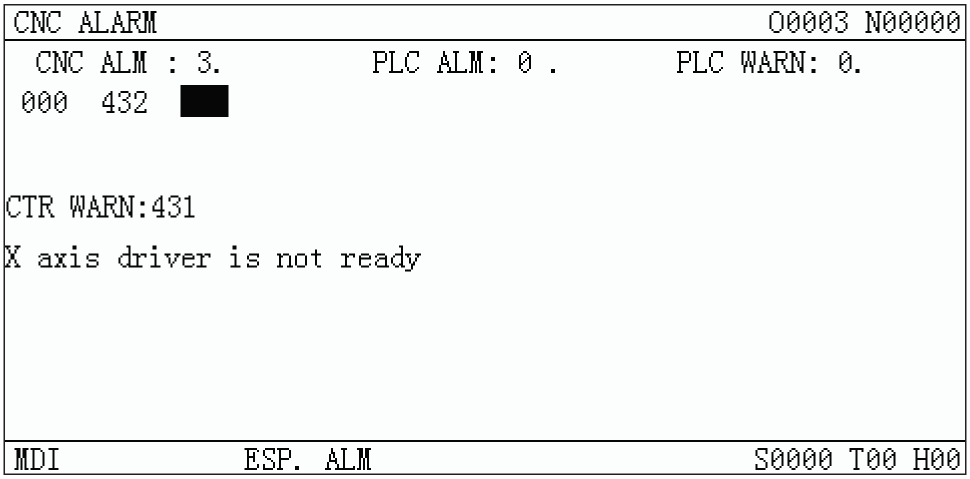
② **Manual clearing of warn log:** under the 2 level authority, press CANCEL + R V key, it may clear all the warn log messages.

4）Alarm clearing: If multiple alarms are issued, only one alarm where the cursor locates could be cleared by pressing RESET **key** each time (In alarm interface, it clears all alarms and warnings by pressing 

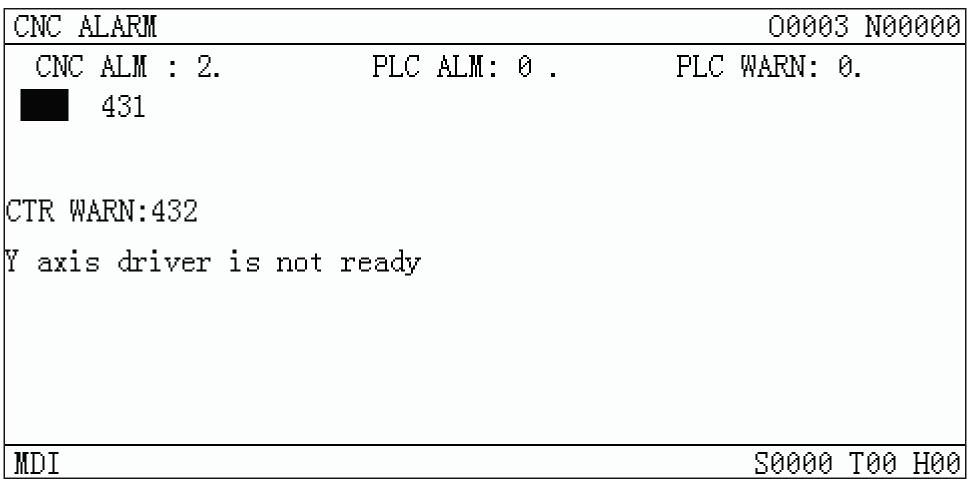
and 

keys).

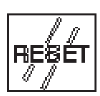
5) The current alarm page is as florrows:



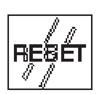
Current page

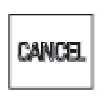


Page after pressing RESET key

6) Clearing PLC warning: If multiple warnings are issued, only one warning where the cursor locates could be cleared by pressing 

or 

key each time (In Alarm interface, it clears all alarms and warnings by pressing 

and 

keys).

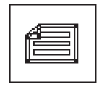
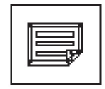
1.3.5 Setting interface

is a compound key, press 

key in other page, it enters setting interface, press it again, it enters the G54~G59 interface, press it three times, it enters Graphic interface. Press 

key repeatedly, it switches among the above nentioned interfaces.

1.Setting interface

There are 3 pages in this interface, which can be viewed by  and  keys.

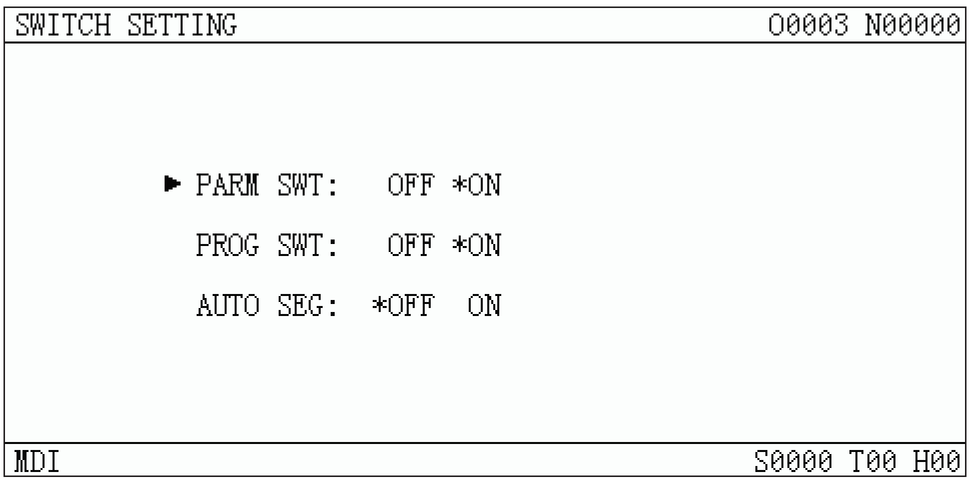
1）SWITCH SETTING: It is used for displaying the parameter, program, auto sequence No. on / off state.

PARM SWT: when it is turned ON, the parameters are allowed to be modified; it is turned OFF, the parameters are unallowed to be modified.

PROG SWT: when it is turned ON, the programs are allowed to be edited; it is turned OFF, the programs are unallowed to be edited.

AUTO SEG: when it is turned ON, the block No. is created automatically; it is turned OFF, the block No. is not created automatically, input manually if it is needed.

In this page, the state of on/off can be switched by 'left / right'key or 'U'and'D'key on the MDI panel.



2）Data backup: In this page, the CNC data (bit parameter, data parameter, pitch parameter, tool offset) can be saved and restored.

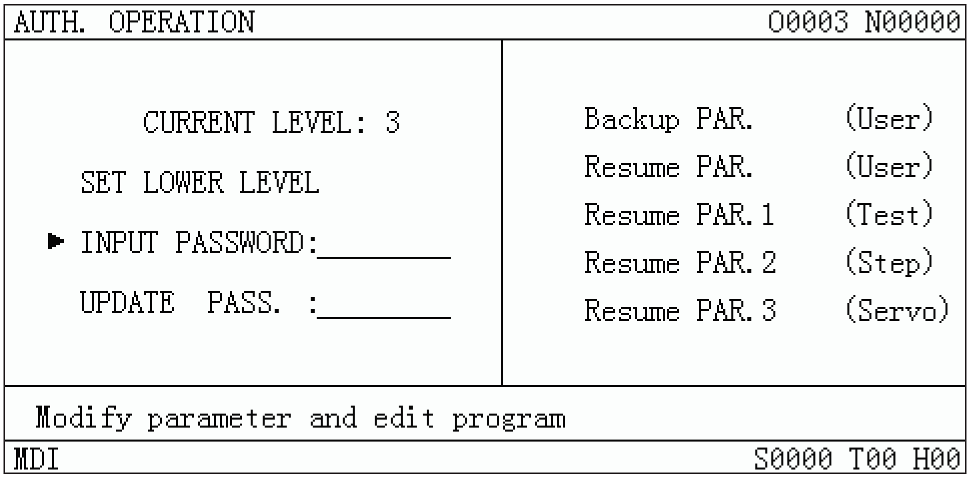
Data backup (user): For CNC data backup by user (save)

Recover backup data (user): For backup data recover by user (read)

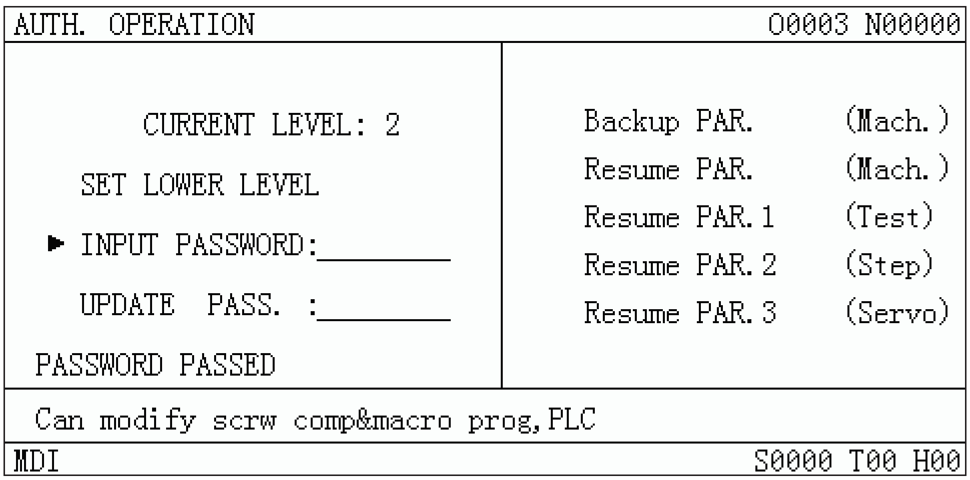
Recover standard parameter 1 (test): For reading original parameter data of CNC test by user

Recover standard parameter 2 (step): For reading original parameter data of suited step drive unit by user

Recover standard parameter 3 (servo): For reading original parameter data of suited servo drive unit by user.



User page of 3, 4, 5 level



User page of 2 level

3）Password setting：Display and set user operation level.

The password of GSK980MDa is composed of 4 levels, including machine builder (level 2), equipment management (level 3), technician (level 4) and machining operation (level 5).

Machine builder (level 2): It allows to modify CNC bit parameter, data parameter, screw- pitch parameter, tool offset parameter, edit part program (including macro program), edit and alter PLC ladder diagram, upload and download ladder diagram.

Equipment management (level 3): Initial password is 12345. The CNC bit parameter, data parameterm screw- pitch parameter, tool offset parameter, part program editing operations are allowed.

Technician (level 4): Initial password is 1234. Tool offset data (for tool setting), macro varibles, part program editing operations are allowed. However, CNC bit parameter, data parameter and pitch parameter editing operations are not allowed.

Machining operation (level 5): No password. Only the mschine panel operation is allowed. The alteration of tool offset data, CNC bit parameter, data parameter, pitch parameter, and the operations of part program selection, program editing are not allowed.

AUTH. OPERATION	O0003 N00000
CURRENT LEVEL: 3	Backup PAR. (User)
SET LOWER LEVEL	Resume PAR. (User)
▶ INPUT PASSWORD:________	Resume PAR.1 (Test)
UPDATE PASS. :________	Resume PAR.2 (Step)
	Resume PAR.3 (Servo)
Modify parameter and edit program	
MDI	S0000 T00 H00

1.Setting page of G54～G59 Page location

Press SETTING key twice, this page is displayed.

SET (G54～G59)		O0003 N00000
(EXT OFFSET)	(G54 COORDINATE)	(ABSOLUTE)
X 0.000	X 0.000	X 0.000
Y 0.000	Y 0.000	Y 0.000
Z 0.000	Z 0.000	Z 0.000
(G55 COORDINATE)	(G56 COORDINATE)	(MACHINE)
X 0.000	X 0.000	X 0.000
Y 0.000	Y 0.000	Y 0.000
Z 0.000	Z 0.000	Z 0.000
DATA		
MDI		S0000 T00 H00

SET (G54～G59)		O0003 N00000
(G57 COORDINATE)	(G58 COORDINATE)	(ABSOLUTE)
X 0.000	X 0.000	X 0.000
Y 0.000	Y 0.000	Y 0.000
Z 0.000	Z 0.000	Z 0.000
(G59 COORDINATE)	(COORDINATE OFFSET)	(MACHINE)
X 0.000	X 0.000	X 0.000
Y 0.000	Y 0.000	Y 0.000
Z 0.000	Z 0.000	Z 0.000
DATA		
MDI		S0000 T00 H00

The zero of the coordinate system: workpiece coordinate system zero offset, G54，G55，G56，G57，G58，G59.

- Moving of the cursor

  The cursor moves at the data of each coordinate system axis. And the data where the cursor locates are highlighted.

The cursor supports up and down, left and right moving, and the corresponding data are backlighted.

By pressing Page key, the 1st group X axis data on the corresponding interface where the cursor locates is backlighted.

- Absolute data input

After "data+ DATA INPUT key" is keyed in by user, the data where the cursor locates is changed to the "data" input by user.

The validity judgement of user input data is the same as that of 980TD coordinate data input in MDI mode.

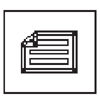
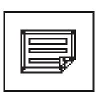
- Relative data input

After "data+ ALTER MACRO EDIT key" is keyed in by user, the original data where the cursor locates is changed by the sum of"data" newly input by user and original data.

- Auto measurement input

After " X (or Z ;, Y &)+ O + DATA INPUT key" is keyed in by user, the original data where the cursor locates is changed by the system current"X (or Z，Y) axis machine coordinate".

3. Graphic interface

There are GRAPH SET, GRAPH TRACK pages in this interface, which can be viewed by  and  keys.

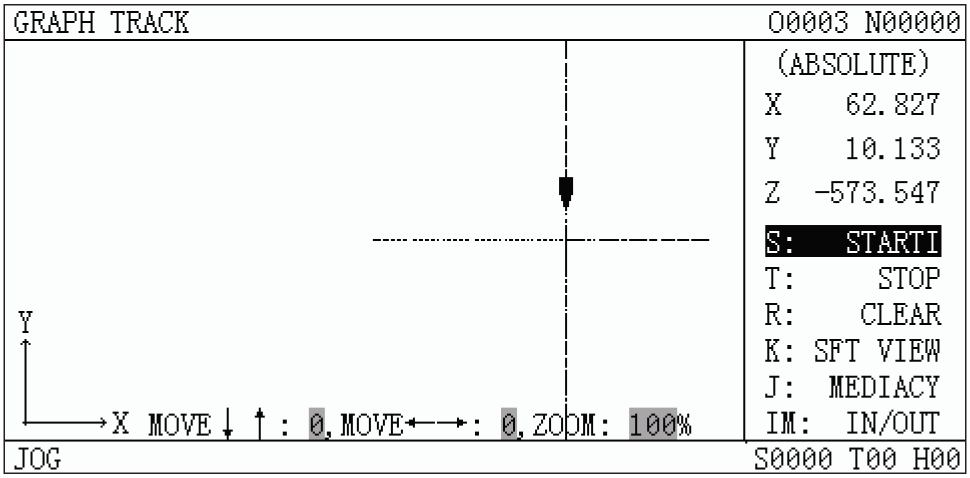
1）GRAPH SET page

In this page, the coordinate system, scaling and scope for graphic display can be selected.

```
GRAPH SET                                        O0003 N00000
►COOR OPT=        0 0XY 1YX 2ZX 3XYZ 4YZ 5ZY 6XZ 7XZY)
  SCALE   =      100%
  CENTER  =     0.000 (X axis value)
  CENTER  =     0.000 (Y axis value)
  CENTER  =     0.000 (Z axis value)
  X MAX.  =   120.000
  Y MAX.  =   120.000
  Z MAX.  =   120.000
  X MIN.  =  -120.000
  Y MIN.  =  -120.000
  Z MIN.  =  -120.000

MDI                                          S0000 T00 H00
```

2）GRAPH TRACK page

In this page, it displays the path within the parameters range (refer to absolute coordinate) of GRAPH SET page.

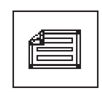
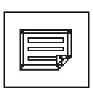


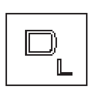
1.3.6 BIT PARAMETER, DATA PARAMETER, PITCH COMP interface

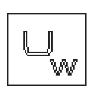
is a compound key, it enters BIT PARAMETER, DATA PARAMETER and PITCH COMP interfaces by pressing this key repeatedly.

1. BIT PARAMETER interface

Press 

key, it enters BIT PARAMETER interface, there are 48 bit parameters which are displayed by 2 pages in this interface, and they can be viewed or modified by pressing  or  key to enter the corresponding page. It is as follows:

As is shown in this page, there are 2 parameter rows at the bottom of the page, the 1st row shows the meaning of a bit of a parameter where the cursor locates, the bit to be displayed can be positioned by pressing 

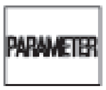
or 

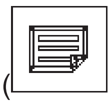
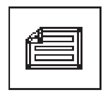
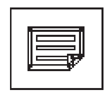
key. The 2nd row shows the abbreviation of all the bits of a parameter where the cursor locates.

BIT PARAMETER					O0003 N00000
NO.	DATA	NO.	DATA	NO.	DATA
001	00000000	_009	00000000	017	00101000
002	00000011	010	00011111	018	00000000
003	00000000	011	00000000	019	10000000
004	01000000	012	00010011	020	00000000
005	00010001	013	10000011	021	00000000
006	00000000	014	00011111	022	00000000
007	00000000	015	10000000	023	00000000
008	00011111	016	00000000	024	00000000

*** *** *** ALM5 ALM4 ALMZ ALMY ALMX
bit7:1/0:Unused
NO. 009
JOG S0000 T00 H00

2. DATA PARAMETER interface

Press 

key repeatedly ( key if in BIT PARAMETER interface), it enters DATA PARAMETER interface, there are 110 data parameters which are displayed by 7 pages in this interface, and they can be viewed or modified by pressing  or  key to enter the corresponding page. It is as follows:

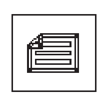
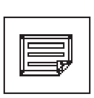
As is shown in this page, there is a cue line at the page bottom, it displays the meaning of the parameter where the cursor locates.

DATA PARAMETER					O0003 N00000
NO.	DATA	NO.	DATA	NO.	DATA
_049	1	057	1	065	100
050	1	058	1	066	100
051	1	059	7600	067	100
052	1	060	7600	068	100
053	1	061	7600	069	400
054	1	062	7600	070	8000
055	1	063	7600	071	50
056	1	064	100	072	100

Command multiplier for X axis.
NO. 049
JOG S0000 T00 H00

- **PITCH COMP interface**

Press 

key repeatedly, it enters PITCH COMP interface, there are 256 pitch parameters which are displayed by 16 pages in this interface, and they can be viewed by pressing  or  key.

SCREW-PITCH PARAMETER O0000 N00000

NO.	X	Y	Z	C	NO.	X	Y	Z	C
000	0	0	0	0	008	0	0	0	0
001	0	0	0	0	009	0	0	0	0
002	0	0	0	0	010	0	0	0	0
003	0	0	0	0	011	0	0	0	0
004	0	0	0	0	012	0	0	0	0
005	0	0	0	0	013	0	0	0	0
006	0	0	0	0	014	0	0	0	0
007	0	0	0	0	015	0	0	0	0

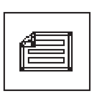
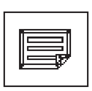
NO. = XYZC(0.001mm)
NO. 000
MDI S0000 T00 H00

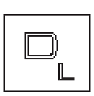
1.3.7 CNC DIAGNOSIS, PLC STATE, PLC VALUE, machine soft panel, VERSION MESSAGE interface

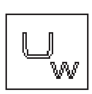
is a compound key, it enters CNC DIAGNOSIS, PLC STATE, PLC VALUE, machine soft panel, VERSION MESSAGE interfaces by pressing this key repeatedly.

1、CNC DIAGNOSIS interface CNC

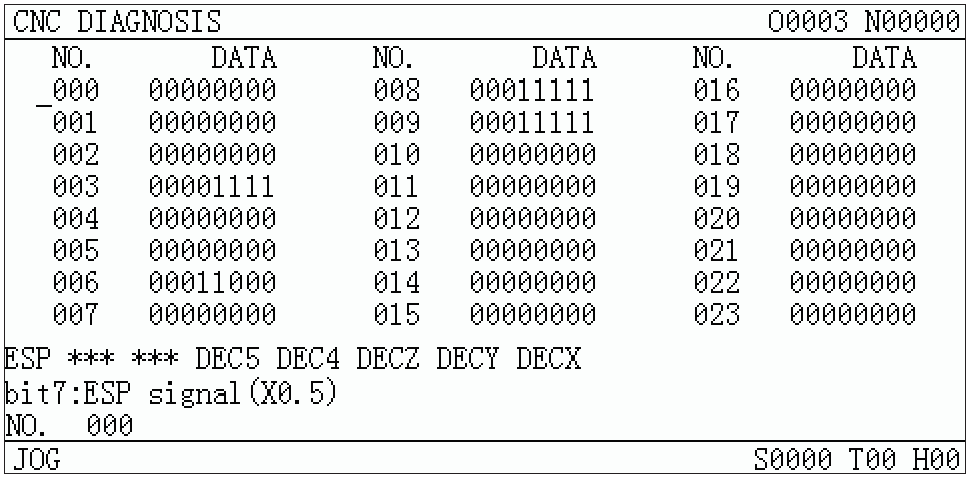
The input/output signal state between CNC and machine, the transmission signal state between CNC and PLC, PLC internal data and CNC internal state can all be displayed via diagnosis. Press 

key it enters CNC DIAGNOSIS interface, the keypad diagnosis, state diagnosis and miscellaneous function parameters etc. can be shown in this interface, which can be viewed by pressing  or  key.

In CNC DIAGNOSIS page, there are 2 diagnosis No. rows at the page bottom, the 1st row shows the meaning of a diagnosis No. bit where the cursor locates, the bit to be displayed can be positioned by pressing 

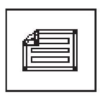
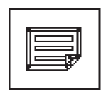
or 

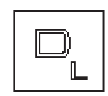
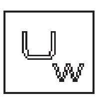
key. The 2nd row shows the abbreviation of all the diaosgnis No. bits where the cursor locates.

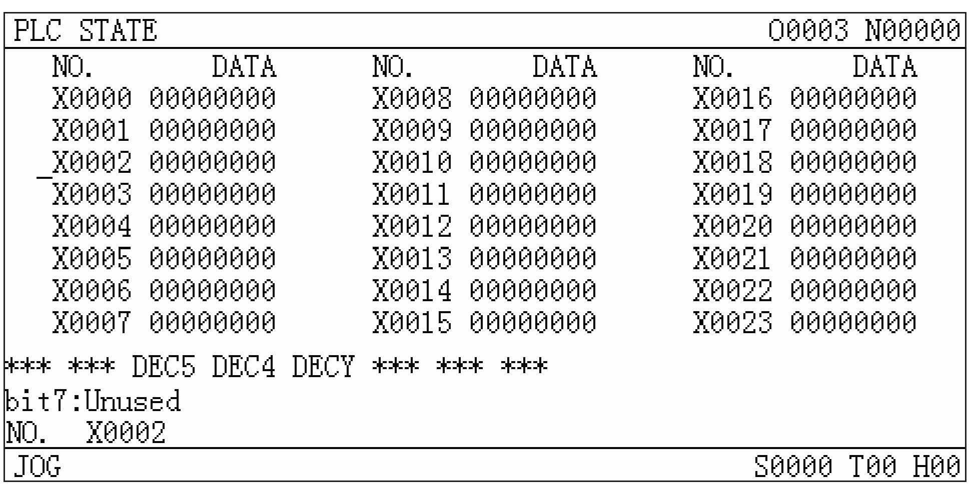


2. PLC STATE interface

In the page of this interface, it orderly displays the state of address X0000~X0029, Y0000~Y0019, F0000~F0255, G0000~G0255, A0000~A0024, K0000~K0039, R0000~R0999 etc..

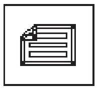
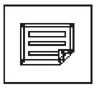
And it enters PLC STATE interface by pressing  key repeatedly. The signal state of PLC addresses can be viewed by pressing  or  key.

In PLC STATE page, there are 2 rows at the page bottom; the 1st row shows the meaning of a bit of an address where the cursor locates, the bit to be displayed can be positioned by pressing  or  key. The 2nd row shows the abbreviation of all the bits of an address where the cursor locates.



3. PLC VALUE interface

In the page of this interface, it orderly displays the values in the registers of T0000～T0099,D0000～D0999,C0000～C0099,DT000～DT099,DC000～DC099 etc.. By pressing 

key repeatedly it enters PLC VALUE interface. The data values of PLC can be viewed by pressing  or  key.

In this PLC VALUE page, there is a cue line at the page bottom, it displays the meaning of the parameter where the cursor locates. As is shown in the following figure:

```
PLC DATA                                              O0003 N00000
   NO.        DATA       NO.        DATA       NO.        DATA
  _DT000         0       DT008         0       DT016         0
   DT001         0       DT009         0       DT017         0
   DT002         0       DT010         0       DT018         0
   DT003         0       DT011         0       DT019       100
   DT004         0       DT012         0       DT020       500
   DT005         0       DT013         0       DT021       500
   DT006         0       DT014         0       DT022       100
   DT007         0       DT015         0       DT023       500

Reserved
NO.  DT000
JOG                                                 S0000 T00 H00
```

4. VERSION MESSAGE interface

It enters VERSION MESSAGE interface by pressing 

key repeatedly. The software, hardware, and PLC version message can be shown in this interface. The figure is as follows:

```
VERSION MESSAGE                                       O0000 N00000

            PRODUCT TYPE : GSK980MDa
            SOFTWARE VER.: V2.00-manu
            HARDWARE VER.: 3.01.002--08.07.21
            SYSTEM     ID: 0

            LADDER DESIGN: GSK
            LADDER VER.  : 09.01.15-839C
            LADDER VERIFY: 839C
            LADDER NOTE  : GSK980MDa

MDI                                                 S0000 T00 H00
```

1.4 List of general operations

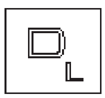
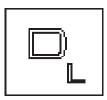
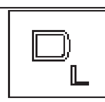
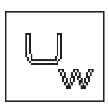
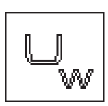
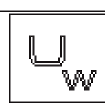
Item	Function	Operation key	Operation mode	Display page	Password level	Program on/off	Parameter switch
Clearing	Relative coordinate of X axis clearing	X. CANCEL		Relative coordinate			
	Relative coordinate of Y axis clearing	Y&. CANCEL		Relative coordinate			
	Relative coordinate of z axis clearing	Z. CANCEL		Relative coordinate			
	Part No. clearing	CANCEL + N		Relative coordinate or absolute coordinate			
	Cutting time clearing	CANCEL + T=					
	Tool radius offset D clearing	0. DATA INPUT		Tool offset	Level 2,3,4		
	Tool length offset H clearing	0. DATA INPUT		Tool offset	Level 2,3,4		
Data input	Bit parameter	Parameter. DATA INPUT	MDI mode	Bit parameter	Level 2,3		On
	Data parameter	Parameter. DATA INPUT	MDI mode	Bit parameter	Level 2,3		On

Item	Function	Operation key	Operation mode	Display page	Password level	Program on/off	Parameter switch
	Input pitch parameter of X axis	X. Compensation value. DATA INPUT	MDI mode	Pitch parameter	Level 2		On
	Input pitch parameter of Y axis	Y &. Compensation value. DATA INPUT	MDI mode	Pitch parameter	Level 2		On
	Input pitch parameter of Z axis	Z |. Compensation value. DATA INPUT	MDI mode	Pitch compensation parameter	Level 2		On
	Macro varibles	Macro varibles. DATA INPUT		Macro varibles	Level 2,3,4		
	Input tool radius offst D	Data value. DATA INPUT		Tool offst	Level 2,3,4		
	Input tool length offset H	Data value. DATA INPUT		Tool offset	Level 2,3,4		
Search	Search down from where the cursor locates	Character. ⇩	Edit mode	Program content	Level 2,3,4	On	
	Search up from where the cursor locates	Character. ⇧	Edit mode	Program content	Level 2,3,4	On	
	Search down from current program	O. ⇩	Edit mode or auto mode	Program content, program list or program state	Level 2,3,4		
	Search up from current program	O. ⇧			Level 2,3,4		

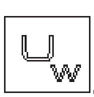
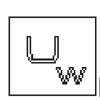
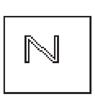
Item	Function	Operation key	Operation mode	Display page	Password level	Program on/off	Parameter switch
	Search defined program	O. program name. ⇩			2 级. 3 级. 4 级 Level 2,3,4		
	Search for bit parameter, data parameter or pitch parameter	P Q. Parameter no.. DATA INPUT		Corresponding page of the data			
	PLC state, PLC data searching	P Q. address No. . DATA INPUT		PLC state, PLC data			
Deletion	Delete the character where the cursor is in	DELETE	Edit mode	Program content	Level 2,3,4	On	
		CANCEL	Edit mode	Program content	Level 2,3,4	On	
	Single block deletion	Move the cursor to the head of the line. DELETE	Edit mode	Program content	Level 2,3,4	On	
	Multi-block deletion	CHANGE. N. order number. DELETE	Edit mode	Program content	Level 2,3,4	On	
	Segment deletion	CHANGE. character. DELETE	Edit mode	Program content	Level 2,3,4	On	

Item	Function	Operation key	Operation mode	Display page	Password level	Program on/off	Parameter switch
	Delete one program	. program name. DELETE	Edit mode	Program content	Level 2,3,4	On	
	Delete all programs	. 999. DELETE	Edit mode	Program content	Level 2,3,4	On	
Change name	Change program name	. program name. ALTER MACRO EDIT	Edit mode	Program content	Level 2,3,4	On	
Duplication	Duplicate program	. program name. CHANGE	Edit mode	Program content	Level 2,3,4	On	
CNC →CNC （send）	Tool offset	DATA OUTPUT	Edit mode	Tool offset	Level 2,3		On
	Bit parameter	DATA OUTPUT	Edit mode	Bit parameter	Level 2,3		On
	Data parameter	DATA OUTPUT	Edit mode	Data parametr	Level 2,3		On
	Pitch parameter	DATA OUTPUT	Edit mode	Pitch parameter	Level 2		On
	Send a part program	, program name, DATA OUTPUT	Edit mode	Program content	Level 2,3,4	On	

Item	Function	Operation key	Operation mode	Display page	Password level	Program on/off	Parameter switch
	Send all part programs	. − + 999. DATA OUTPUT	Edit mode	Program content	Level 2,3,4	On	
CNC →CNC （receive）	Tool offset		Edit mode		Level 2,3,4		On
	Bit parameter		Edit mode		Level 2,3		On
	Data parameter		Edit mode		Level 2,3		On
	Pitch parameter		Edit mode		Level 2		On
	Part program		Edit mode		Level 2,3,4	On	
CNC →PC (upload)	Tool offset	DATA OUTPUT	Edit mode	Tool offset	Level 2,3,4		On
	Bit parameter	DATA OUTPUT	Edit mode	State parameter	Level 2,3,4		On
	Bit parameter	DATA OUTPUT	Edit mode	Data parameter	Level 2,3		On
	Pitch parameter	DATA OUTPUT	Edit mode	Pitch compensation parameter	Level 2		On
	Send a program	, program name, DATA OUTPUT	Edit mode	Program content	Level 2,3,4	On	
	Send all programs	. − + 999. DATA OUTPUT	Edit mode		Level 2,3,4	On	

Item	Function	Operation key	Operation mode	Display page	Password level	Program on/off	Parameter switch
PC→CNC (download)	Tool offset		Edit mode		Level 2,3,4		On
	Bit parameter		Edit mode		Level 2,3		On
	Data parameter		Edit mode		Level 2,3		On
	Pitch parameter		Edit mode		Level 2		On
	Part program		Edit mode		Level 2,3,4	On	
Switch setting	Turn on parameter switch			Switch setting	Level 2,3		
	Turn on program switch			Switch setting	Level 2,3,4		
	Turn on auto sequence No.			Switch setting			
	Turn off parameter switch			Switch setting	Level 2,3		
	Turn off program switch			Switch setting	Level 2,3,4		
	Turn off auto sequence No.			Switch setting			

Explanations: “. ” in the column “operation” indicates operate two keys successively, “+” indicates operate two keys simultaneously.

Example: **.** **CANCEL** **indicates that press** **key first，and then press** **CANCEL** **key;** **CANCEL** **+**  **indicates that press two keys simultaneously.**

CHAPTER 2 POWER ON OR OFF AND PROTECTION

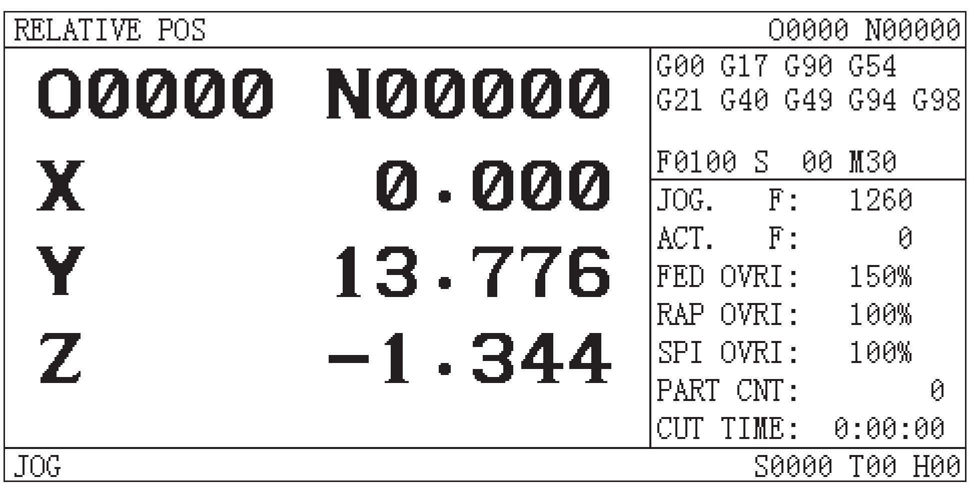
2.1 System Power On

Before this GSK980MDa is powered on, the following should be confirmed:

1. The machine is in a normal state.
2. The power voltage conforms to the requirement of the machine.
3. The connection is correct and secure.

The following page is displayed after GSK980MDa is powered on:

The current position (RELATIVE POS) page is displayed after system auto detection and initiation are finished.



2.2 System Power Off

Before power is off, ensure that:

1. The axes of the CNC are at halt;
2. Miscellaneous functions (spindle, pump etc.) are off;
3. Cut off CNC power prior to machine power cutting off.

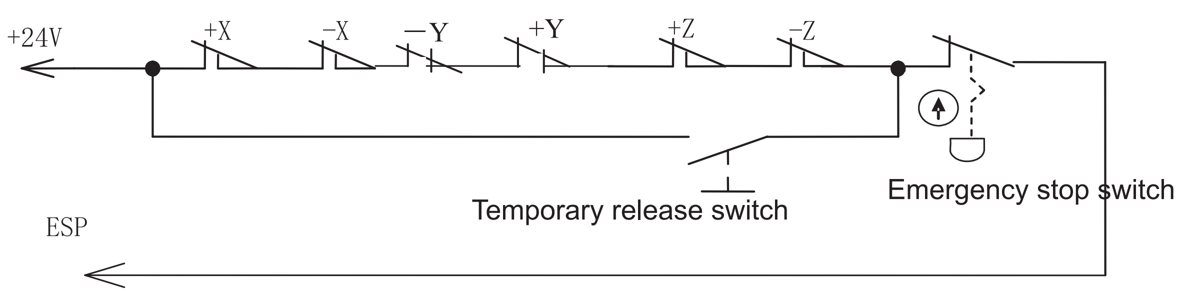
Note: Please see the machine builder's manual for the machine power cut-off operation.

2.3 Overtravel Protection

Overtravel protection should be employed to prevent the damage to the machine due to the overtravel of the axes.

2.3.1 Hardware overtravel protection

The stroke switches are fixed at the positive and negative maximum travel of the machine axes X, Y, Z, 4th, 5th respectively，they are connected by the following figure. And the “MESP”of bit parameter No.017 must be setted to 0. If the overtravel occurs, the stroke switch acts to make the machine stop, and the emergency alarm issues.



When the hardware overtravel occurs, there will be an “emergency stop”alarm. The steps to eliminate this alarm is press the OVERTRAVEL button to reversely move the table to detach the stroke switch (for positive overtravel, move negatively; vice versa).

2.3.2 Software overtravel protection

When the “MOT” of bit parameter No.17 is set to 0, the software limit is valid.

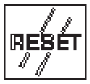
The software travel stroke is set by data parameter NO.135~ NO.144, they refer to machine coordinate. No.135~No.139 are for axes (X, Y, Z, 4th, 5th) positive max.overtravel, №140～№144 are for negative max.overtravel.

If the machine position (coordinate) exceeds the setting range, overtravel alarm will occur. The steps to eliminate this alarm is press RESET key to clear the alarm, then moves reversely (for positive overtravel, move out negatively; vice versa)

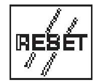
2.4 Emergency Operation

During the machining, some unexpected incidents may occur because of the user programming, operation and product fault.So this GSK980MDa should stopped immediately for these incidents. This section mainly describes the resolutions that this GSK980MDa are capable of under the emergency situation. Please see the relative explanation for these resolutions under the emergency by machine builder.

2.4.1 Reset

Press 

key to reset this GSK980MDa system if there are abnormal outputs and axis actions in it:

1. All axes movement stops;
2. M, S function output is invalid (PLC ladder defines whether automatically cut off signals such as spindle CCW/CW, lubrication, cooling by pressing 

key);
3. Auto run ends, modal function and state held on.

2.4.2 Emergency stop

During machine running, if the emergency button is pressed under the dangerous or emergent situation, the CNC system enters into emergency status and the machine movement is stopped immediately. If the emergency button is released, the emergency alarm is cancelled and the CNC resets. Its circuit wiring is shown in section 2.2.1 of this chapter.

Note 1 Ensure the fault is eliminated before the emergency alarm is cancelled.

Note 2 pressing down the Emergency button prior to power on or off may alleviate the electric shock to the machine system.

Note 3 Reperform the machine zero return operation to ensure the correct position coordinate after the emergency alarm is cancelled (machine zero return operation is unallowed if there is no machine zero on the machine.).

Note 4 Only the MESP of the bit parameter No.017 is set to 0, is the external emergency stop valid.

2.4.3 Feed hold

Key can be pressed during the machine running to make the running pause. However, in thread cutting, cycle running, this function can not stop the running immediately.

2.4.4 Power off

Under the dangerous or emergency situations during the machine running, the machine power should be cut off immediately to avoid the accidents. However, it should be noted that there may be a big error between the CNC displayed coordinate and the actual position. So the tool setting operation should be performed again.

CHAPTER 3 MANUAL OPERATION

Press 

key, it enters Manual mode. In this mode, the manual feed, spindle control, override adjustment operations can be performed.

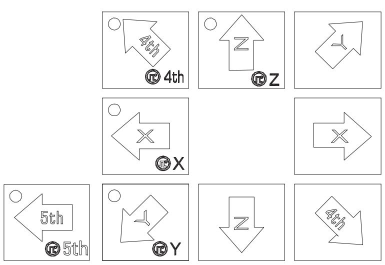
> **Note!**
>
> The keys functions of this 980MDa machine panel are defined by Ladder Diagram; please refer to the respective materials by the machine builder for the function significance.
>
> Please note that the following function introduction is described based on the 980MDa standard PLC programs!

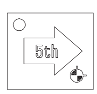
3.1 Coordinate axis moving

In Manual mode, the coordinate axis can be moved manually for feeding and rapid traverse.

3.1.1 Manual feed

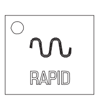
Press feed axis and axis direction key in the direction selection area



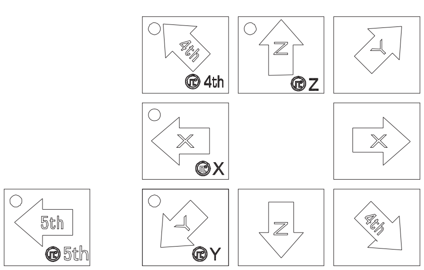
, 

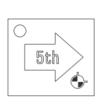
the corresponding axis may be moved positively or negatively, and the axis stops moving if releasing these two keys; and the direction selection keys of X. Y. Z. 4th. 5th axes can be hold on at a time to make the 5 axes to move simultaneously.

3.1.2 Manual rapid traverse

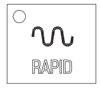
First press 

key in the feed axis and direction selection area



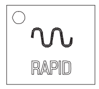


till the rapid traverse indicator in the State area lights up. The corresponding axis can be rapidly moved positively or negatively by pressing direction selection key, and the axis stops moving if releasing the key; and the direction selection keys of X. Y. Z. 4th. 5th axes can be hold on at a time to make the 5 axes to move simultaneously.

In Manual rapid mode, press 

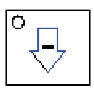
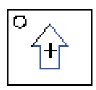
key to make the indicator go out, and the rapid traverse is invalid, it enters the Manual feed mode.

Note 1: Before machine zero return, the validity of manual rapid traverse is set by the "ISOT" of the bit parameter No.012.

Note 2: In Edit or MPG mode, 

key is invalid.

3.1.3 Manual feedrate override adjustment

In Manual mode, the  or  key in 

can be pressed to modify the Manual feedrate override, and the override has 16 levels. The relation of the feedrate override and the feedrate is as the following table:

Feedrate override (%)	Feedrate (mm/min)
0	0
10	2.0
20	3.2
30	5.0
40	7.9
50	12.6
60	20
70	32
80	50
90	79
100	126
110	200
120	320
130	500
140	790
150	1260

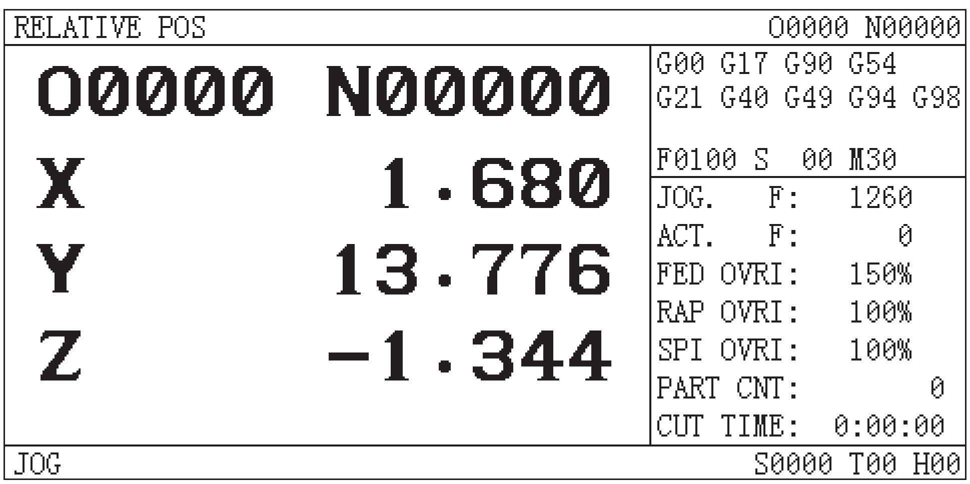
Note: There is about 2% fluctuating error for the data in the table.

3.1.4 Manual rapid override adjustment

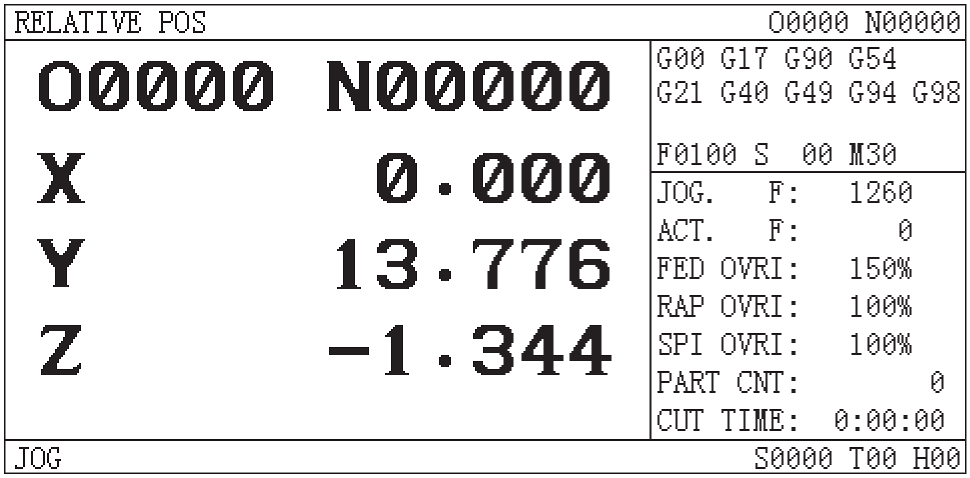
In the manual rapid traverse, or key in R. OVERRIDE can be pressed (also by X1 F0, X10 25%, X100 50%, X1000 100% key with the respective override F0, 25%,50%，100%)to modify the Manual rapid override, and there are 4 gears of F0, 25%, 50%，100% for the override.(F0 is set by data parameter No.069)

3.1.5 Relative coordinate clearing

1）Press POSITION key to enter Position interface, then press or key to select the RELATIVE POS page;



2）Press X key to make the “X”in the page to blink,then press CANCEL key;

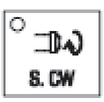


3）The clearing operations of other coordinates are the same as above.

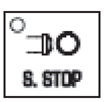
3.2 Other Manual operations

Note: The following operations are also valid in Machine zero, MPG/Step mode.

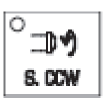
3.2.1 Spindle CCW, CW, stop control



: In Manual mode, the spindle rotates conterclockwise if pressing this key;;

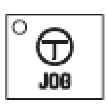


: In Manual mode, the spindle stops if pressing this key;



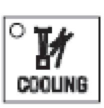
: In Manual mode, the spindle rotates clockwise if pressing this key;

3.2.2 Spindle Jog

Press and hold 

key, the spindle rotates conterclockwise, release it, the spindle stops.

3.2.3 Cooling control



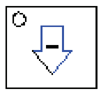
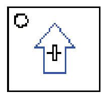
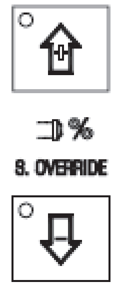
: In Manual mode, press this key, the coolant is switched on/off.。

3.2.4 Lubrication control

See details in Appendix for its function.

3.2.5 Spindle override adjustment

In Manual mode, if the spindle speed is controlled by analog voltage output, the spindle speed may be adjusted.

By pressing the  or  key in Spindle Override keys , the spindle speed can be changed by real-time adjusting of the spindle override that has 8 levels of 50％～120％.

CHAPTER 4 MPG/STEP OPERATION

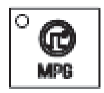
In MPG/Step mode, the machine moves by a specified increment.

Note！

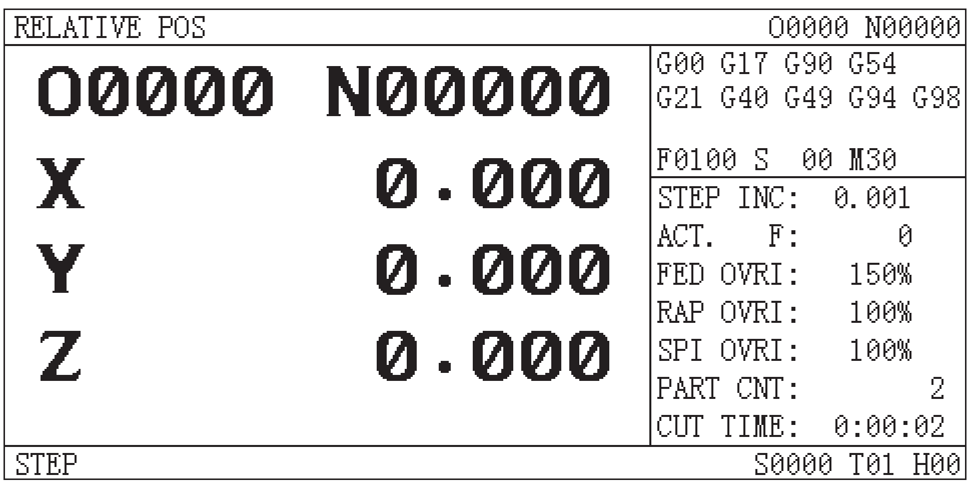
The keys functions of this 980MDa machine panel are defined by Ladder; please refer to the respective materials by the machine builder for the function significance.

Please note that the following function introduction is described based on the 980MDa standard PLC programs!

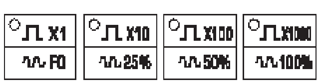
4.1 Step Feed

Set the BIT3 of the bit parameter No.001 to 0, and press 

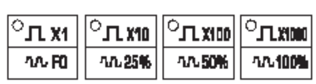
key to enter the Step mode, it displays as follows:



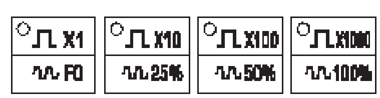
4.1.1 Increment selection

Press 

key to select the move increment, the increment will be shown in the page..

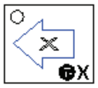
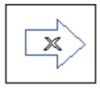
Note: In the EDIT or REF modes, 

keys are invalid. In the AUTO or MDI modes, rapid override will be changed by pressing the above-mentioned keys. In the MANUAL mode, press rapid move key 

and 

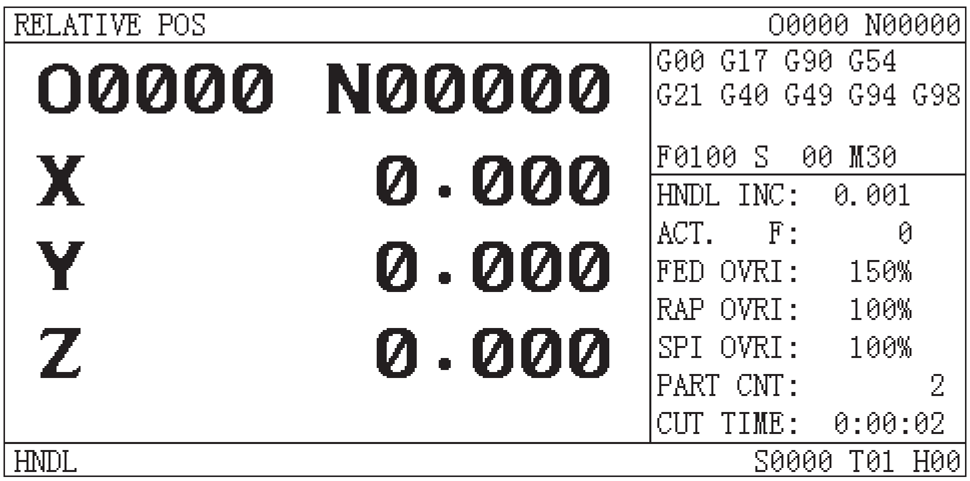
keys together, these keys are valid, otherwise, they are invalid.

4.1.2 Moving direction selection

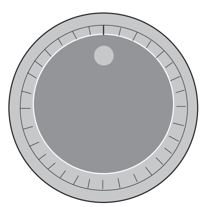
Press  or  key once, X axis can be moved negatively or positively by a step increment, other axises are the same.

4.2 MPG (Handwheel) Feed

Set the BIT3 of the bit parameter No.001 to 1, and press  key to enter the MPG mode, it displays as following:

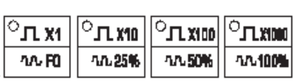


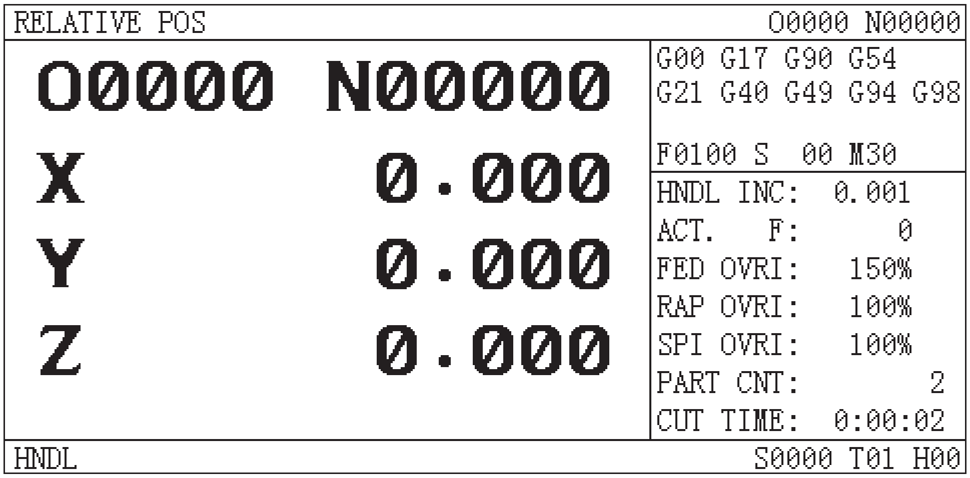
The handwheel figure is as follows：



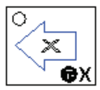
The handwheel figure

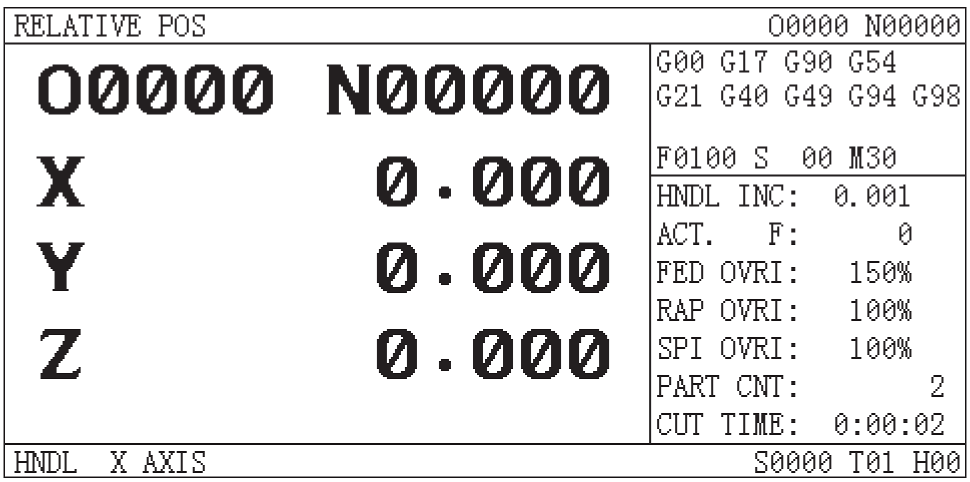
4.2.1 Increment selection

Press  key to select the move increment, the increment will be shown in the page：



4.2.2 Moving axis and direction selection

In MPG mode, press  key to select the corresponding axis. The page is as follows (Other axises are the same):



The handwheel feed direction is defined by its rotation direction. Generally, the handwheel CW is for positive feed, and CCW is for negative feed. In case of that handwheel CW is for negative feed, CCW for positive feed, it may exchange the A, B signals of the handwheel terminals,also you can modify the HNGX. HNGY. HNGZ. HNG4. HNG5 of the bit parameter №019.

4.2.3 Explanation items

1. The correspondence between the handwheel scale and the machine moving amount is as following table：

	Moving amount of each handwheel scale			
Handwheel increment	0.001	0.0100	0.100	1.000
Specified coordinate value	0.001mm	0.010mm	0.100mm	1.000mm

2. The rotation speed of the handwheel should be less than 5 r/s, if it is over that, the scale may be not coincide with the moving amount

3. The handwheel axis selection key is valid only in the MPG mode.

CHAPTER 5 MDI OPERATION

In MDI mode, the operations of parameter setting, words input and execution can be performed.

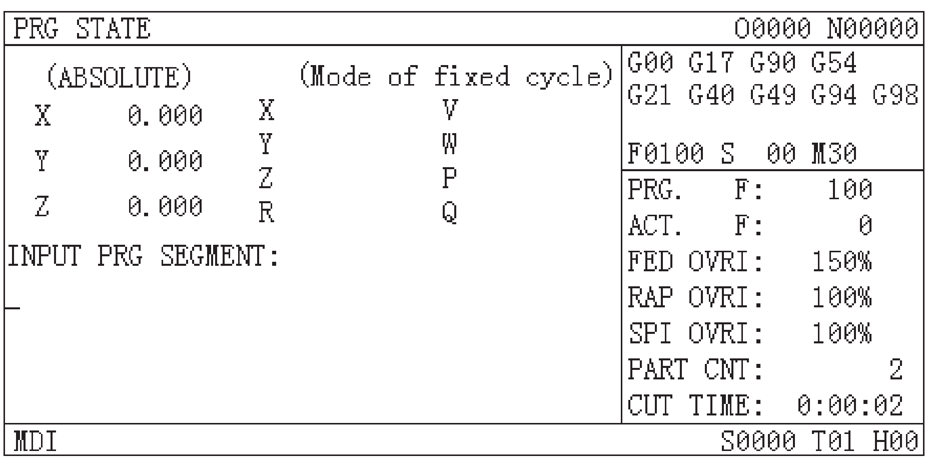
> **Note!**
>
> The keys functions of this 980MDa machine panel are defined by Ladder; please refer to the respective materials by the machine builder for the function significance.
>
> Please note that the following function introduction is described based on the 980MDa standard PLC programs!

5.1 Code Words Input

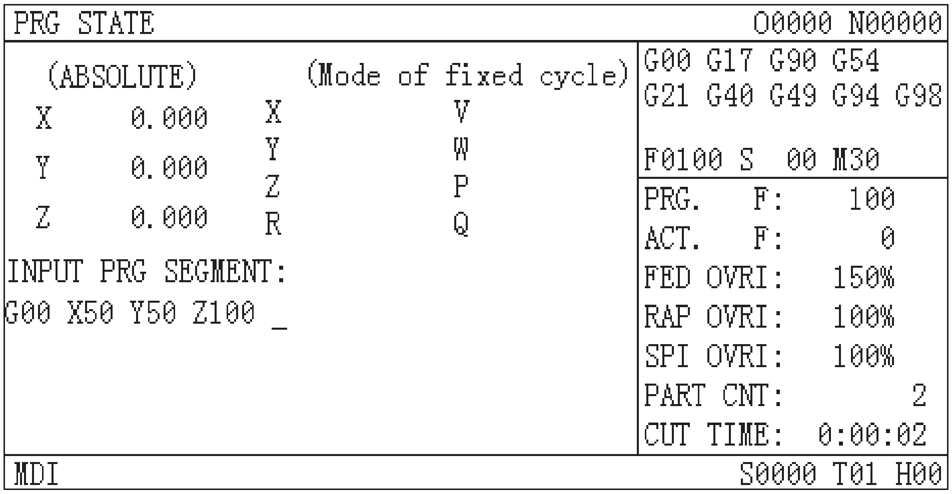
Select MDI mode to enter the PRG STATE page, to input an block “G00 X50 Z100”，the steps are as follows:

1. Press MDI key to enter MDI mode;

2. Press PROGRAM key to enter PRG STATE page:

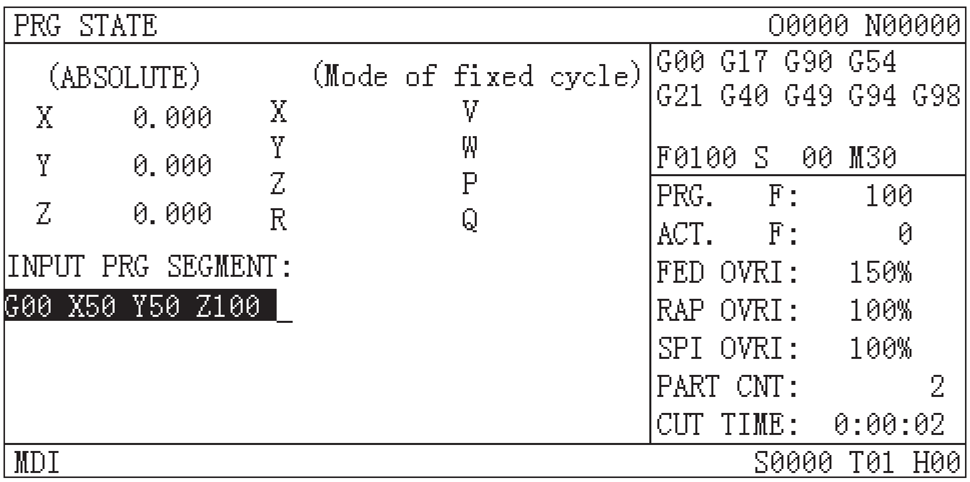


3. Input G. 0. 0, X. 5. 0, Y&. 5. 0, Z;. 1. 0. 0 by sequence, the page is as follows:

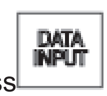


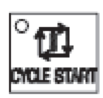
4. Press 

, the page is as follows:

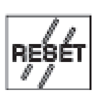


5.2 Code Words Execution

After the words are input, and press 

, the background color of program segment becomes white, these MDI words are executed after the 

key is pressed. During the execution,Press 

, 

and Emergency Stop button may be pressed to terminate the MDI words execution.If 

key is pressed,the background color of program segment will becomes black,then words can be input again.

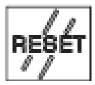
Note: The subprogram call command (M98 P ; etc.) is invalid in MDI mode.

5.3 Parameter Setting

In MDI mode, the parameter value can be modified after entering the parameter interface. See details in Chapter 9 of this part.

5.4 Data Modification

In the PRG STATE page, before the inputted words will be executed, if there is an error in inputted words, press 

to cancel highligt state, then program segment can be modified. It may press 

key to clear all the words, then input the correct words; for example ,"Z1000" will be inputted to replace Z100 in Section 5.1 of this chapter, the steps are as follow.

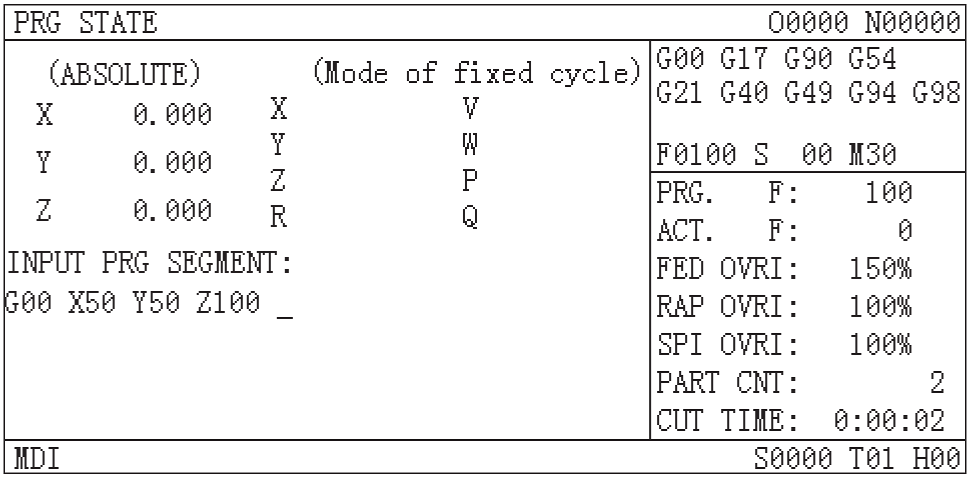
1. press 

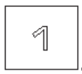
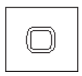
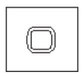
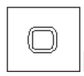
key，the page is as follows：

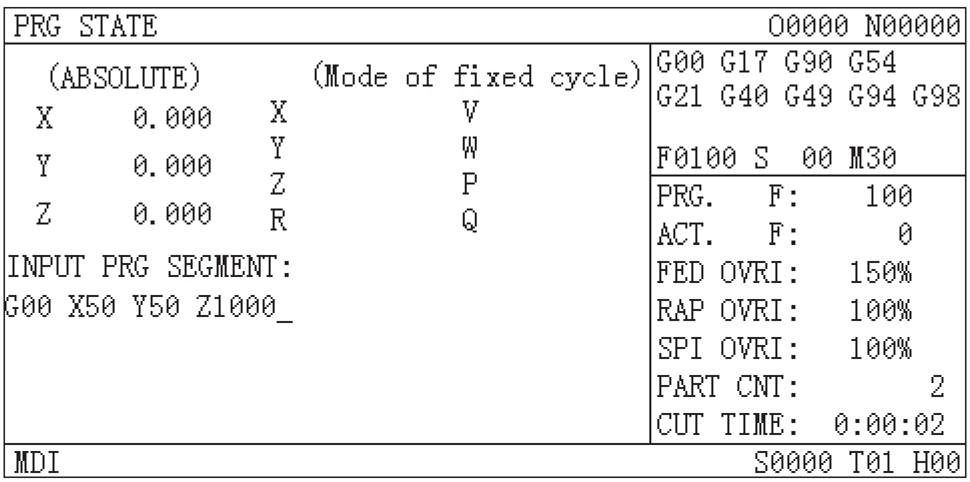
```
PRG STATE                                              O0000 N00000
  (ABSOLUTE)          (Mode of fixed cycle) | G00 G17 G90 G54
   X      0.000    X          V              | G21 G40 G49 G94 G98
   Y      0.000    Y          W              |
   Z      0.000    Z          P              | F0100 S  00 M30
                   R          Q              | PRG.   F:    100
INPUT PRG SEGMENT:                           | ACT.   F:      0
G00 X50 Y50 Z100 _                           | FED OVRI:   150%
                                             | RAP OVRI:   100%
                                             | SPI OVRI:   100%
                                             | PART CNT:      2
                                             | CUT TIME:  0:00:02
MDI                                                S0000 T01 H00
```

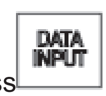
2. press 

key，the page is as follows：

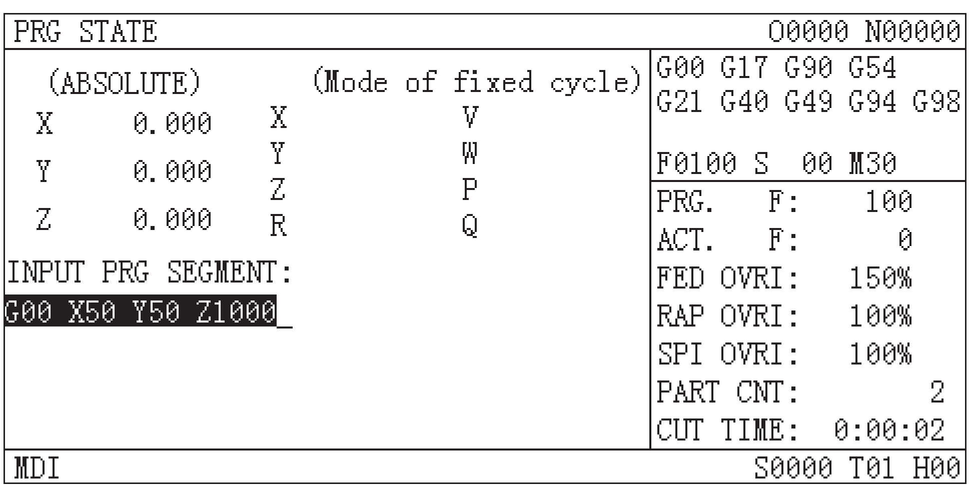


3. press. . . . by sequence, the page is as follows：

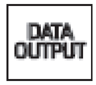
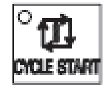


4. At last ,press

, the page is as follows：



5.5 OUT Key Start

When the “OUTR” of the K parameter K0010 is set to 1, the current words inputted may be executed by pressing  key in MDI mode. It is the same as  .

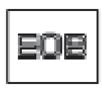
CHAPTER 6 PROGRAM EDIT AND MANAGEMENT

In Edit mode, the programs can be created, selected, modified, copied and deleted, and the bidirectional communication between CNC and CNC, or CNC and PC can also be achieved. To prevent the program to be modified or deleted accidentally, a program switch is set for this GSK980MD system. And it must be turned on before program editing. Also 3 level user authority is set in this GSK980MD system to facilitate the management. Only the operation authority is above 4 level (4 or 3 level etc.) can open the program switch for program editing.

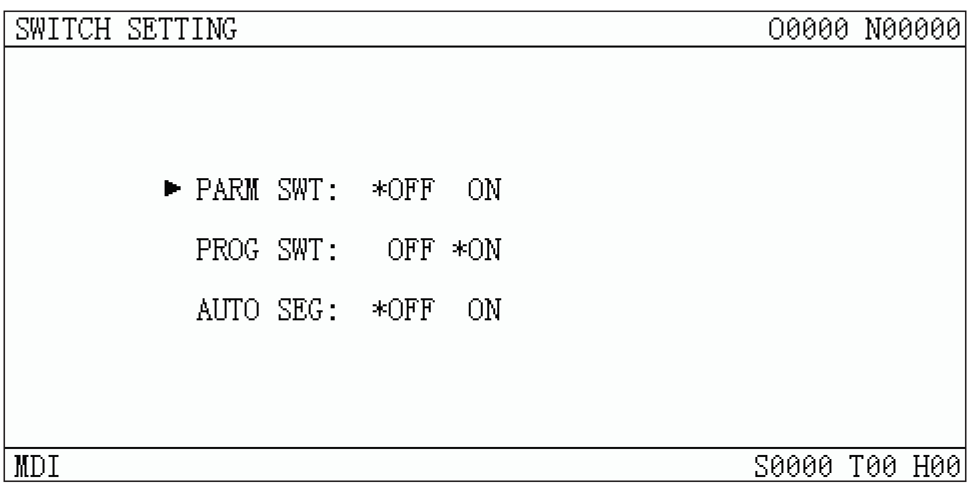
6.1 Program Creation

6.1.1 Creation of the block number

The program can be with or without a block No. The program is executed by the block numbered sequence (except the calling). When the "AUTO SEG"switch in setting page is OFF, the CNC doesn't generate the block number automatically, but the blocks may be edited manually.

When "AUTO SEG" switch in switch setting page is on, the CNC generates the block number automatically. In editing, press 

key to generate block number of the next block automatically. The increment of this block number is set by №216.

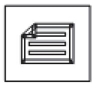
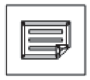


6.1.2 Input of the program content

1 Press 

key to enter the Edit mode;

2 Press 

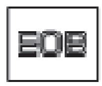
key to enter the Program interface, select the PRG CONTENT page by pressing  or  key

```
PRG CONTENT                SEG1      COL:1    C:/O0000.CNC
O0000 (O0000);
G0 G54 G90 X0 Y0 Z0;
X10 Y10;
X-10 Y-10;
M99;
%




EDIT                                           S0000 T00 H00
```

3 Key in address key O, numerical key 0, 0, 0 and 1 key by sequence (e.g. Program O0001 creation);

```
PRG CONTENT                SEG1      COL:1    C:/O0000.CNC
O0000 (O0000);
G0 G54 G90 X0 Y0 Z0;
X10 Y10;
X-10 Y-10;
M99;
%



    O0001_
EDIT                                           S0000 T00 H00
```

4 Press EOB key to setup the new program;

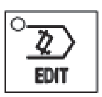
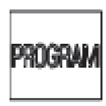
5 Input the edited part program one by one, the character will be displayed on the screen immediately as it is input(as for compound key, press this key repeatedly for alternate input),after a block is finished, press  to terminate it.

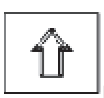
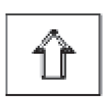
6 Other blocks cab be input by step 5 above.

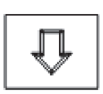
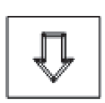
Volume II Operation

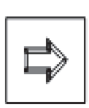
6.1.3 Search of the character

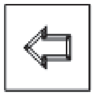
1 Scanning: To scan the character one by one by cursor

Press  key to enter the Edit mode, then press  key to enter the PRG CONTENT page;

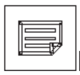
1）Press  key, the cursor shifts a line upward; if the number of the column where the cursor locates is over the total columns of the previous line, the cursor moves to the previous block end (at“;”sign) after  key is pressed;

2）Press  key, the cursor shifts a line downward; if the number of the column where the cursor locates is over the total columns of the next line, the cursor moves to the next block end (at“;”sign) after the  key is pressed;

3） Press  key, the cursor shifts a column to the right; if the cursor locates at the line end, it moves to the head of the next block;

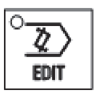
4）Press 
key, the cursor shifts a column to the left; if the cursor locates at the line head, it moves to the end of the next block;

5） Press  key to page upward, the cursor moves to the 1st line and the 1st column of the previous page, if it pages to the head of the program, the cursor moves to the 2nd line and 1st column;

6）Press  key to page downward, the cursor moves to the 1st line and 1st column of the next page, if it pages to the end of the program, the cursor moves to the last line and 1st column of the program;

2 Searching: To search for the specified character upward or downward from the cursor current location

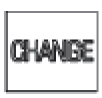
The steps of searching are as follows:

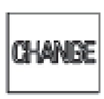
1）Press 

key to enter Edit mode;

2）Press 

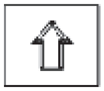
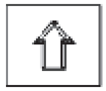
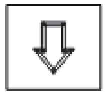
key to enter the PRG CONTENT page;

3）Press 

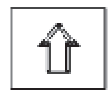
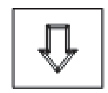
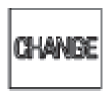
key to enter Search mode, Max. 50 bytes can be input, but only 10 of them can be searched. If the characters are over 10 bytes, searching will fail. E.g. to search command ——G2, press 

key, then input G2, and operate as step 4.

```
PRG CONTENT        ITOR SEG8     COL:1    C:/O0008.CNC
O0008 (CNC PROGRAM);
G40 G49 G80;
G0 G90 G54 X0 Y0 Z0;
Z50;
G1 X20 Z20 F1500;
G2 I-20;
G3 I-20;
G4 X5;
G1 X0 Y20 Z0 F1000;
X-20 Y0;
FIND G2_
EDIT                                    S0000 T00 H00
```

4）Press  key（  or  by the location relation between the character to be searched and the character where the cursor locates), it displays as follows:

```
PRG CONTENT          ITOR SEG8      COL:1    C:/O0008.CNC
O0008 (CNC PROGRAM);
G40 G49 G80;
G0 G90 G54 X0 Y0 Z0;
Z50;
G1 X20 Z20 F1500;
G2 I-20;
G3 I-20;
G4 X5;
G1 X0 Y20 Z0 F1000;
X-20 Y0;
FIND G2_
EDIT                                      S0000 T00 H00
```

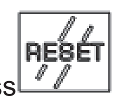
5）After the searching, the CNC system is still in searching state, press  or  key again, the next character can be searched. Or press 

key to exit the searching state.

6）If the character is not found, the prompt of “Srch fail” will be displayed.

Note:During the searching, it doesn’t search the characters in the called subprogram

3 Method to return to the program head

1) In the Program Display page of the Edit mode, press 

key, the cursor returns to the program head

2) Search the program head character by the methods in Section 6.1.3 of this part.

6.1.4 Insertion of the character

Steps:

1）Select the PRG CONTENT page in Edit mode, the page is as follows:

Volume II Operation

```
PRG CONTENT                SEG5      COL:1    C:/O0008.CNC
O0008 (CNC PROGRAM);
G40 G49 G80;
G0 G90 G54 X0 Y0 Z0;
Z50;
G1 X20 Z20 F1500;
G2 I-20;
G3 I-20;
G4 X5;
G1 X0 Y20 Z0 F1000;
X-20 Y0;

EDIT                                        S0000 T00 H00
```

2）Input the character to be inserted(to insert G98 code before G2 in the above figure, input G. 9. 8. − + ␣), the page is as follows:

```
PRG CONTENT                SEG5      COL:5    C:/O0008.CNC
O0008 (CNC PROGRAM);
G40 G49 G80;
G0 G90 G54 X0 Y0 Z0;
Z50;
G1 X20 Z20 F1500;
G98 G2 I-20;
G3 I-20;
G4 X5;
G1 X0 Y20 Z0 F1000;
X-20 Y0;

EDIT                                        S0000 T00 H00
```

Note 1:In the Insert mode, if the cursor is not located at the line head, a space will be automatically generated when inserting the command address; if the cursor is located at the line head, the space will not be generated, and it should be inserted manually.

Note 2: In program content edit mode or MDI mode of program state page，press ALTER MACRO EDIT **key to enter insertion or macro edit state.**

In macro editting mode，special symbols can be input are: ‘[’. ‘]’. ‘=’. ‘+’. ‘>’. ‘<’. ‘/’. ‘&’. ‘|’.

Above symbols are frequently used for macro edit.

Difference between two states	Automatic space	Process of character ‘O’	Input special symbols
Insertion state	In program editting, insert blank automatically to separate words.	Program switch, duplication and deletion can be done by pressing ‘O’.	Special symbols can not be inputted.
Macro edit state	Blank can not be inserted automatically.	Only input character ‘O’.	Special symbols can be inputted.

Volume II Operation

6.1.5 Deletion of the character

Steps:

1）Select the PRG CONTENT page in Edit mode;

2）Press CANCEL key to delete the character before the cursor; press DELETE key to delete the character where the cursor locates.

6.1.6 Modification of the character

Cancel or delete the character and re-enter new ones.

6.1.7 Deletion of a single block

This function is only applied to the block with a block No.(N command) , which is at the head of a line and followed by blocks which are divided by space.

Steps:

1）Select the PRG CONTENT page in Edit mode;

2）Move the cursor to the head of the block to be deleted (column 1— where N locates), then press DELETE key.

Note: If the block has no block No.N, key in “N”at the head of the block, and move the cursor to “N”, then press DELETE key.

6.1.8 Deletion of the blocks

It deletes all the content (including the specified block)from the current character where the cursor locates to the block with the specified No.(searching downward), and the specified block must has a block No..

```
PRG CONTENT                 SEG5      COL:5    C:/O0008.CNC
O0008 (CNC PROGRAM);
G40 G49 G80;
G0 G90 G54 X0 Y0 Z0;
Z50;
G1 X20 Z20 F1500;
N10 G98 G2 I-20;
G3 I-20;
G4 X5;
G1 X0 Y20 Z0 F1000;
X-20 Y0;

EDIT                                          S0000 T00 H00
```

Steps

1）Select the PRG CONTENT page in Edit mode;

2）Press CANCEL key to enter the FIND state, and key in the block No.

```
PRG CONTENT                 SEG2      COL:1    C:/O0008.CNC
O0008 (CNC PROGRAM);
G40 G49 G80;
G0 G90 G54 X0 Y0 Z0;
Z50;
G1 X20 Z20 F1500;
N10 G98 G2 I-20;
G3 I-20;
G4 X5;
G1 X0 Y20 Z0 F1000;
X-20 Y0;
FIND N10_
EDIT                                          S0000 T00 H00
```

3）Press DELETE key to delete blocks from G0 (block 2) to N10 (including block N10). It displays as follows:

```
PRG CONTENT                 SEG2      COL:1    C:/O0008.CNC
O0008 (CNC PROGRAM);
G40 G49 G80;
G3 I-20;
G4 X5;
G1 X0 Y20 Z0 F1000;
X-20 Y0;
X0 Y-20 Z-10;
X20 Y0 Z-20;
X5 Y5 Z-50;
M99;

EDIT                                          S0000 T00 H00
```

6.1.9 Segment deletion

It deletes the content downward from the current character where the cursor locates to the word specified.

```
PRG CONTENT                SEG2      COL:4    C:/O0008.CNC
O0008 (CNC PROGRAM);
G40 G49 G80;
G3 I-20;
G4 X5;
G1 X0 Y20 Z0 F1000;
X-20 Y0;
X0 Y-20 Z-10;
X20 Y0 Z-20;
X5 Y5 Z-50;
M99;

EDIT                                      S0000 T00 H00
```

Steps

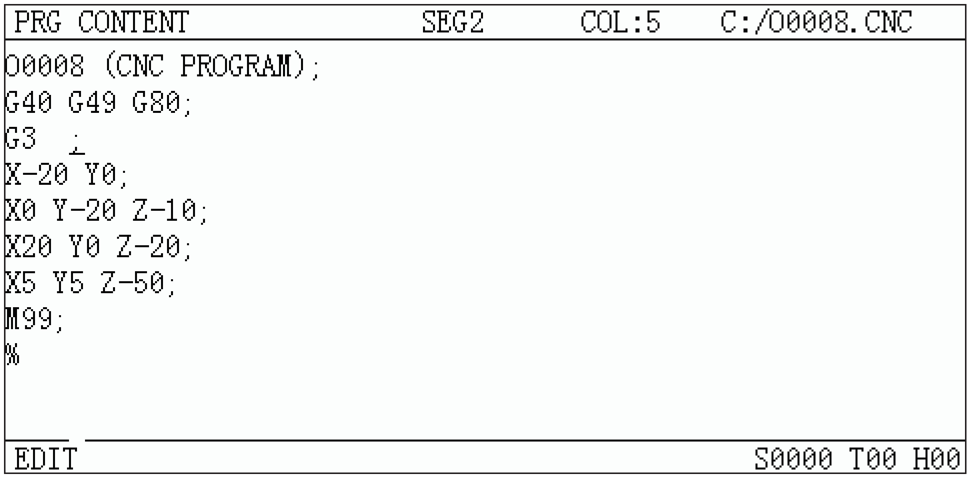
1）Select the PRG CONTENT page in Edit mode

2）Press CANCEL key to enter the FIND state, and key in the characters (see the following figure: input F1000)

```
PRG CONTENT                SEG2      COL:4    C:/O0008.CNC
O0008 (CNC PROGRAM);
G40 G49 G80;
G3 I-20;
G4 X5;
G1 X0 Y20 Z0 F1000;
X-20 Y0;
X0 Y-20 Z-10;
X20 Y0 Z-20;
X5 Y5 Z-50;
M99;
FIND F1000_
EDIT                                      S0000 T00 H00
```

3）Press DELETE key, and all programs from I-20 where the cursor locates to F1000. It displays as follows:

Volume II Operation



Note 1:If the specified character is not found or the specified character is located before the current cursor, the prompt of "Srch fail" will be displayed. If there are multiple same characters specified downward, it defaults the nearest one to the current cursor.

Note 2: If the command address is input, both the address and the command value behind it are Deleted.

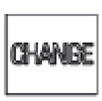
6.2 Program annotation

To facilitate the user to search, manage and edit program, the system provides program name annotation and block annotation functions.

6.2.1 Annotation for program name

The program annotation can be added in the brackets behind it. For exa mple: program O0005 is used for machining bolt holes, the annotation can be added in program contents as follows:

1）Select edit mode, and then enter program content display page.

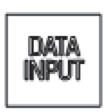
2）Press 

key，search is displayed at the left bottom of the screen, the displayed figure is as follows:

```
PRG CONTENT                  SEG1      COL:1    C:/O0005.CNC
O0005 (O0005);
G90 G00 X0 Y0 Z0;
(I:cir r,A:first hole angle,B:angle inc,H:hole number);
G65 P9020 X100 Y50 R30 Z-50 F1800 I100 A45 B30 H5;
M30;
%

FIND
EDIT                                           S0000 T00 H00
```

3）Input annotation behind search (input max. 50 characters except for brackets). If BOLT PROC is inputted (bolt holes machining), the page displayed is as follows:

```
PRG CONTENT                  SEG1      COL:1    C:/O0005.CNC
O0005 (O0005);
G90 G00 X0 Y0 Z0;
(I:cir r,A:first hole angle,B:angle inc,H:hole number);
G65 P9020 X100 Y50 R30 Z-50 F1800 I100 A45 B30 H5;
M30;
%

FIND BOLT PROC_
EDIT                                           S0000 T00 H00
```

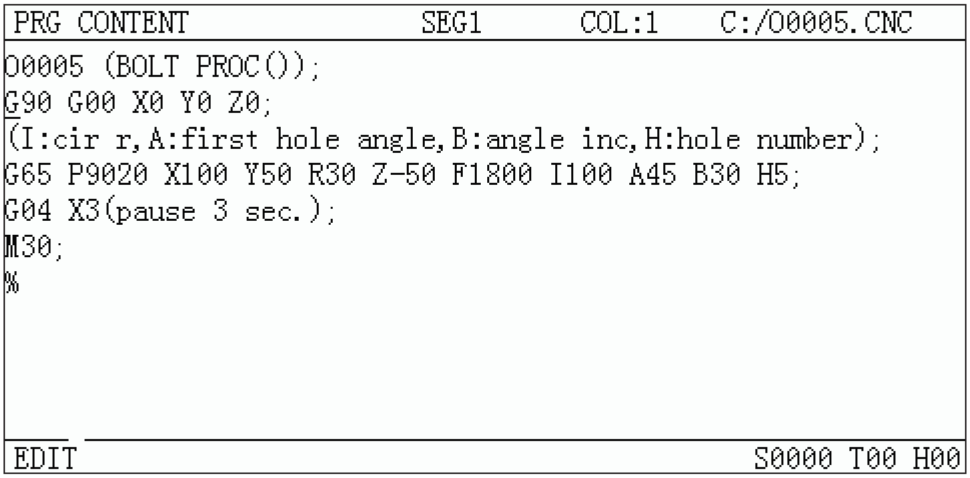
4）Press 

key，program annotation setting up is finished，the displayed page is as follows：

```
PRG CONTENT                  SEG1      COL:1    C:/O0005.CNC
O0005 (BOLT PROC);
G90 G00 X0 Y0 Z0;
(I:cir r,A:first hole angle,B:angle inc,H:hole number);
G65 P9020 X100 Y50 R30 Z-50 F1800 I100 A45 B30 H5;
M30;
%

EDIT                                           S0000 T00 H00
```

6.2.2 Block annotation

Take contents in brackets '（'and'）'as program annotation, which can be put at any position of a block and displayed with green characters. The page is as follows:



Related explanations：

1）Because symbols'（'and '）'are not provided in the system，block annotation can not be inputted by edit mode in the system. If block annotation is needed to added, edit annotation on the PC and download it to the CNC by software.

2）The system is not support Chinese characters. If Chinese characters are edited on PC, which will be displayed as blanks in the system after it is saved in the CNC.

Note 1: After a program is set up, if the program name annotation is not added, CNC defaults program name as program name annotation

Note 2: Program annotation in the CNC must be English, but the CNC supports Chinese annotation display (except for Chinese decimal points). The way of adding Chinese annotation is as follows: Edit Chinese annotation in the PC machine, and then download it to the CNC by communication software.

6.2.3 Alter program annotation

Operation steps are the same as program annotation setting steps on section 6.2.1 of this chapter.

6.3 Deletion of the Program

6.3.1 Deletion a single program

Steps:

1）Select the PRG DISPLAY page in Edit mode;

2）Key in address key O, numerical key 0. 0. 0. 1 by sequence(take program O0001 for an example);

3) Press key, program O0001 will be deleted

Note: Press 'DELETE ' key in page 'program preview'or'file list'to delete program.

6.3.2 Deletion of all programs

Steps

1）Select the PRG DISPLAY page in Edit mode

2）Key in address key O , symbol key − + ␣ numerical key 9. 9. 9. 9 by sequence

3）Press DELETE key, all the programs will be deleted.

Note: Press 'delete key'in page 'file list'to delete all programs.

6.4 Selection of the Program

When there are multiple programs in CNC system, a program can be selected by the following 4 methods:

6.4.1 Search method

1）Select Edit mode;

2）Press PROGRAM key to enter the PRG CONTENT page;

3）Press address key O and key in the program No.;

4）Press ⇩ or EOB key, the searched program will be displayed.

Note:In step 4, if the program does not exist, a new program will be created by CNC system after EOB key is pressed

6.4.2 Scanning method

1）Select Edit or Auto mode;

2）Press PROGRAM key to enter the PRG DISPLAY page;

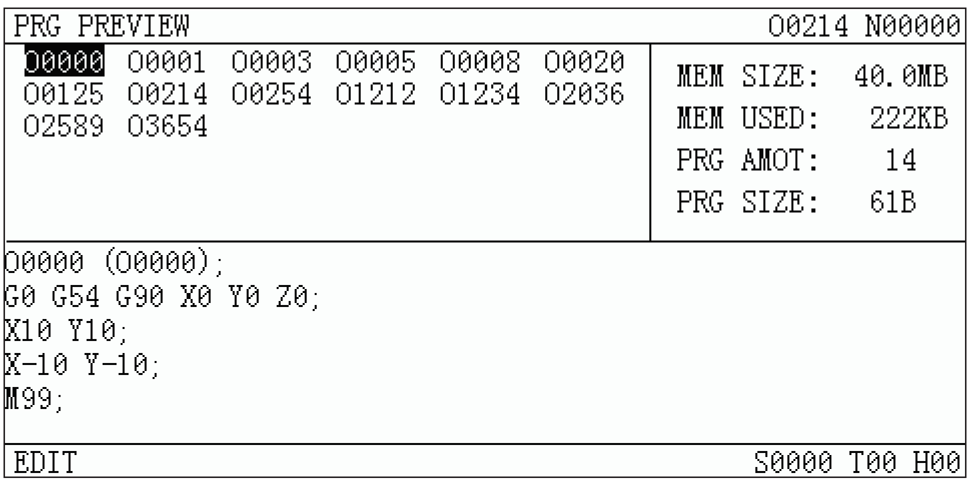
3）Press address key O

4）Press ⇩ or ⇧ key to display the next or previous program;

5）Repeat step 3 and 4 to display the saved programs one by one.

6.4.3 Cursor method

1）In Program Preview mode **(must be in non-running state)**;

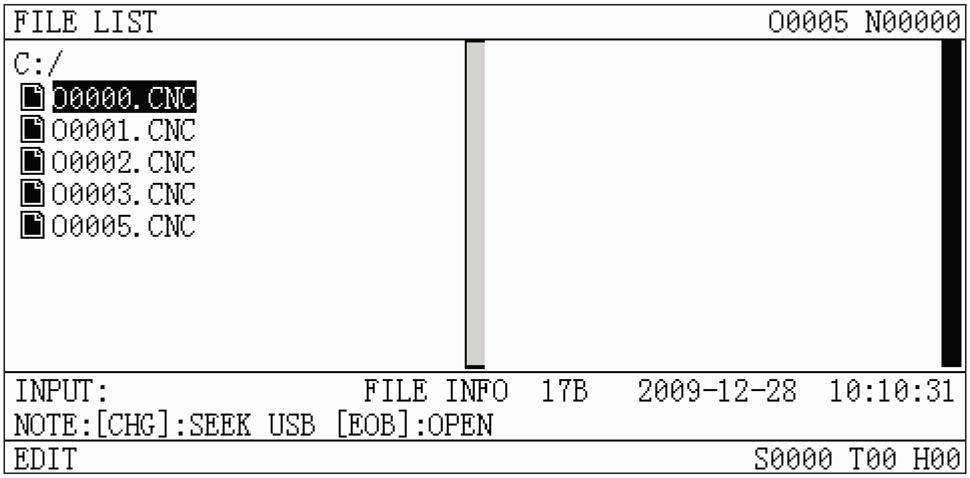


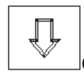
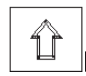
2）Press ⇧ . ⇩ . ⇦ or ⇨ key to move the cursor to the program name to be selected (change “PRG SIZE”, “NOTE” content as the cursor moves);

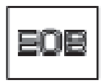
3）Press EOB to open the program.

6.4.4 Select file by using file list

1） On file list page（Edit mode is operation mode）

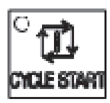


2）Select program to be opened by pressing or key.

3) Open program by pressing 

key.

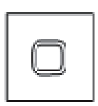
6.5 Execution of the Program

After the program to be executed is selected by the method in Section 6.4 of this part, select the Auto mode, then press 

key (or press external cycle start key), the program will be executed automatically.

6.6 Rename of the Program

1）Select the PRG CONTENT page in Edit mode;

2）Press address key  and key in the new program name;

3）Press 

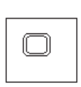
key.

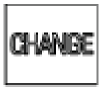
Note: No matter whether the program is altered or not, program annotation is changed into new program name automatically after program is renamed.

6.7 Copy of the Program

To save the current program to a location:

1）Select the PRG CONTENT page in Edit mode;

2）Press address key  and key in the new program No

3) Press 

key.

6.8 Program positioning

- To the position where the program stops last time by TO

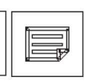
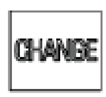
  Search for the point where the program execution stops by TO. Select edit mode to enter program content page and press conversion key, input TO to search which is displayed at the left bottom. Then press up or down key, searching and positioning are displayed at this time, the cursor will move to the position where program stops last time.

- Position to specified block by TO＋num（num is the block number specified by user. For example: TO10000 means position to the 10000th block）

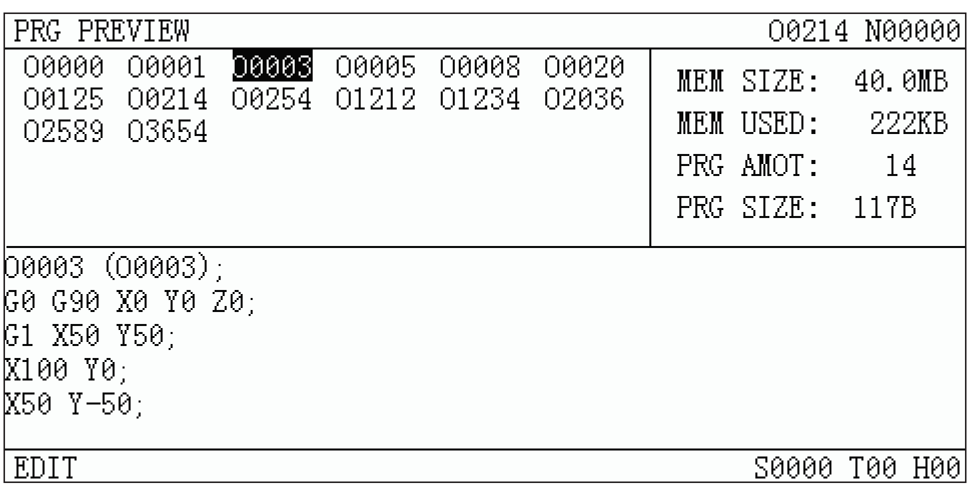
  On program content page, locate to specified block by inputing TO block number. Press conversion key after entering program content page, input TO to search which is displayed at the left bottom and then press up or down key, the cursor will move to the specified program.

6.9 Program preview

In non-edit mode，press 

key to enter program preview page. In this page, program names saved in CNC are displayed in the form of list. Max. 36 program names can be displayed In one page, if programs saved are over 36, press   

key to display programs in other page.



- Program capacity display：

On top right window, “storage capacity”displays the max. capacity of program which can be saved in CNC. “Used capacity”displays the capacity of saved program in CNC system.. “Program number"displays the program number saved in the CNC system. "Program size"displays the size of the currently opened program.

- Program preview selection：

On top left of the window, the name of currently previewed program will be displayed in blue characters on white ground. Program size on top left window is the size of currently previewed program. The following window displays currently previewed progam, display 5-line program.

- Usage of cursor key and conversion key：

When select program in a program list, select the program to be previewed by cursor moving key on MDI panel. If the size is very big, max. 36 program names can be displayed in program list. Select program by pressing right moving key or pressing conversion key directly, turn pages to display the program list, and then select it by cursor moving key on MDI panel.

- Open a program：

In edit, auto, MDI modes, when open the program on program preview window, this program can be opened by pressing EOB key on MDI panel. At the same time, the name of currently opened program is displayed on top right page.

- Deletion of program

Move cursor to the program will be deleted, press delete key and then press Y key or N key on multiple select manue to select wether delete it or not

CHAPTER 7 AUTO OPERATION

Note！

The keys functions of this 980MDa machine panel are defined by Ladder; please refer to the respective materials by the machine builder for the function significance.

Please note that the following function introduction is described based on the

7.1Auto Run

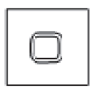
7.1.1 Selection of the program to be run

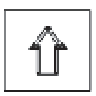
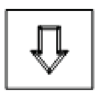
1. Search method

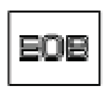
1）Select the Edit or Auto mode;

2）Press 

key to enter the PRG CONTENT page;

3）Press the address key  and key in the program No.

4）Press  or  key, the program retrieved will be shown on the screen, if the program doesn’t exist an alarm will be issued

Note In step 4, if the program to be retrieved does not exist, a new program will be setup by CNC system after pressing 

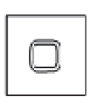
key.

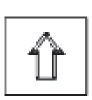
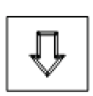
2 Scanning method

1）Select the Edit or Auto mode

2） Press 

key to enter the PRG display page

3）Press the address key 

4）Press the  or  key to display the next or previous program;

5）Repeat the step 3, 4 above to display the saved program one by one.

3 Cursor method

a) Select the Auto mode (must in non-run state)

b) Press PROGRAM key to enter the PRG LIST page;

c) Press ⇧ . ⇩ . ⇦ . ⇨ key to move the cursor to the name of the program to be selected;

d) Press EOB key.

4. File open method

Select the edit or operation mode:

1）Press PROGRAM key twice to enter the page of file list.；

2）Press ⇧, ⇩ keys to move the cursor to the file will be selected.

3）Press DATA INPUT key to select a file.

4）Press EOB key to open the selected file.

Note: The file can not be opened if the expanded name is not“.CNC”.

7.1.2 Program start

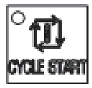
1. Press AUTO key to select the Auto mode

2. Press CYCLE START key to start the program, and the program execution begins

Note Since the program execution begins from the block where the cursor locates, before pressing the CYCLE START key, make a check whether the cursor is located at the block to be executed. If begins from the start line, but the cursor is not in this line, move the cursor to the line.

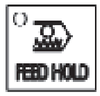
7.1.3 Stop of the auto run

- **Stop by command (M00)**

the block containing M00 is executed, the auto run is stopped. So the modal function and state are all reserved. Press the key 

or the external Run key, the program execution continues.

- **Stop by a relevant key**

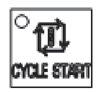
1 In Auto run, by pressing key 

or external dwell key, the machine remains at the following state:

（1）The machine feed decelerate to stop;

（2）During the execution of the dwell command (G04), it pauses after G04 command execution is finished.

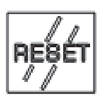
（3）The modal function and state are saved;

（4）The program execution continues after pressing the 

key

2 Stop by Reset key 

（1）All axes movement is stopped.

（2）M, S function output is invalid (the automatic cut-off of signals such as spindle CCW/CW, lubrication, cooling by pressing 

key can be set by the parameters)

（3）Modal function and state is held on after the auto run.

3 Stop by Emergency stop button

If the external emergency button (external emergency signal valid) is pressed under the dangerous or emergent situation during the machine running, the CNC system enters into emergency state, and the machine moving is stopped immediately, all the output (such as spindle rotation, coolant) are cut off. If the Emergency button is released, the alarm is cancelled and CNC system enters into reset mode.

4 By Mode switching

When the Auto mode is switched to the Machine zero, MPG/Step, the current block “dwells”immediately; when the Auto mode is switched to the Edit, MDI mode, the “dwell”is not displayed till the current block is executed.

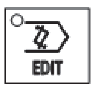
Note 1 Ensure that the fault has been resolved before cancelling the emergency alarm.

Note 2 The electric shock to the device may be decreased by pressing the Emergency button before power on and off.

Note 3 The Machine zero return operation should be performed again after the emergency alarm is cancelled to ensure the the coordinate correctness (but this operation is unallowed if there is no machine zero in the machine)

Note 4 Only the BIT3 (ESP) of the bit parameter No.017 is set to 0, could the external emergency stop be valid.

7.1.4 Auto run from an arbitrary block

1. Press 

key to enter the Edit mode, press 

key to enter the Program interface, or press 

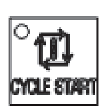
key several times to select the PRG CONTENT page:

2. Move the cursor to the block to be executed (for example, move the cursor to the 3th line head if it executes from the 3th line);

```
PRG CONTENT              SEG3     COL:1    C:/O0000.CNC
O0000 (O0000);
G0 G54 G90 X0 Y0 Z0 G49;
G01 X100 Y100 F500;
G02 I20;
G01 X52 Z01;
G91 X2 Z-6.3;
G00 X0 Y0 Z0;
M30;
%

EDIT                                      S0000 T00 H00
```

3. If the mode （G, M, T, F command）of the current block where the cursor locates is defaulted and inconsistent with the running mode of this block, the corresponding modal function should be executed to continue the next step.

4. Press 

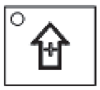
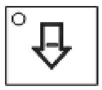
key to enter the Auto mode, then press 

key to start the program.

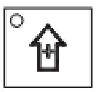
7.1.5 Adjustment of the feedrate override, rapid override

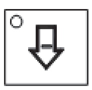
In Auto mode, the running speed can be altered by adjusting the feedrate override, rapid override with no need to change the settings of the program and parameter.

- Adjustment of the feedrate override

Press the  or  key in 

, it can realize 16-level real time feedrate adjustment.

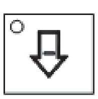
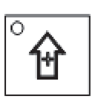
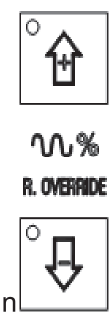
Press the  key each time, the feedrate override ascends a gear level till 150%

Press the  key each time, the feedrate override decends a gear level till 0;

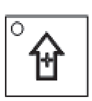
Note 1 The actual feedrate value is specified by F in program feedrate override adjustment;

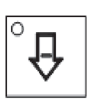
Note 2 Actual feedrate= value specified by F× feedrate override

- Adjustment of rapid override

It can realize the 4-level real time rapid override Fo. 25%. 50%. 100% adjustment by pressing the  or  key in 

.

Press the  key each time, the rapid override ascends a level till 100%;

Press the  key each time, the rapid override decends a level till F0

Note 1 The max. rapid traverse speeds of X, Y, Z axis are set by bit parameter No.059, No.060, No.061 respectively;

X axis actual rapid traverse rate = value set by parameter No.059×rapid override

Y axis actual rapid traverse rate = value set by parameter No.060×rapid override

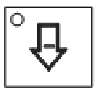
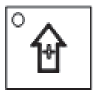
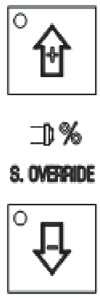
Volume II Operation

Z axis actual rapid traverse rate = value set by parameter No.061×rapid override

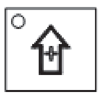
Note 2 When the rapid override is F0, the rapid traverse rate is set by bit parameter No.069.

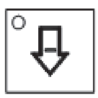
7.1.6 Spindle override adjustment

While the spindle speed is controlled by the analog voltage output in Auto mode, it can be adjusted by spindle override.

Press the  or  key in 

to adjust the spindle override for the spindle speed, it can realize 8-level real-time override adjustment between 50％～120％.

Press the  key each time, the feedrate override ascends a level till 120%

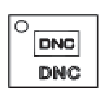
Press the  key each time, the rapid override decends a level till 50%.

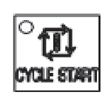
Note 1 The actual output analog voltage=analog voltage by parameter×spindle override

Example: When the bit parameter No.101 is set to 9999, No.100 to 645, execute S9999 command to select the spindle override 70%, the actual output analog voltage≈10×70%=7V

7.2 DNC running

This CNC system has a DNC function, by the connection of the DNC communication software with this system, the high speed, high capacity program can be performed in this system.

In Auto mode, press the 

key, it enters the DNC mode. Then press the 

key to start the program DNC machining under the condition that the PC is get ready

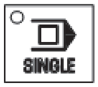
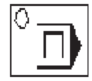
Please refer to the DNC communication software for details.

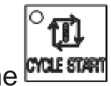
7.3 Running state

7.3.1 Single block execution

When the program is to be executed for the 1st time, to avoid the programming errors, it may select Single block mode to execute the program.

In Auto mode, the methods for turning on single are as follows.

Press the  key to make the single block indicator  in State area to light up, it means that the single block function has been selected

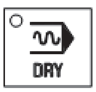
In Single block mode, when the current block execution is finished , the CNC system stops;if next block is to be executed,it needs to press the  key.

Note Even at the mid point, the single block stops in G28,G29, G30 commands

7.3.2 Dry run

Before the program is to be executed, in order to avoid the programming errors, it may select the Dry run mode to check the program. And the machine runs by a constant speed other than the speed specified by the program.

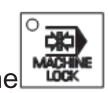
In Auto mode, the method for turning on the Dry run switch are as follows.

Press  key to make the dry run indicator in State area to light up, it means that the dry run function is selected

The speed specified by the program is invalid in Dry run, and actural feedrate is set by the DATA parameter No.174.

7.3.3 Machine lock

In Auto mode, the ways to make machine lock function valid are as follows.

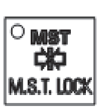
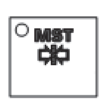
Press the  key to make the machine lock indicator  in State area to light up, it means that it has enterd the machine lock state.

While in the machine lock mode:

1. The machine carriage doesn't move, the "MACHINE"in the INTEGRATED POS page of the POSITION interface doesnt' vary too. The RELATIVE POS and ABSOLUTE POS, DIST TO GO are refreshed normally
2. M, S, T commands can be executed normally.

7.3.4 MST lock

In Auto mode, the ways to make MST lock function valid are as follows.

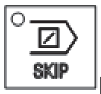
Press the  key to make the MST lock indicato  in State area to light up, it means that it has entered the MST lock state. And the carriage move is not performed by M, S, T commands

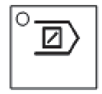
Note: When the MST lock is valid, it has no effect on the execution of M00, M30, M98,M99.

7.3.5 Block skip

If a block in program is not needed to be executed and not to be deleted, this block skip function can be used. When the block is headed with “/”sign and Block skip function is valid, this block is skipped without execution in Auto mode

In Auto mode, the way to make block skip function valid is as follows.

Press the 

key to make the block skip indicator  in State area to light up, it means that the block skip function is valid.

Note While the block skip function is invalid, the blocks headed with “/”signs are executed normally in Auto mode.

7.3.6 Optional stop

In AUTO mode, the valid optional stop function is as follows:

Press 

key to enter optional stop and the indicator lights up.

The program will be “stopped” at command M01. Press 

key again to continue program execution.

7.4 Memorizing at power-down

7.4.1 Program interruption in non-DNC auto operation

Operation method 1 (Manual)

1. After power on, press conversion key →press letter “T”+letter“O”→up, down moving keys on pages“program content, edit” to the block where the execution stops last time.
2. Switch to the pages “coordinate & program, machine zero”.
3. Enter the next step after machine zero is performed.
4. Switch to manual or MDI mode. Locate to the block where it stops last time. (At this moment, it is necessary to confirm whether it is at state G40, G49, G54. Ensure that tools are in a safe range during positioning.)
5. Switch to manual mode, press conversion key. It prompts “Locate to the block where it stops last time. It will recover the mode before power-down（Y/N）”.
6. Press Y to recover the mode before power-down.
7. Switch to auto mode, press cycle start key to execute the block continuously from where it stops last time.

Operation method 2 (Auto)

1. After power on, press conversion key →press letter "T"+letter"O"→up, down moving keys on pages"program content, edit" to the block where the execution stops last time.
2. Switch to the pages "coordinate & program, machine zero".
3. Perform machine zero operation.
4. After machine zero is performed, press conversion key. It prompts at the bottom of the screen: "Locate to the block automatically where it stops last time. It will recover the mode before power-down（Y/N）". Input Y (Ensure that tools moving path is in a safe range at this moment.). Coordinates start move, it locates to the block where it stops last time, and recovers the mode before power-down.
5. Switch to auto mode, press cycle start key to execute the block continuously where it stops last time.

7.4.2 Interruption at power-down on DNC auto operation

Operation method (Auto)

1. Switch to "coordinate program, machine zero return" after power on.
2. Execute machine zero return.
3. After machine zero return is finished, press conversion key. It prompts at the bottom of the screen: "Locate to the block automatically where it stops last time. It will recover the mode before power-down（Y/N）". Input Y (Make sure tools moving path is in a safe range at this moment.). Coordinates start move, it locates to the block where it stops last time, and recovers the mode before power-down.
4. Switch to the highlighted block when DNC, CNC power down.
5. Search for the interrupted block in DNC transmission software, then press RESET key on panel to continue PC software transmission. Press cycle start key to continue execution.

CHAPTER 8 MACHINE ZERO RETURN OPERATION

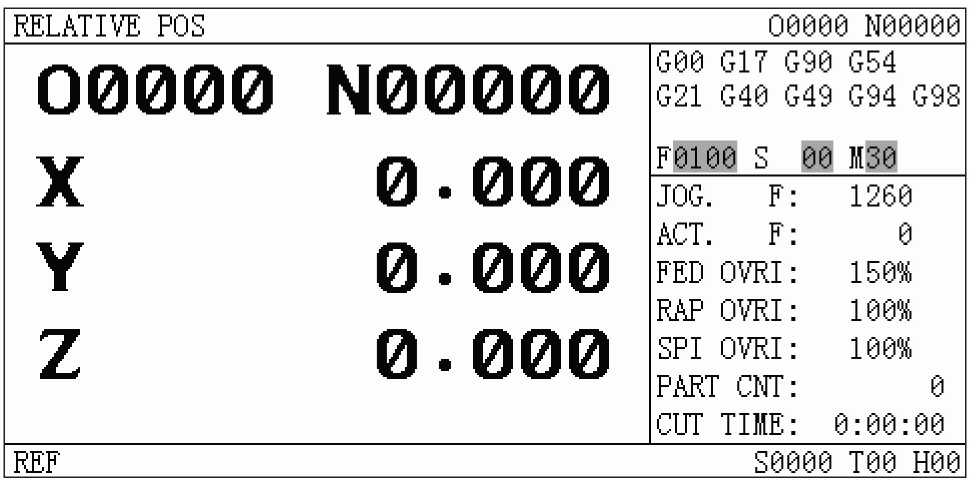
8.1 Machine Zero

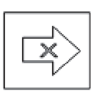
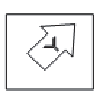
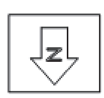
The machine coordinate system is a basic coordinate system for CNC coordinate calculation. It is an inherent coordinate system of the machine. The origin of the machine coordinate system is called **machine zero** (or mechanical reference point). It is defined by the zero return switches fixed on the machine. Usually the switch is fixed on the positive max. Strokes of X, Y, Z axes.

8.2Machine Zero Return Steps

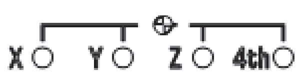
1 Press 

key, it enters the Machine zero mode, the bottom line of the screen page shows "REF", the figure is as follows:



2 Press  or  or  key to select the machine zero of X, Y or Z axis

3 The machine moves along the machine zero direction, and returns to the machine zero via the deceleration signal, zero signal detection. And the axis stops with the machine zero finish indicator lighting up.



Machine zero finish indicators

Note1：If the machine zero is not fixed on the machine, machine zero operation B/C/D is unallowed.

Note2：While the coordinate is moved out from the machine zero, the machine zero finish indicators go out.

Note3：After the machine zero operation, the cancellation of the tool length offset for the

CNC is set by the BIT7 of the bit parameter No.22

Note4：See details in the 3rd part INSTALLATION AND CONNECTION for the parameters concerning with the machine zero.

Note 5: When machine zero return, bit parameter №011 ZNIK determines whether axis movement is locked automatically.

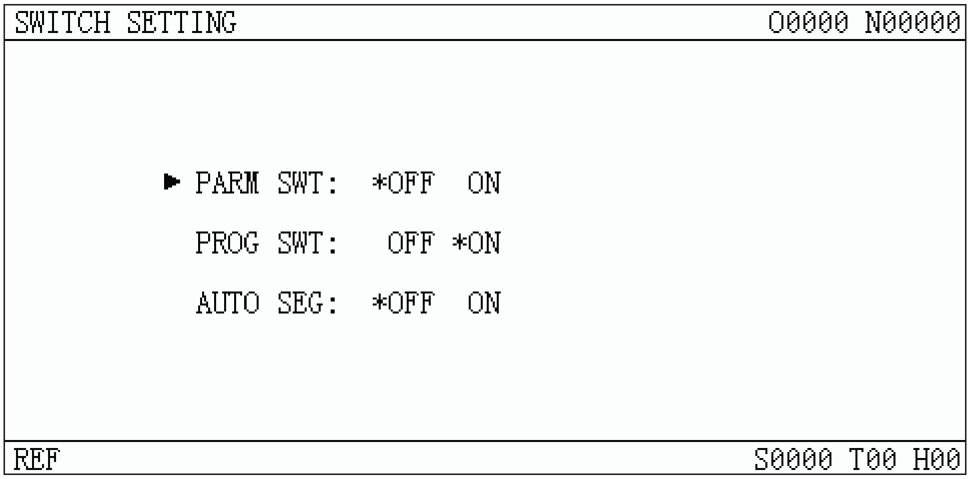
Note 6: Only machine zero D mode can be used for rotary axis.

CHAPTER 9 DATA SETTING, BACKUP and RESTORE

9.1Data Setting

9.1.1 Switch setting

In SWITCH SETTING page, the ON-OFF state of PARM SWT (parameter switch), PROG SWT (program switch), AUTO SEG (auto sequence No.) can be displayed and set, the figure is as follows:



1 Press key to enter the Setting interface, then press or key to enter SWITCH SETTING page

2 Press or key to move the cursor to the item to be set

3 Press . and . key to shift the ON-OFF state, press or key, "*"moves to the left to set the switch for OFF, Press or key, "*"moves to the right to set the switch for ON.

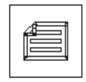
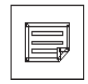
Only the PARM SWT is set to ON, could the parameter be altered; so are PROG SWT and AUTO SEG

Note 1: When parameter switch is shifted from "off"to"on"for the first time, CNC alarm occurs. Press , keys together to eliminate the alarm. Alarm will not occur when parameter switch is shifted again. For security, set parameter switch to "off" after parameter alteration is finished.

Note 2: When parameter switch is shifted from "off"to"on", CNC alarm occurs. Alarm will occur again when parameter switch is shifted from "on"to"off"for the first time. Press , keys together to eliminate the alarm.

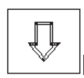
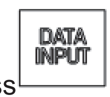
9.1.2 Graphic setting

Press 

key to enter graphic interface. Press  or  key to access the following graphic parameter page.

```
GRAPH SET                                              O0000 N00000
  COOR OPT=         0 0XY 1YX 2ZX 3XYZ 4YZ 5ZY 6XZ 7XZY)
  SCALE   =      100%
  CENTER  =     0.000 (X axis value)
  CENTER  =     0.000 (Y axis value)
  CENTER  =     0.000 (Z axis value)
► X MAX.  =   120.000
  Y MAX.  =   120.000
  Z MAX.  =   120.000
  X MIN.  =  -120.000
  Y MIN.  =  -120.000
  Z MIN.  =  -120.000

REF                                                  S0000 T00 H00
```

A：The way of setting graphic parameter

1. In MDI mode, press  or key to move the cursor to the parameter to be set,
2. Input corresponding valus,
3. Press 

key，and the setting is finished.

B：Significance of graphic parameter

Coordinate selection: Display view angle of the graphic path can be selected by setting different values. Corresponding coordinate for 0~7is as follows.

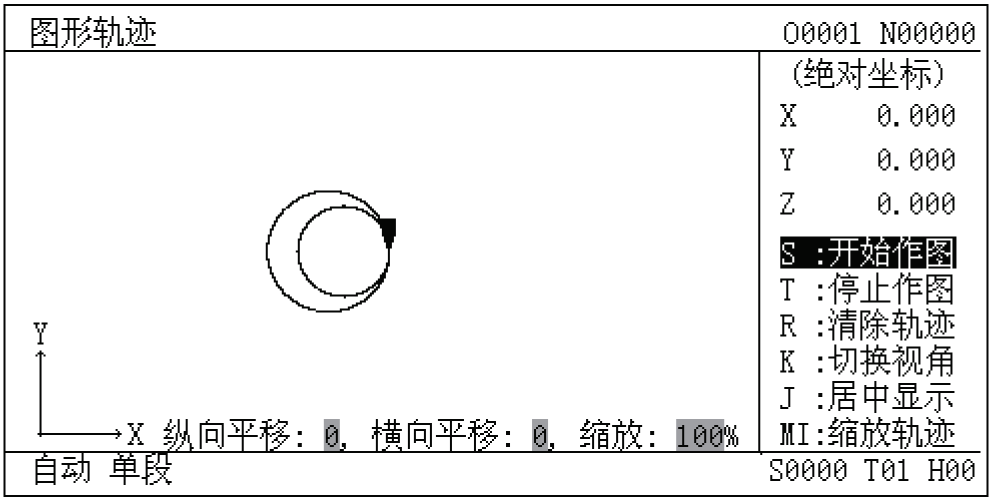
Scaling: Display the scaling of current graphic path.

Graphic center: Display the center of each axis.

Maximum, minimum: Set the maximum and minimum scope can be displayed by each axis.

C： Graphic track operation

Graphic track is as follows:



Vertical move: Display upper and lower part of the graphic.

Horizontal move: Display right and left part of the graphic.

Scaling: Display scaling of current graphic.

Absolute coordinate: Display the absolute coordinate of the program.

S：Start drawing, S is highlighted by pressing S key. Display drawing track.

T：Stop drawing, T is highlighted by pressing S key. I t stops drawing.

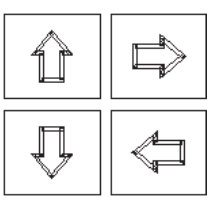
R：Clear graphic track, clear graphic track displayed before.

K：Switch view angle, coordinate value can be switched between 0~7 by pressing K key each time.

J： Display graphic in the center, that is, vertical move and horizontal move are 0.

I：Scale up the track, the graphic is scaled up 2 fold by pressing I key once.

M：Scale down the track, the graphic is scaled down 2 fold by pressing M key once.


：Graphic moving up, down, left ,right.

9.1.3 Parameter setting

By the parameter setting, the characteristics of the drive unit and machine can be adjusted. See Appendix 1 for their significance

Press PARAMETER key to enter the Parameter interface, then press or key to switch the parameter page, the figure is as follows:

```
BIT PARAMETER                                          O0000 N00000
   NO.      DATA         NO.       DATA         NO.        DATA
  _001     00000000      009     00011111       017     00101000
   002     00000010      010     00011111       018     00000000
   003     00000000      011     00000000       019     10000000
   004     01000000      012     00010011       020     00000000
   005     00010001      013     10000011       021     00000000
   006     00000000      014     00011111       022     00000000
   007     00000000      015     10000000       023     00000000
   008     00011111      016     00000000       024     00000000
*** *** ***  ACS  HWL  ***  *** ***
bit4:1/0:Analog vol./switch ctrl spindle
NO.  001
AUTO    SBK                                          S0000 T00 H00
```

A Alteration of the bit parameter

1 Byte alteration

1） Turn on the parameter switch

2） Enter the MDI mode

3） Move the cursor to the parameter No. to be set

Method 1: Press or key to enter the page containing the parameter to be set, press ⇧ or ⇩ key to move the cursor to the No. of the parameter to be set;

Method 2: Press address key P Q, key in parameter No, then press DATA INPUT key.

4） Key in the new parameter value

5） Press DATA INPUT key, the parameter value is entered and displayed

6） For security , the PARM SWT needs to be set to OFF after all parameters setting is finished

Example:

Set the BIT5 (DECI) of the bit parameter No.004 to 1, and the other bits unchanged.

Move the cursor to No.004, key in 01100000 by sequence in the prompt line, the figure is as follows:

```
BIT PARAMETER                                    O0000 N00000
   NO.       DATA       NO.       DATA       NO.       DATA
   001   00000000       009   00011111       017   00101000
   002   00000010       010   00011111       018   00000000
   003   00000000       011   00000000       019   10000000
  _004   00100000       012   00010011       020   00000000
   005   00010001       013   10000011       021   00000000
   006   00000000       014   00011111       022   00000000
   007   00000000       015   10000000       023   00000000
   008   00011111       016   00000000       024   00000000
*** RDRN DECI *** PROD *** *** SCW
bit5:1/0:DEC signal is low/high level
NO.  004 =  01100000_
MDI                                            S0000 T00 H00
```

Press 

key to finish the parameter alteration. The page is as follows:

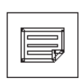
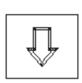
```
BIT PARAMETER                                    O0000 N00000
   NO.       DATA       NO.       DATA       NO.       DATA
   001   00000000       009   00011111       017   00101000
   002   00000010       010   00011111       018   00000000
   003   00000000       011   00000000       019   10000000
  _004   01100000       012   00010011       020   00000000
   005   00010001       013   10000011       021   00000000
   006   00000000       014   00011111       022   00000000
   007   00000000       015   10000000       023   00000000
   008   00011111       016   00000000       024   00000000
*** RDRN DECI *** PROD *** *** SCW
bit5:1/0:DEC signal is low/high level
NO.  004 =
MDI                                            S0000 T00 H00
```

2 Bit alteration

1） Turn on the parameter switch

2） Enter the MDI mode

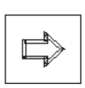
3） Move the cursor to the No. of the parameter to be set

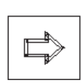
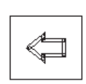
Method 1: Press  or  key to enter the page of the parameter to be set, press  or  key to move the cursor to the No. of the parameter to be set

Method 2: Press address key 

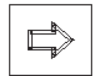
key in parameter No., then press 

key

4） Press and hold  key for 2 seconds or press 

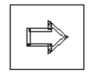
key to skip to a bit of the parameter, and the bit is backlighted. Press  or  key to move the cursor to the bit to be altered, then key in 0 or 1

5） After all parameters setting is finished, the PARM SWT needs to be set for OFF for security

Note: After entering a bit of the parameter, press and hold  key for 2 seconds or press 

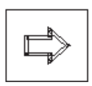
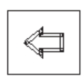
key, it may skip out of the bit and back to the parameter No.

Example:

Set the BIT5 (DECI) of the bit parameter No.004 to 1, and the other bits unchanged Move the cursor to "No.004" by the steps above, press and hold  key for 2 seconds or press 

key to skip to a bit of the parameter, the figure is as follows:

```
BIT PARAMETER                                      O0000 N00000
   NO.        DATA        NO.        DATA        NO.        DATA
   001    00000000        009    00011111        017    00101000
   002    00000010        010    00011111        018    00000000
   003    00000000        011    00000000        019    10000000
  _004    01100000        012    00010011        020    00000000
   005    00010001        013    10000011        021    00000000
   006    00000000        014    00011111        022    00000000
   007    00000000        015    10000000        023    00000000
   008    00011111        016    00000000        024    00000000
*** RDRN DECI *** PROD *** *** SCW
bit7:1/0:Unused
NO.  004
MDI                                               S0000 T00 H00
```

Move the cursor to "BIT5" by pressing  or  key, the figure is as follows:

```
BIT PARAMETER                                      O0000 N00000
   NO.        DATA        NO.        DATA        NO.        DATA
   001    00000000        009    00011111        017    00101000
   002    00000010        010    00011111        018    00000000
   003    00000000        011    00000000        019    10000000
  _004    01000000        012    00010011        020    00000000
   005    00010001        013    10000011        021    00000000
   006    00000000        014    00011111        022    00000000
   007    00000000        015    10000000        023    00000000
   008    00011111        016    00000000        024    00000000
*** RDRN DECI *** PROD *** *** SCW
bit5:1/0:DEC signal is low/high level
NO.  004
MDI                                               S0000 T00 H00
```

Key in "1" to finish the alteration

```
BIT PARAMETER                                          O0000 N00000
   NO.        DATA       NO.        DATA       NO.        DATA
   001    00000000       009    00011111       017    00101000
   002    00000010       010    00011111       018    00000000
   003    00000000       011    00000000       019    10000000
  _004    01100000       012    00010011       020    00000000
   005    00010001       013    10000011       021    00000000
   006    00000000       014    00011111       022    00000000
   007    00000000       015    10000000       023    00000000
   008    00011111       016    00000000       024    00000000

*** RDRN DECI *** PROD *** *** SCW
bit5:1/0:DEC signal is low/high level
NO.  004
MDI                                                   S0000 T00 H00
```

B Alteration of the data parameter, pitch data

1 Data parameter alteration

1） Turn on the parameter switch;

2） Enter the MDI mode

3） Move the cursor to the No. of the parameter to be set

4） Key in the new parameter value

5） Press 

key, the value is entered and displayed

6） After all parameters setting is finished, the PARM SWT needs to be set to OFF for security

Example 1: Set the data parameter №059 to 4000.

Move the cursor to “№059” by the steps above, key in “4000” by sequence in the prompt line, the figure is as follows:

```
DATA PARAMETER                                         O0000 N00000
   NO.        DATA       NO.        DATA       NO.        DATA
   049           1       057           1       065         100
   050           1       058           1       066         100
   051           1      _059        7600       067         100
   052           1       060        7600       068         100
   053           1       061        7600       069         400
   054           1       062        7600       070        8000
   055           1       063        7600       071          50
   056           1       064         100       072         100

Max.speed of rapid locating in X(mm/min)
NO.  059    4000_
MDI                                                   S0000 T00 H00
```

Press 

key to finish the alteration. The page is as follows

```
DATA PARAMETER                                          O0000 N00000
   NO.         DATA       NO.         DATA       NO.         DATA
   049            1       057            1       065          100
   050            1       058            1       066          100
   051            1      _059         4000       067          100
   052            1       060         7600       068          100
   053            1       061         7600       069          400
   054            1       062         7600       070         8000
   055            1       063         7600       071           50
   056            1       064          100       072          100

Max.speed of rapid locating in X(mm/min)
NO.  059
MDI                                                 S0000 T00 H00
```

Example 2: Set the X axis value of the pitch data No.000 to 12, set the value of Z axis to 30

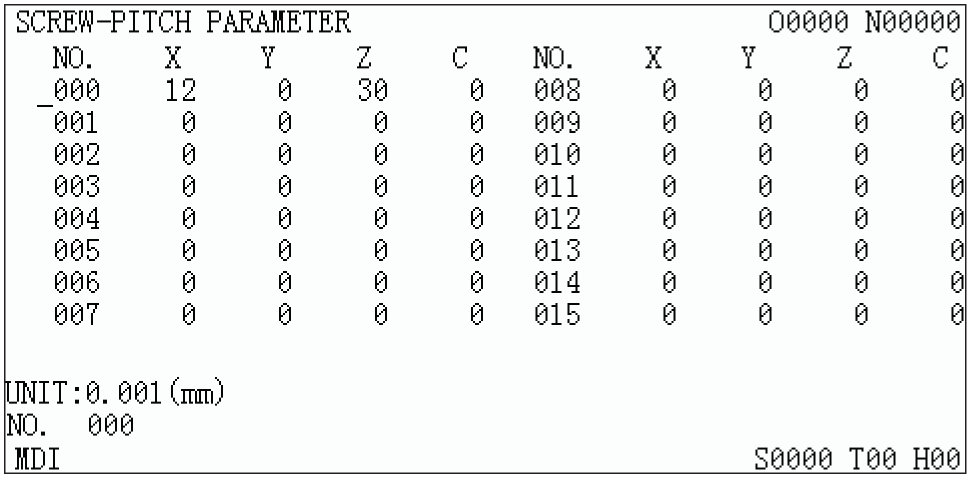
Move the cursor to pitch data No.000 by the steps above, key in “X12” by sequence in the cue line, the figure is as follows:

```
SCREW-PITCH PARAMETER                                   O0000 N00000
   NO.     X      Y      Z      C     NO.     X      Y      Z      C
  _000     0      0      0      0     008     0      0      0      0
   001     0      0      0      0     009     0      0      0      0
   002     0      0      0      0     010     0      0      0      0
   003     0      0      0      0     011     0      0      0      0
   004     0      0      0      0     012     0      0      0      0
   005     0      0      0      0     013     0      0      0      0
   006     0      0      0      0     014     0      0      0      0
   007     0      0      0      0     015     0      0      0      0

UNIT:0.001(mm)
NO.  000  X 12_
MDI                                                 S0000 T00 H00
```

Pres DATA INPUT key to finish the alteration. The page is as follows:

```
SCREW-PITCH PARAMETER                                   O0000 N00000
   NO.     X      Y      Z      C     NO.     X      Y      Z      C
  _000    12      0      0      0     008     0      0      0      0
   001     0      0      0      0     009     0      0      0      0
   002     0      0      0      0     010     0      0      0      0
   003     0      0      0      0     011     0      0      0      0
   004     0      0      0      0     012     0      0      0      0
   005     0      0      0      0     013     0      0      0      0
   006     0      0      0      0     014     0      0      0      0
   007     0      0      0      0     015     0      0      0      0

UNIT:0.001(mm)
NO.  000
MDI                                                 S0000 T00 H00
```

The same as above, key in “Z30”by sequence in the prompt line, press DATA INPUT key to finish the alteration. The page is as follows:



SCREW-PITCH PARAMETER O0000 N00000

NO.	X	Y	Z	C	NO.	X	Y	Z	C
000	12	0	30	0	008	0	0	0	0
001	0	0	0	0	009	0	0	0	0
002	0	0	0	0	010	0	0	0	0
003	0	0	0	0	011	0	0	0	0
004	0	0	0	0	012	0	0	0	0
005	0	0	0	0	013	0	0	0	0
006	0	0	0	0	014	0	0	0	0
007	0	0	0	0	015	0	0	0	0

UNIT:0.001(mm)
NO. 000
MDI S0000 T00 H00

9.2 The Password Setting and Alteration

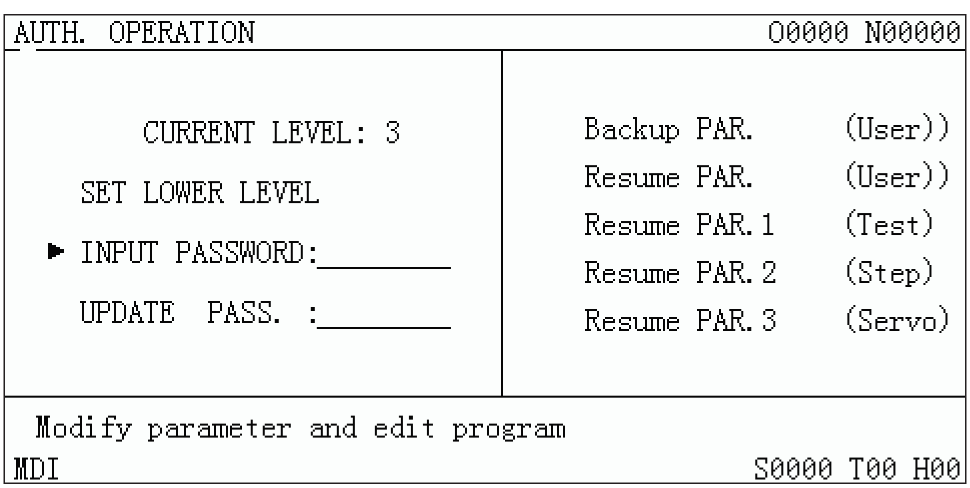
To prevent the part programs, CNC parameters from malignant alteration, this GSK980MD provides an authority setting function that is graded for 4 levels. By decending sequence, they are machine builder (2nd) level, equipment management (3rd) level, technician (4th) level, machining operation (5th) level

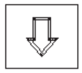
The 2nd level: Modification of the CNC bit parameter, data parameter, pitch data, tool offset data, part program edit, PLC ladder transmission etc. are allowed

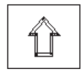
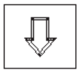
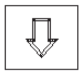
The 3rd level: initial password 2345, the CNC bit parameter, data parameter, tool offset data, part program edit operations are allowed;

The 4th level: initial password 1234, tool offset data (for tool setting), macro variables, part program edit operations are allowed; but the CNC bit parameter, data parameter, pitch data operations are unallowed.

The 5th level: no password. Only the machine panel operation is allowed, and the operations of part program edit and selection, the alteration operations of CNC bit parameter, data parameter, pitch data, tool offset data are unallowed



After entering the authority setting page, the cursor locates at the “INPUT PASSWORD:”line. It may press the  or  key to move the cursor to the corresponding item.

- Press  key once, the cursor shifts a line upward. If the current cursor locates at the “SET LOWER LEVEL”line (1st line) , press  key, the cursor shifts to the “UPDATE PASS:”line (end line)
- Press  key once, the cursor shifts a line upward. If the current cursor locates at the end line, by pressing  key once, the cursor moves to the 1st line.

9.2.1 Entry of the operation level

1 After entering the PASSWORD SETTING page, move the cursor to the “INPUT PASSWORD:”line;

2 Key in the password (an “*”sign added each time inputting a character)

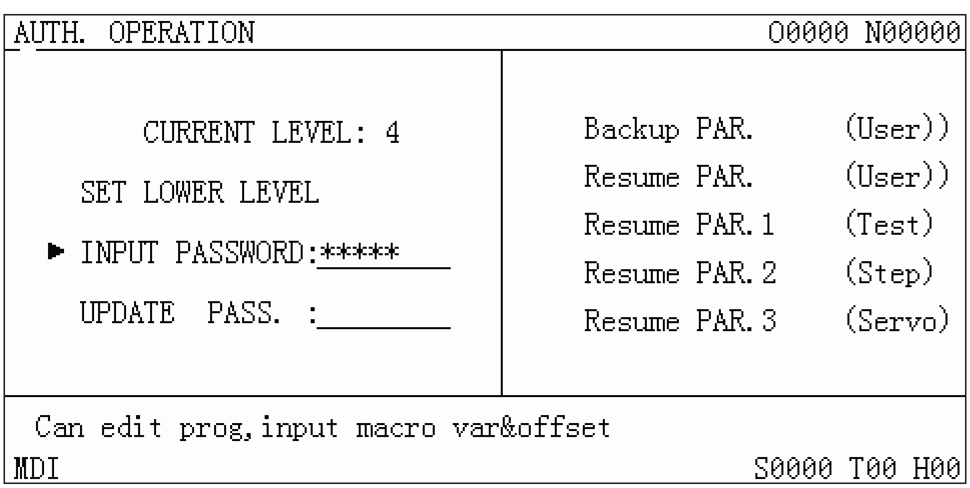
3 Press 

key to finish the inputting, and it will enter the corresponding password level.

Note The length of this GSK980MD system password corresponds to the operation level, which can’t be added or decreased by user at will.

Operation level	Password length	Initial password
3rd	5 bits	12345
4th	4 bits	1234
5th	No	No

Example: The current CNC level is the 4th level, as the following page shows. The 3rd level password of CNC is 12345, please alter the current level to the 3rd level.



Move the cursor to the "INPUT PASSWORD:"line, key in 12345, then press the 

key, the CNC prompts "Modify parameter and edit program", "Password passed", and the current level is the 3rd level. The page is as follows:

```
AUTH. OPERATION                                    O0000 N00000

     CURRENT LEVEL: 3             Backup PAR.       (User))
   SET LOWER LEVEL                Resume PAR.       (User))
 ► INPUT PASSWORD:________        Resume PAR.1      (Test)
   UPDATE  PASS.  :________       Resume PAR.2      (Step)
                                  Resume PAR.3      (Servo)

 Modify parameter and edit program
MDI                              IMAGE STORED    S0000 T00 H00
```

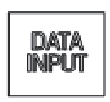
Note: When current operation authority is lower than or equal to the 3rd level (3rd, 4th, 5th level), the password level is not changed if repower the CNC system. If previous level is higher than the 3rd level (0, 1st, or 2nd level), it defaults the 3rd level.

9.2.2 Alteration of the password

Steps for password alteration:

1 After entering the PASSWORD SETTING page, enter the password by the methods in Section10.3.2;

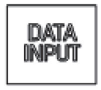
2 Move the cursor to the"ALTER PASSWORD:"line;

3 Key in the new password, and press 

key

4 The CNC system prompts "PLEASE INPUT USER PASSWORD AGAIN", the page is as follows:

```
AUTH. OPERATION                                    O0000 N00000

     CURRENT LEVEL: 3             Backup PAR.       (User))
   SET LOWER LEVEL                Resume PAR.       (User))
 ► INPUT PASSWORD:________        Resume PAR.1      (Test)
   UPDATE  PASS.  :________       Resume PAR.2      (Step)
                                  Resume PAR.3      (Servo)

 Modify parameter and edit program
MDI                              IMAGE STORED    S0000 T00 H00
```

5 After reinputting the password, press 

key, if the two passwords input are identical, CNC prompts “PASSWORD UPDATED”. So the password alteration is successful.

```
AUTH. OPERATION                                  O0000 N00000
       CURRENT LEVEL: 3          Backup PAR.     (User))
    SET LOWER LEVEL              Resume PAR.     (User))
    INPUT PASSWORD:_______       Resume PAR.1    (Test)
  ► UPDATE  PASS. :_______       Resume PAR.2    (Step)
 PASSWORD UPDATED.               Resume PAR.3    (Servo)
 Modify parameter and edit program
MDI                                          S0000 T00 H00
```

6 If the two passwords input are not identical, CNC prompts “PASSWORD CHECKOUT ERROR.”, the page is as follows:

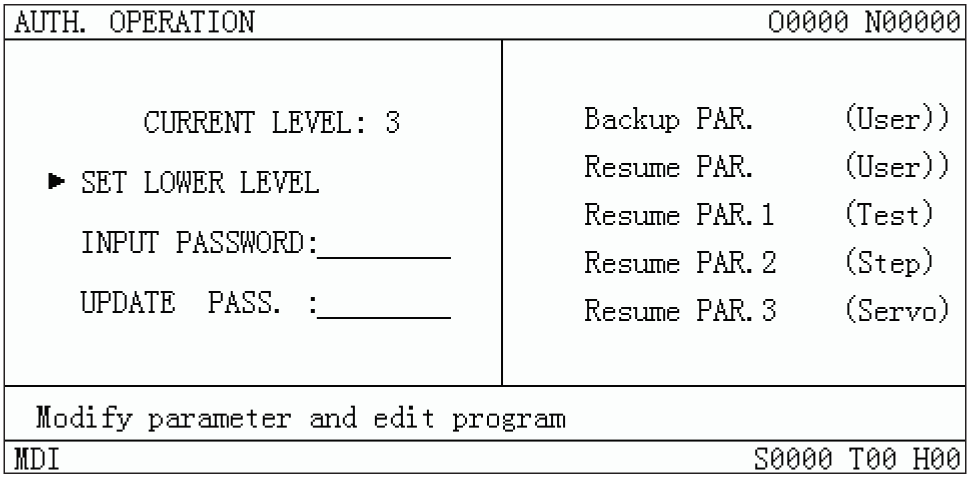
```
AUTH. OPERATION                                  O0000 N00000
       CURRENT LEVEL: 3          Backup PAR.     (User))
    SET LOWER LEVEL              Resume PAR.     (User))
    INPUT PASSWORD:_______       Resume PAR.1    (Test)
  ► UPDATE  PASS. :_______       Resume PAR.2    (Step)
 PASSWORD CHECKOUT ERROR.        Resume PAR.3    (Servo)
 Modify parameter and edit program
MDI                                          S0000 T00 H00
```

9.2.3 Lower level set

The demotion of the operation level is used to enter a lower level from a higher level, the steps are as follows:

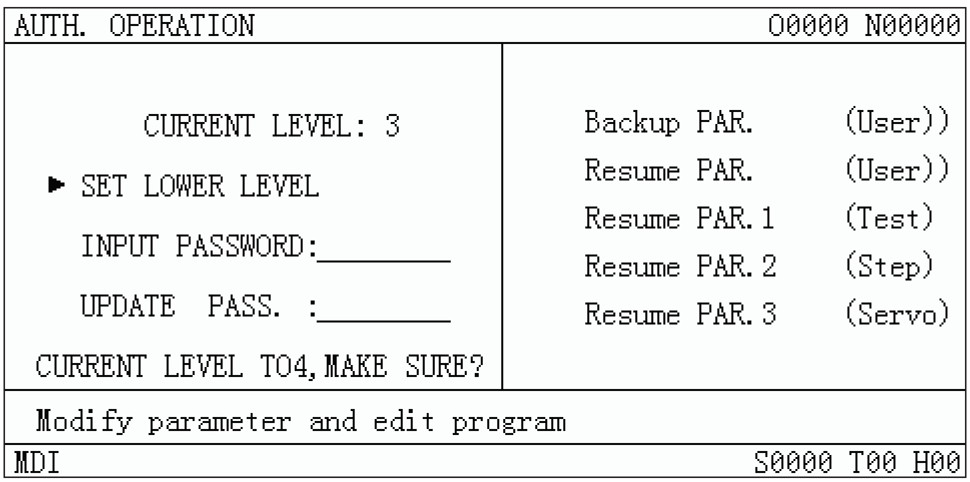
1 After entering the PASSWORD SETTING page, key in the password by the method in Section 10.3.2

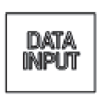
2 Move the cursor to the“SET LOWER LEVEL”line, if the current CNC operation is the 3rd level, the page is as follows:



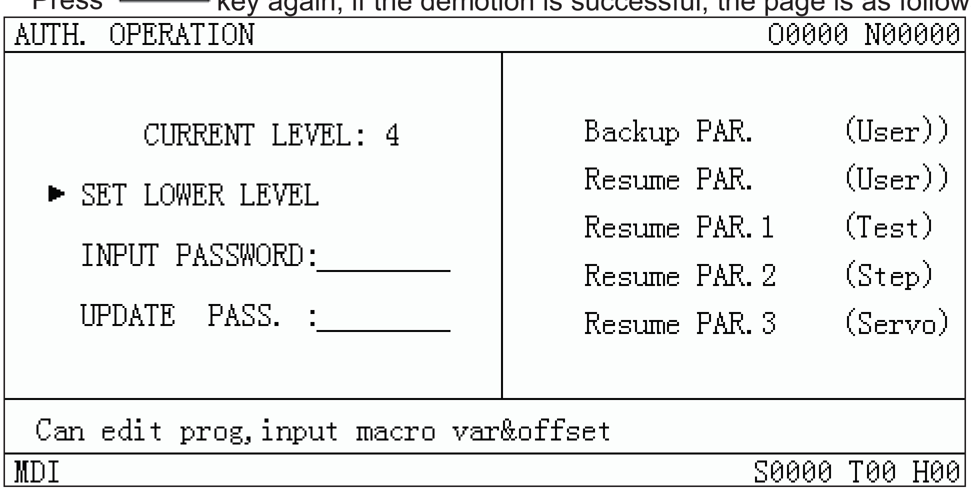
3 Press 

key, the CNC prompts “CURRENT LEVEL TO 4, OK？ ”; the page is as follows:



4 Press 

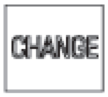
key again, if the demotion is successful, the page is as follows:



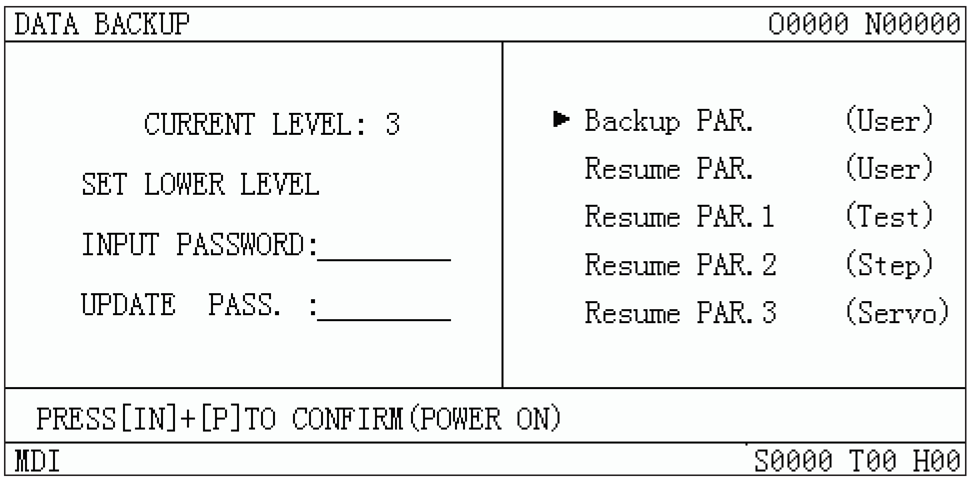
Note If the current level is the 5th level, the demotion operation is unallowed.

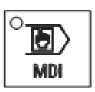
9.3 Data Restore and Backup

The user data (such as bit parameter and pitch data) can be backup (saved) and restored (read) in this GSK980MD system. It doesn't affect the part programs stored in the CNC system while backuping and restoring these data. The backup page is as follows:

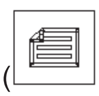
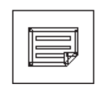
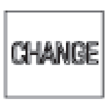
Press 

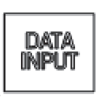
key repeatedly, "PASSWORD SETTING" and "DATA BACKUP" pages can be switched.



- Turn on the parameter switch
- Press 

key to enter the MDI mode, then press 

key ( or  key if necessary) to enter PASSWORD SETTING page;
- Press 

, and switch to the Data Backup page.
- Move the cursor to the desired item;
- Press 

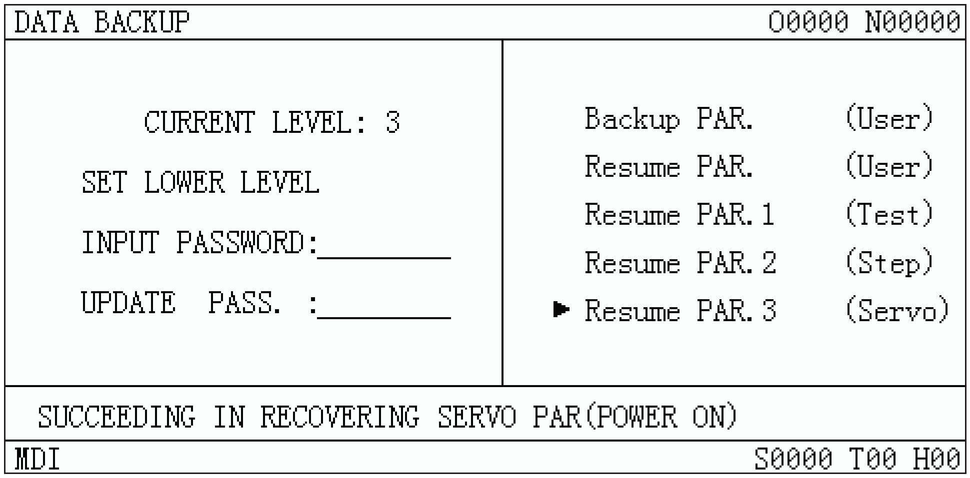
. 

keys together.

Note Don't cut off the power in the backup and restore operation of the data, and no other operation is suggested to be performed before the aforesaid operation is prompted to be finished.

Example: to restore the CNC parameter to 1μ level servo standard parameter, the steps are as follows:

Turn on the parameter switch, and enter the Backup PAR. page of MDI mode, move the cursor to "Recover Default PAR. (1μ level)", as the following figure shows:

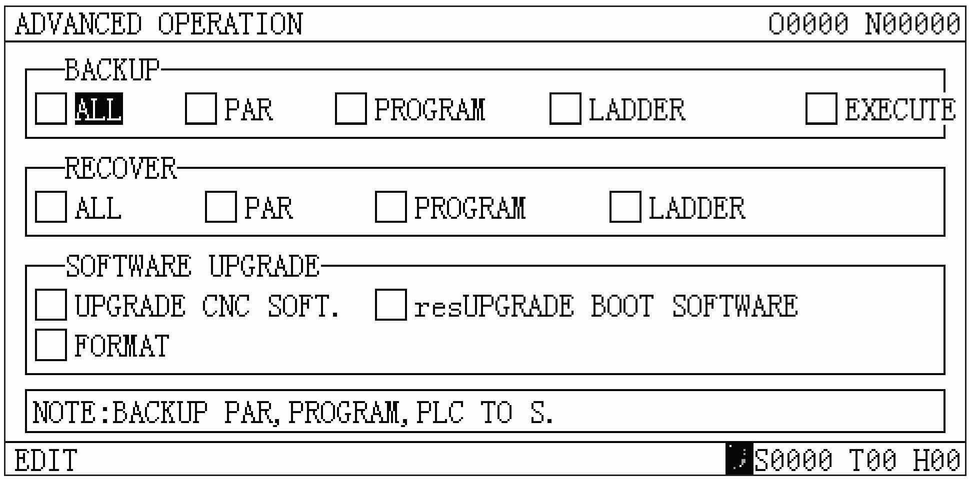


Press 

keys together, the CNC system prompts “SERVO PAR BACKUP RECOVERED (POWER ON)”.

CHAPTER 10 ADVANCE OPERATION

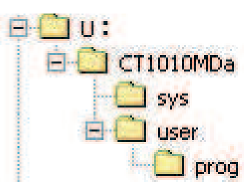
Advance operation interface of GSK980MDa, which is as follows, is started by connecting CNC to USB. In this interface, communication between CNC & USB and system update operations can be done. Its transmission speed is much faster than traditional serial communication speed, greatly increases the efficiency of file transmission. More over, USB is easy to carry, to use and it supports hot plugging, plug and play at once.



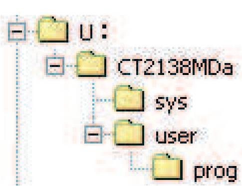
10.1 Operation path

USB operation in 980MDa is searching and setting up destination list on U disk with its number. Therefore, the system with different number is corresponding to different U disk list in advance operation.

Example: If the number of system A is CT1010MDa, the list of advance operation on U disk is as follows:



If the number of system B is CT2138MDa, the list of advance operation on U disk is as follows:



If the system has no number, the list of advance operation on U disk is as follows:

Note: The number of the system can be found in version information page of diagnosis. The following contents are described by list of gsk980mda_backup.

- **Path explanations**

Path file folder		Explanation
user\		Target position for parameter and PLC file backup and restore
	prog\	Target position for part program file backup and restore

- **File specification**

	File name	Expended name	Remark
Parameter file	Para1, Para2, Para3	.par	Case sensitive
Part program	O0000 ~ O9999	.CNC	Case sensitive
PLC file	plc ~ plc7	.ldx	Case sensitive

- **Operation authority**

Backup operation	Parameter	Authority level 3 (including level 3)
	Part program	Authority level 3 (including level 3)
	Ladder diagram	Authority level 3 (including level 3)
Restore operation	Parameter	Authority level 3 (including level 3)
	Part program	Authority level 3 (including level 3)
	Ladder diagram	Authority level 2 (including level 2)

Note: Level 2 or above authority is needed for part program operation above number 9000.

10.2 Operation instructions

➢ **Key descriptions**

Cursor moving：Press direction keys ⇧ ⇩ ⇨ ⇦ CHANGE to move the cursor.

Menu selection: Press DATA INPUT key to select the operation item which cursor is in.

Menu cancellation: Press CANCEL key to cancel the operation item which cursor is in.

Operation execution：Press DATA OUTPUT key to execute all operation items selected in current column.

Operation confirmation：Execution needs to be confirmed, please press Y & key to confirm or press N key to cancel the execution.

➢ **Parameter restore and backup**

Backup the parameter: Copy all parameter states and values to U:\gsk980MDa_backup\user\ of USB memory unit in the form of file Para1.par，Para2.par，Para3.par. If the above-mentioned file does not exist, set up a new one: If the file exists, this file will be overwritten by the new one.

Restore the parameter: Copy parameter files from USB memory unit U:\gsk980MDa_backup\user\ back to the CNC system to restore the system parameter. Restore operation cannot be done if the above-mentioned path is moved or altered or irregular file name is renamed.

Note: Repower the CNC system after parameter load is successful.

➢ **Part program restore and backup**

Backup the part parameter: Copy all part programs of current system to U:\gsk980MDa_backup\user\prog\ of USB memory unit in the form of file .CNC. If the above-mentioned file does not exist, set up a new one: If the file exists, this file will be overwritten by the new one.

Restore the part program: Copy all part programs from USB memory unit U:\gsk980MDa_backup\user\prog\ back to the CNC system to restore the part program. Restore operation cannot be done if the above-mentioned path is moved or altered or irregular file name is renamed.

- **Ladder diagram (PLC) restore and backup**

The ladder diagram backup: Copy all ladder diagrams (.ldx file) of the current system to U:\gsk980MDa_backup\user\ of USB memory unit. If the above-mentioned file does not exist, set up a new one: If the file exists, this file will be overwritten by the new one.

Restore the ladder diagram: Copy parameter files from USB memory unit U:\gsk980MDa_backup\user\ back to the CNC system to restore the ladder diagram. Restore operation cannot be done if the above-mentioned path is moved or altered or irregular file name is renamed.

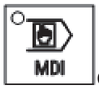
Note: Repower the CNC system after the ladder diagram restore is successful.

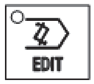
10.3 Attentions

- **Notice：**If a file or list on target path has the same name as the one will be copied, it will be overwritten and replaced by the system automatically. Therefore, to prevent the file or list from overwriting or replacing, please copy and save it separately.
- It forbids doing any other operation in advance operation. Once operation is performed, it can not be interrupted until it is finished.
- If the file to be saved or restored is large, operation time will be long. Please wait.
- Pull out USB if abnormal conditions occur, then connect it again.

CHAPTER 11 FLASH OPERATION

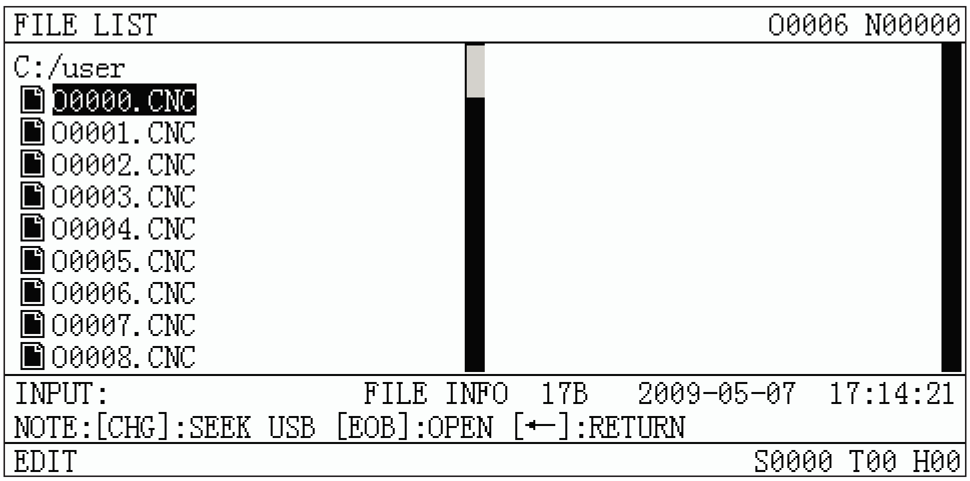
11.1. File list

Press 

or 

key to select［MDI］or [EDIT] mode, press 

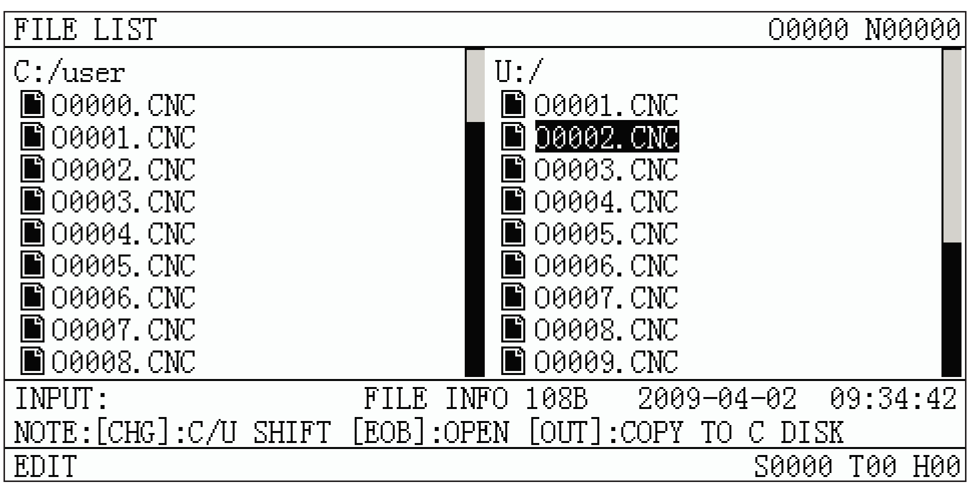
key to enter［file list interface, the page is as follows：



In edit or MDI mode, press 

key to identify U disk.

If identification is unsuccessful, it prompts: "Fail to connect U disk". If identification is successful, the following file list will be displayed.



Special explanation:

The list information of disk CNC is displayed at the page left and list information of disk USB is displayed at the page right. The display column will not display any information if U disk is not detected. Character entry box, file attributes information and user operation prompts are displayed at the bottom of the page.

1. Current list page only display the list information of the currently opened folder.
2. U disk can be identified in edit or MDI mode.

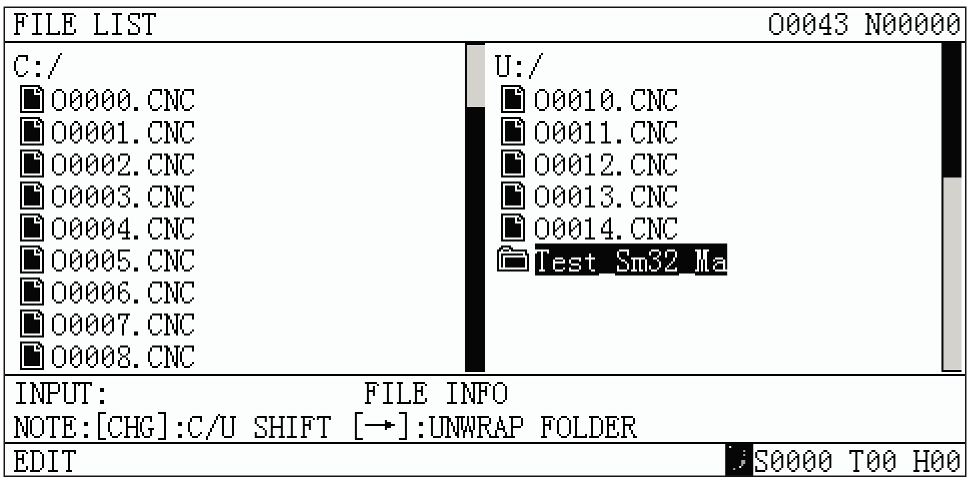
3. It not support Chinese complex characters.
4. It not support Chinese long file name, only the first three characters .+“～1”of this file name can be displayed.
5. Non-CNC file of C disk and U disk is displayed.

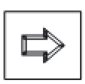
Note: The file name，which consists of “O”+“4 digits”+“.CNC ”, is considered to be CNC format file.

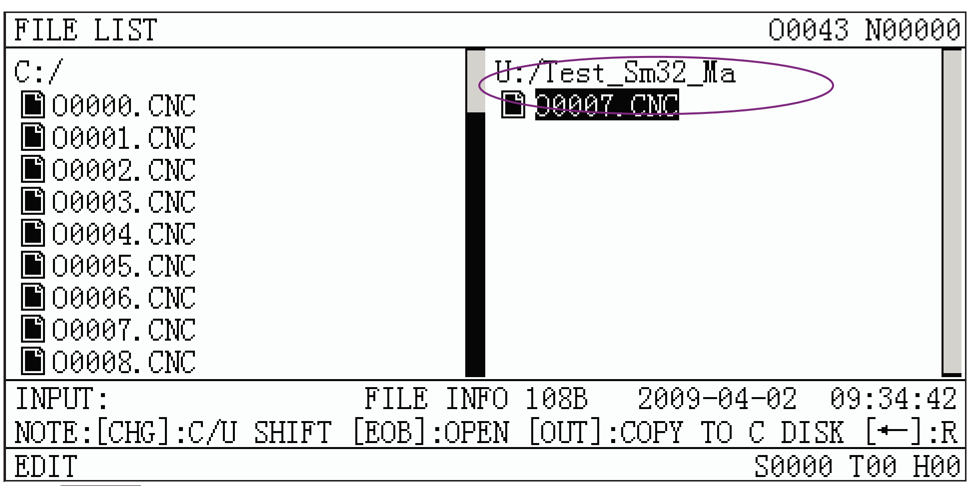
11.2. Introduction of general file operation function

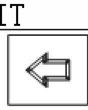
11.2.1 Open and close file folder

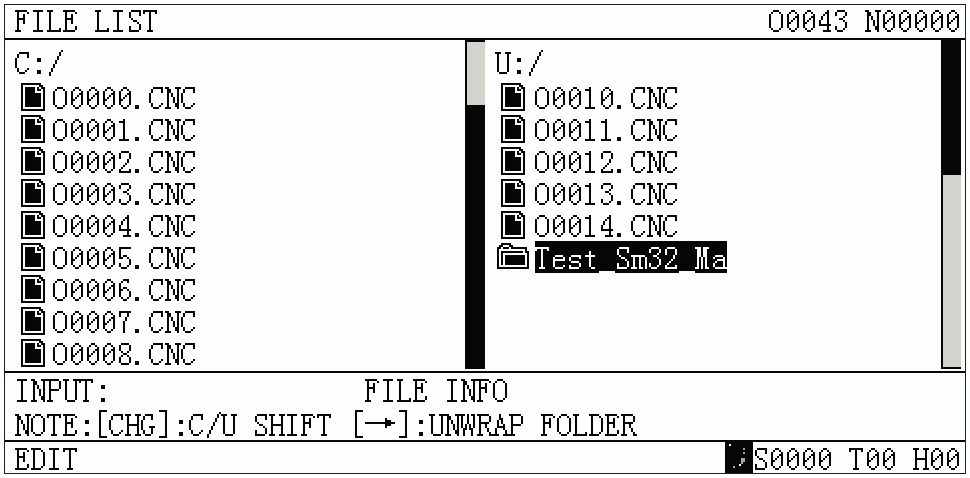
Move the cursor to the folder will be opened.



Press  key to open the folder. The list which the file locates is displayed in the first line (long list is scrolling display)



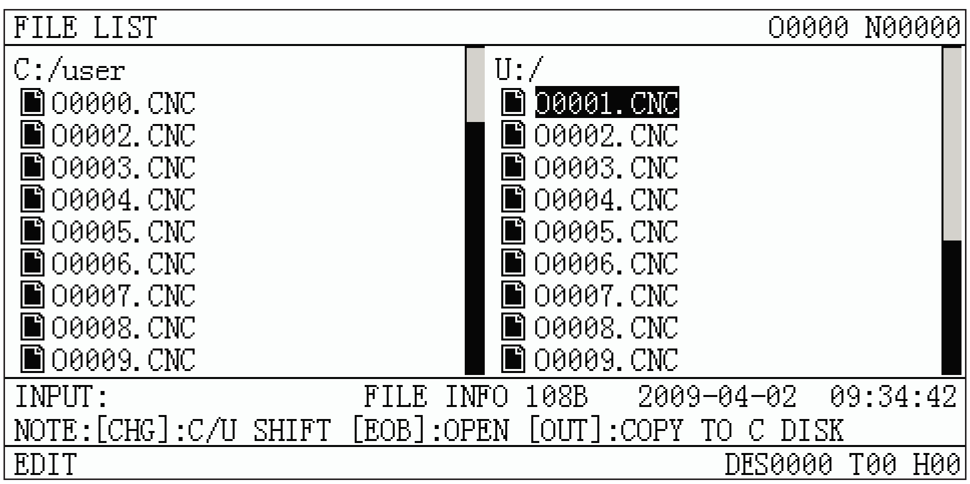
Press  key to close the folder and return to the next higher level of the list.



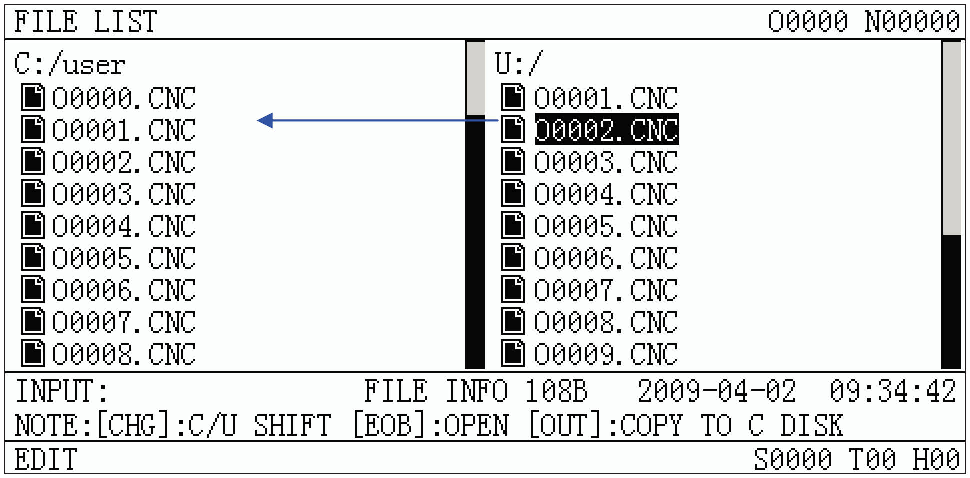
11.2.2 Copy the file by one key(current list in C disk←→current list in U disk)

In “edit”mode, select the CNC format file, press  key to copy it. See the following figure：

① Select CNC file, press ；

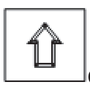
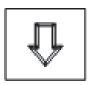


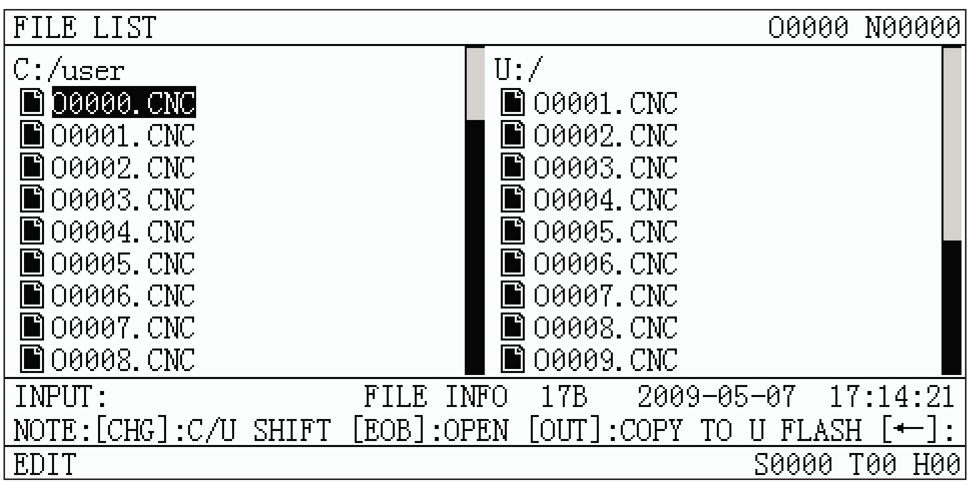
② After duplication is successful, the cursor moves to the next file in current list. The list on the other side is refreshed at once.



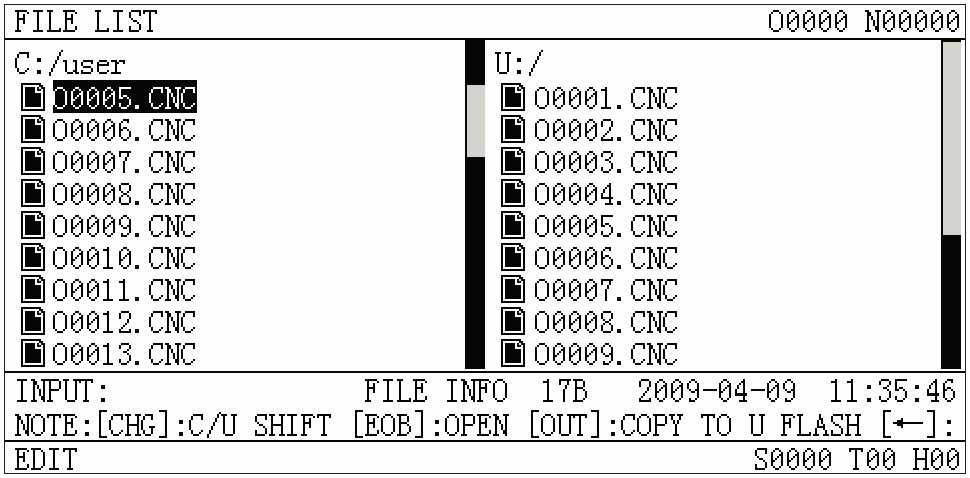
Special explanation：Duplication can not be done under 5-level authority.

11.2.3 CNC file search

In “EDIT”and“AUTO”mode, input target program number in input column, and press or  to search this program.

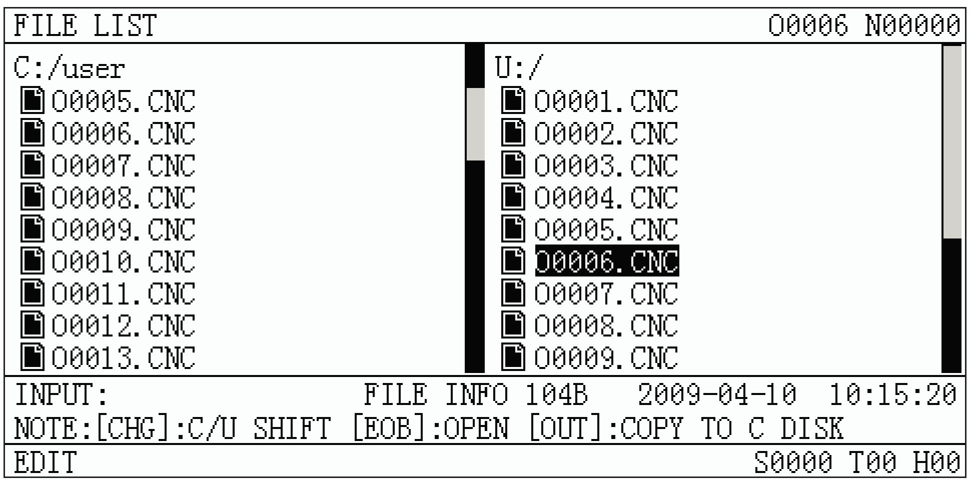


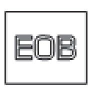
If program search is successful after input “O5”, the cursor moves to target program. If this program can not be searched, “the file dose not exist” will be prompted at message column.

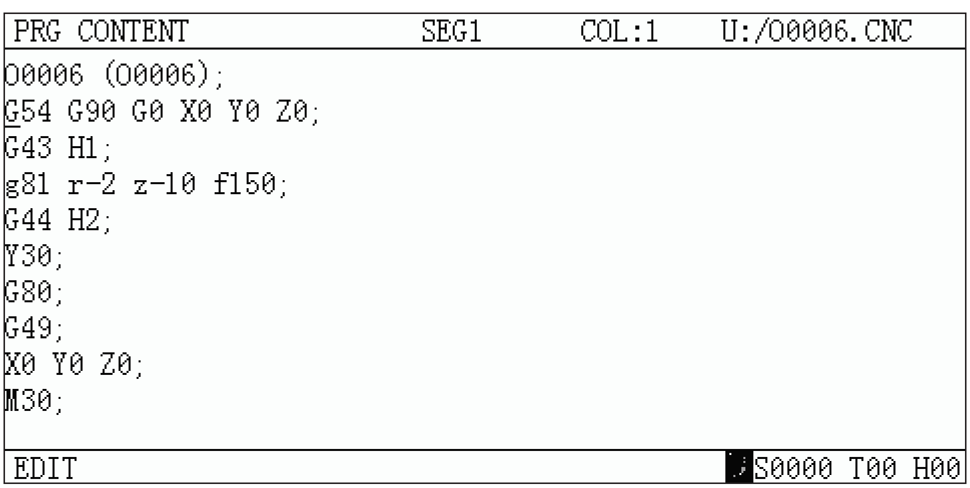


11.2.4 Open CNC file

1. In“EDIT”and“AUTO”mode, select the CNC format file when there is no program execution.



2. Press  key to open the file. Current page is switched to［program content］page.



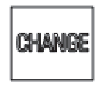
Special explanations:

1. The program above number 9000 can not be opened with authority level 3 or under

level 3.

2. The program file can not be opened with authority level 5.

Attentions:

1. In “program content”, it is not allowed to do any operation on U disk. These operations are: setting-up, duplication, rename, deletion, editing, save, etc.. Process and check operations can be done for programs on U disk in page“program content”.
2. The called subprogram in auto-run should in a same level of list with main program.
3. Pull out U disk when it is open, system alarm occurs“U disk is not connected”. At this time, plug in U disk again, press 

key to detect U disk in MDI mode, or press 

+ 

keys to clear the alarm.

VOLUME Ⅲ INSTALLATION

CHAPTER 1 INSTALLATION LAYOUT

1.1 GSK980MDa Connection

1.1.1 GSK980MDa back cover interface layout

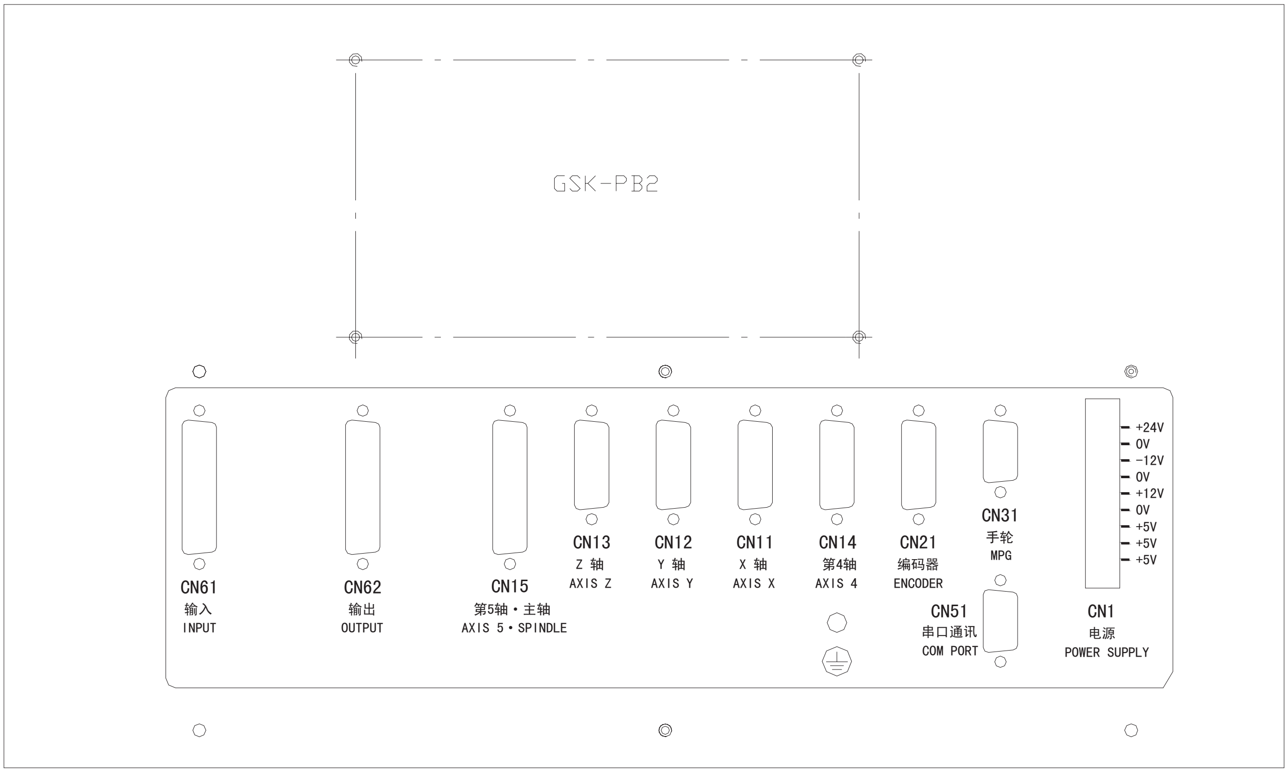


Fig 1-1 GSK980MDa back cover interface layout

1.1.2 Interface explanation

- Power box: GSK-PB2,for +5V, +24V, +12V, -12V, GND power supply
- CN11: X axis, 15-core DB female socket,for connecting X axis drive unit
- CN12: Y axis, 15-core DB female socket,for connecting Y axis drive unit
- CN13: Z axis, 15-core DB female socket,for connecting Z axis drive unit
- CN14: 4th axis，15-core DB female soket,for connecting 4th axis drive unit
- CN21: coder, 15-core DB female socket,for connecting Encoderd
- CN51: inverter, 9-core DB male socket,for connecting pc RS232 interface
- CN15: 5th axis&spindle port, 25-core DB male socket,for connecting inverter & 5th axis
- CN31: handwheel, 26-core 3 line famele socket,for connecting handwheel;
- CN62: ouput，44-core 3 lines famele socket，for sending the signal of CNC to machine
- CN61:input, 44-core 3 line male socket，for sending the signal of machine to CNC

1.2 GSK980MDa Installation

1.2.1 GSK980MDa external dimensions

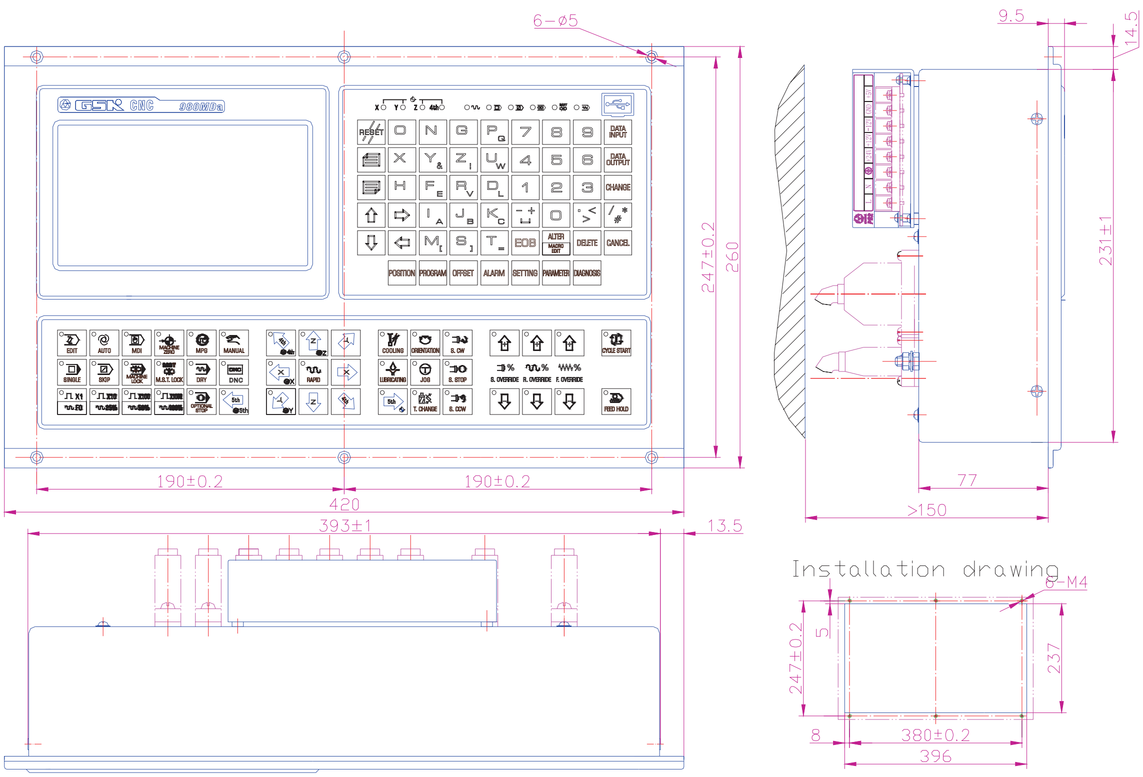


Fig. 1-2 GSK980MDa external dimensions

1.2.2 Installation conditions of the cabinet

- The dust, cooling liquid and organic resolution should be effectively prevented from entering the cabinet;
- The designed distance between the CNC back cover and the cabinet should be not less than 20cm, the inside and outside temperature difference of the cabinet should be no les than 10℃ temperature rises when the cabinet inside temperature rises;
- Fans should be fixed in the cabinet to ventilate it;
- The panel should be installed in a place where the coolant can't splash;
- The external electrical interference should be taken into consideration in cabinet design to prevent it from transferring to CNC system.

1.2.3 Protection methods against interference

In order to ensure the CNC stable working, the anti-interference technology such as space electromagnetic radiation shielding, impact current absorbing, power mixed wave filtering are employed in CNC design.And the following measures are necessary during CNC connection:

1. Make CNC far from the interference devices (inverter, AC contactor, static generator, high-pressure generator and powered sectional devices etc.);

2. To supply the CNC via an isolation transformer , the machine with the CNC

should be grounded, the CNC and drive unit should be connected with independent grounding wires at the grounding point;

3. To supress interference: connect parallel RC circuit at both ends of AC coil (Fig. 1-4), RC circuit should approach to inductive loading as close as possible; reversely connect parallel freewheeling diode at both ends of DC coil (Fig. 1-5); connect parallel surge absorber at the ends of AC motor coil (Fig. 1-6);

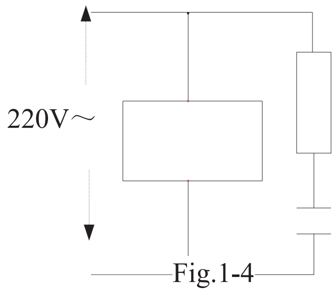


Fig.1-4

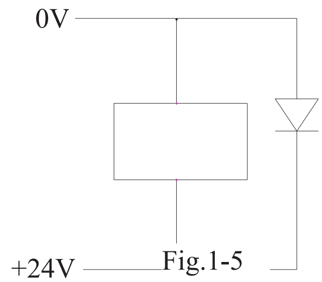


Fig.1-5

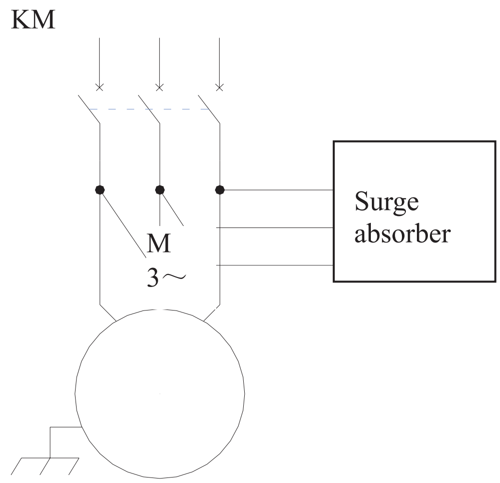


Fig.1-6

4. To employ with twisted shield cable or shield cable for the leadout cable of CNC, the cable shield tier is grounded by single end at CNC side, signal cable should be as short as possible;

5. In order to decrease the mutual interference between CNC cables or CNC cables with strong-power cables,the wiring should comply to the following principles:

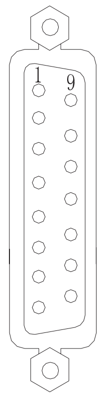
Group	Cable type	Wiring requirement
A	AC power line	Tie up A group cables with a clearance at least 10cm from that of B, C groups, or shield A group cables from electromagnetism
	AC coil	
	AC contactor	
B	DC coil（24VDC）	Tie up B and A group cables separately or shield B group cables; and the further B group cables are from that of C group, the better it is
	DC relay（24VDC）	
	Cables between CNC and strong-power cabinet	
	Cables between CNC and machine	
C	Cables between CNC and servo drive unit	Tie up C and A group cables separately, or shield C group cables; and the cable distance between C group and B group is at least 10cm with twisted pair cable applied.
	Position feedback cable	
	Position encoder cable	
	MPG cable	
	Other cables for shield	

CHAPTER 2 DEFINITION&CONNECTION OF INTERFACE SIGNALS

2.1 Connection to Drive unit

2.1.1 Drive interface definition

1：CPn+
2：DIRn+
3：PCn
4：+24V
5：ALMn
6：SETn
7：ENn
8：RDYn/ZSDn



9：CPn-
10：DIRn-
11：GND
12：VCC
13：VCC
14：GND
15：GND

Fig.2-1 CN11, CN12, CN13 interface（DB15 female）

Signal	Explanation
CPn+, CPn-	Command pulse signal
DIRn+, DIRn-	Command direction sigal
PCn	Zero signal
ALMn	Drive unit alarm signal
ENn	Axis enable signal
SETn	Pusle disable signal

2.1.2 Command pulse and direction signals

nCP+，nCP- are command pulse signals, nDIR+，nDIR- are command direction signals. These two group signals are both difference output（AM26LS31）, the interior circuit for them is shown in Fig. 2-2.

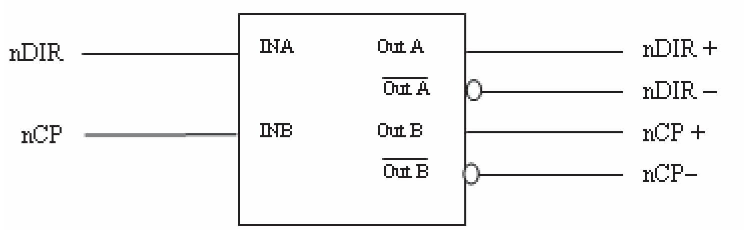


Fig. 2-2 Interior circuit of command pulse and direction signals

2.1.3 Drive unit alarm signal

The low or high level of the drive unit alarm level is set by the CNC bit parameter No.009 BIT0～BIT4，whose interior circuit is shown in Fig. 2-3:

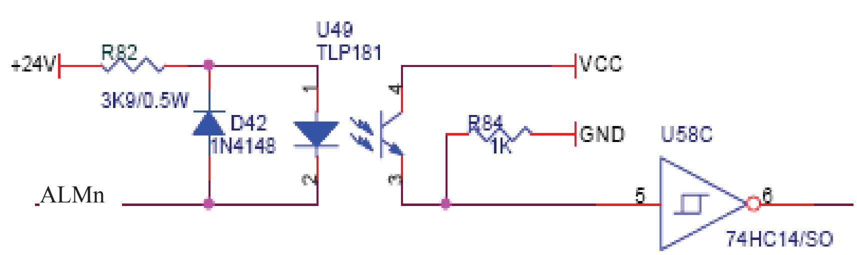


Fig.2-3 interior circuit of drive unit alarm signal

This input circuit requires that the drive unit transmits signal by the following types in Fig. 2-4:

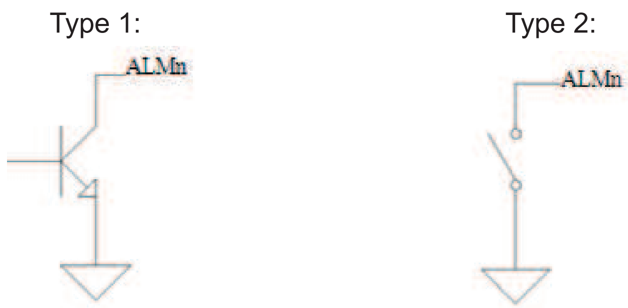


Fig.2-4 Signal types of drive unit

2.1.4 Axis enable signal ENn

nEN signal output is valid as CNC works normally (nEN signal to 0V); when the drive unit alarm or emergency alarm occurs, CNC cuts off nEN signal output (nEN signal to 0V off). The interior interface circuit is shown in Fig.2-5:

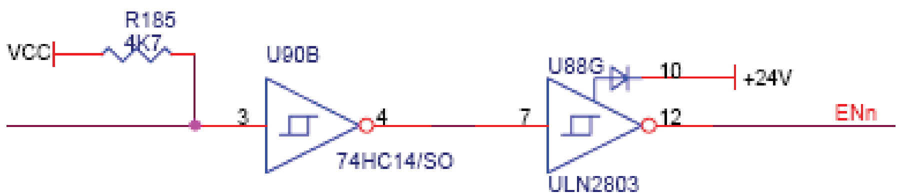


Fig.2-5 interior interface circuit for axis enable signal

2.1.5 Pulse disable signal SETn

nSET signal is used to control servo input disable which can enhance the anti-disturbance capability between CNC and drive unit. This signal is at low level if there is pulse output from CNC, high resistance if not. The interior interface circuit of it is shown in Fig. 2-6:

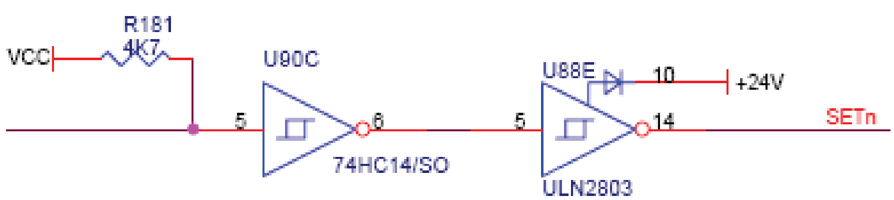


Fig.2-6 Interior interface circuit for pulse disable signal

2.1.6 Zero signal nPC

The one-rotation or approach switch signal is taken as zero signal for machine zero return. Its interior connection circuit is shown in Fig.2-7.

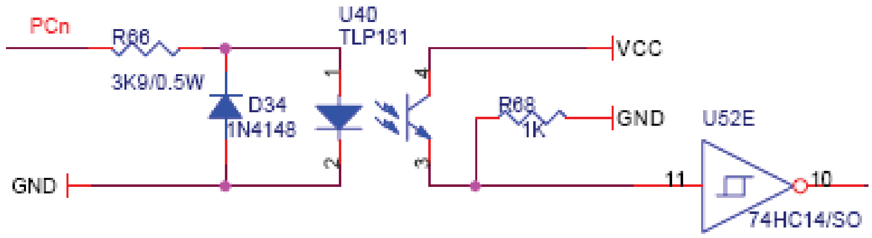


Fig.2-7 Zero signal circuit

Note: nPC signal uses +24V level.

a) The connection for NPN Hall elements taken as both deceleration signal and zero signal is shown in Fig. 2-8:

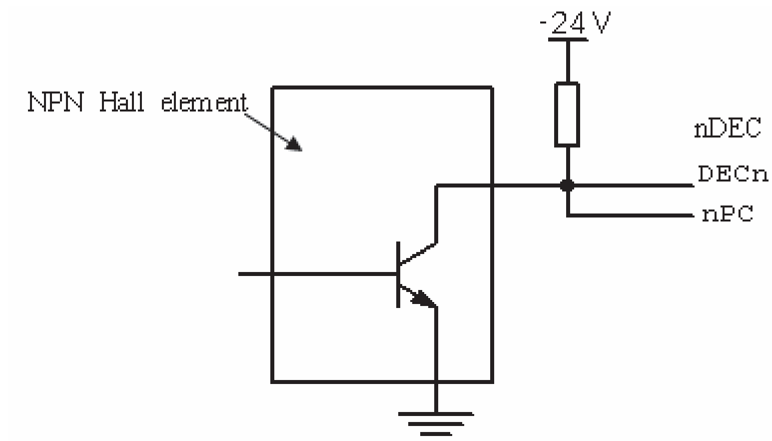


Fig. 2-8 Connection using NPN Hall elements

b) The connection for PNP Hall elements taken as both deceleration signal and zero signal is shown in Fig. 2-9:

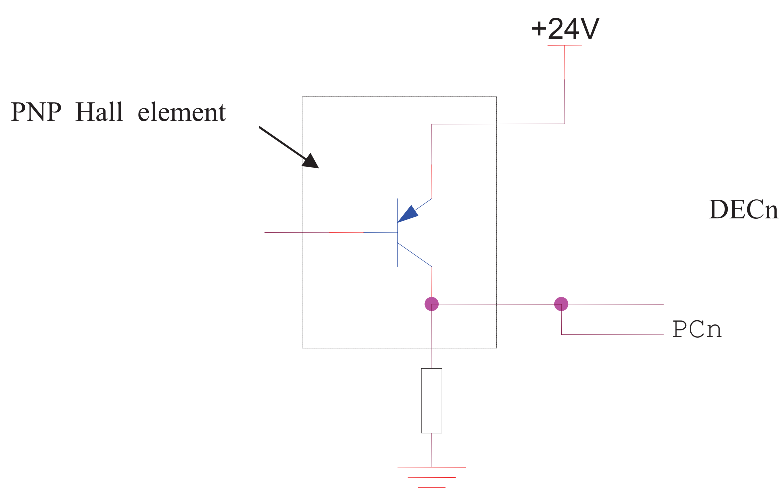


Fig 2-9 Connection using PNP Hall elements

2.1.7 Connection to drive unit

The connection of GSK 980MDa to GSK drive unit is shown in Fig. 2-10:

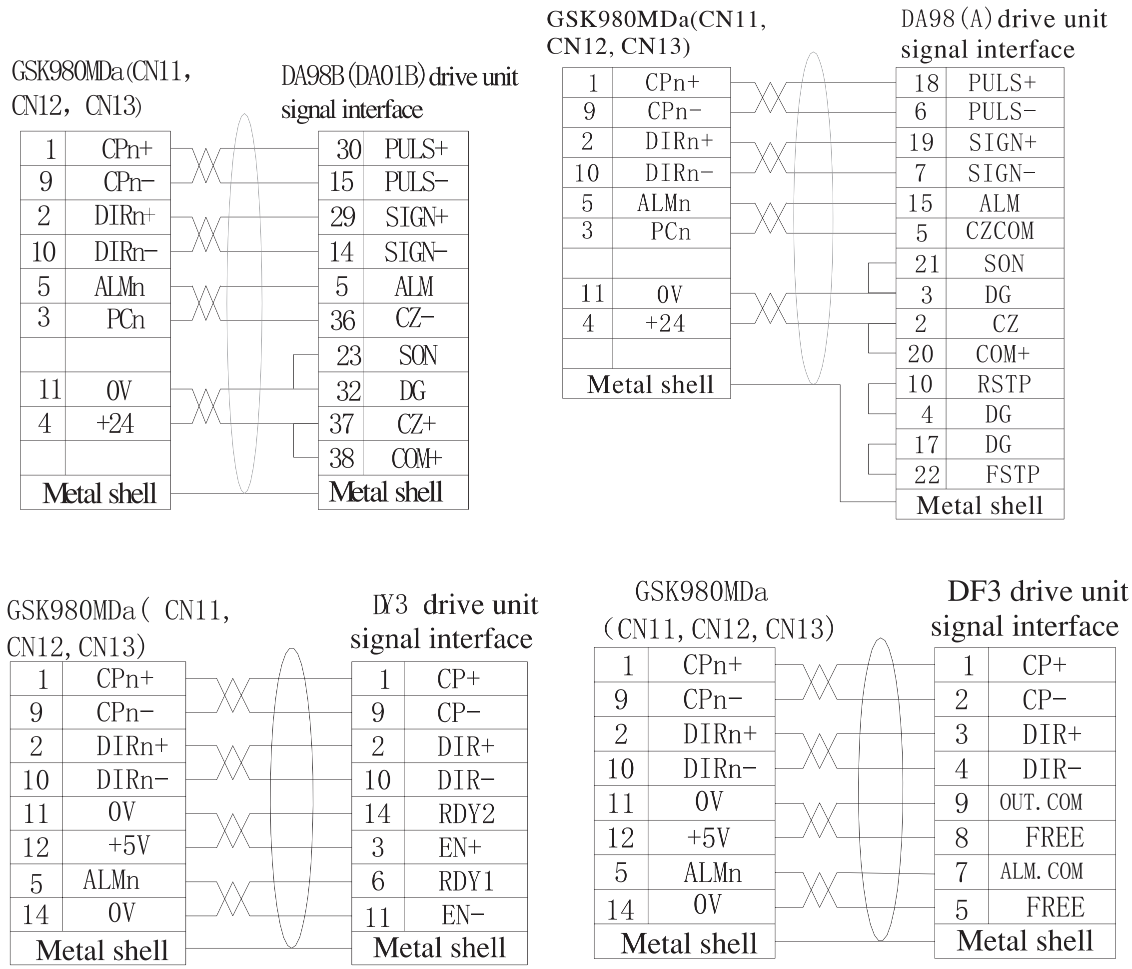
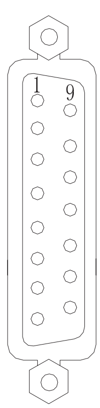


Fig.2-10 Connection of 4th axis interface to drive unit

2.2 Connection of 4th axis

2.2.1 4th axis interface definition

1：CP4+
2：DIR4+
3：PC4
4：+24V
5：ALM4
6：SET4
7：EN4
8：RDY4/ZSD4



9：CP4-
10：DIR4-
11：GND
12：VCC
13：VCC
14：GND
15：GND

Signal	Explanation
CP4+, CP4-	Command pulse signal
DIR4+, DIR4-	Command direction signal
PC4	Zero signal
ALM4	Drive alarm signal
EN4	Axis enable signal
SET4	Pulse disable signal

Fig.2-11 Interface CN14（DB15 female）

2.2.2 Connection of 4th axis interface as linear axis

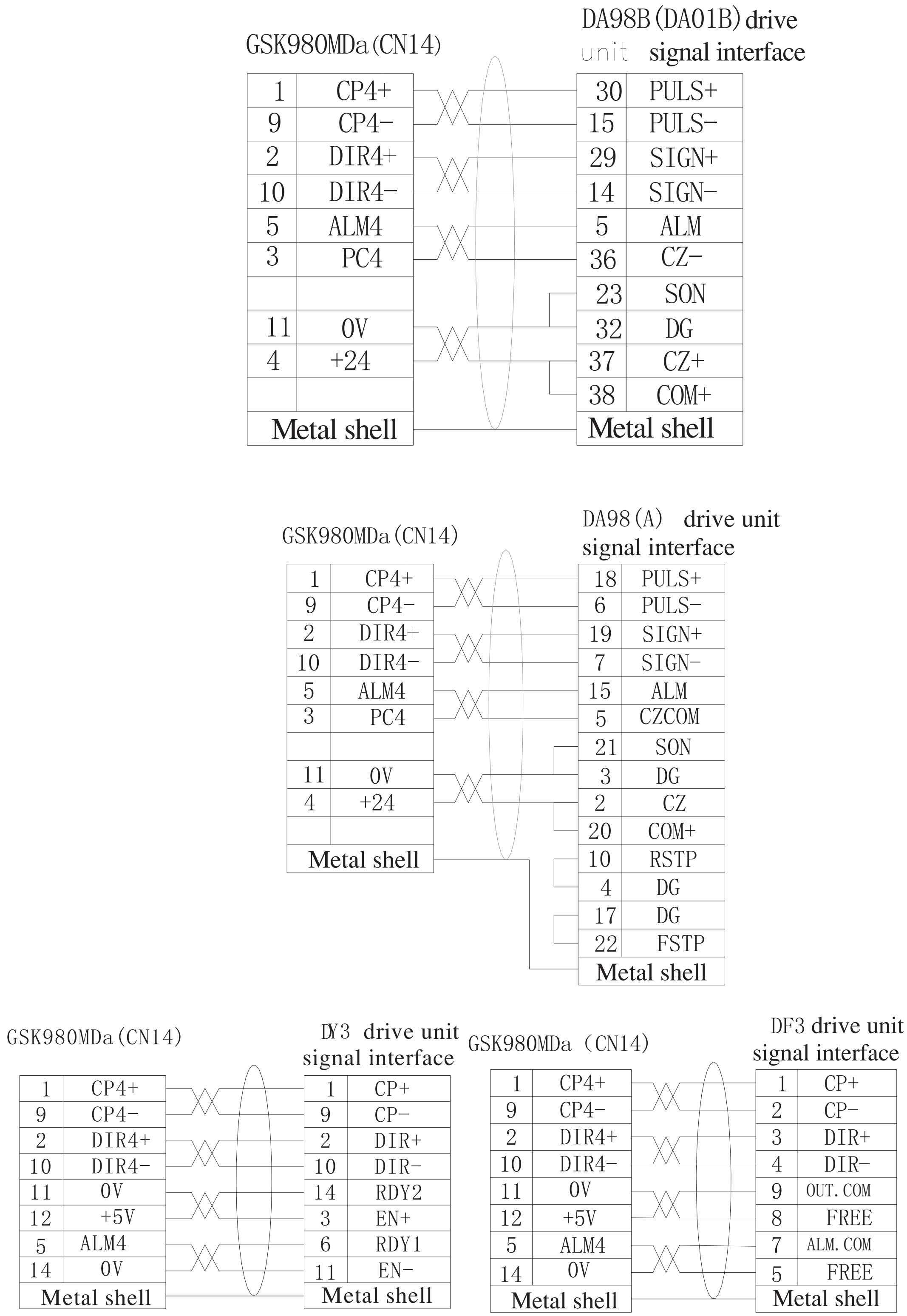


Fig.2-12 Connection of 4th axis interface to drive unit

2.2.3 Connection of 4th axis interface as rotary axis

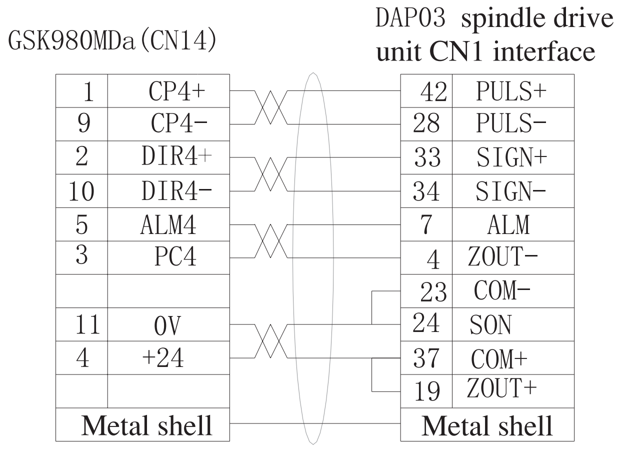


Fig.2-13 Connection of 4th axis interface to spindle drive unit

2.3 Connection of spindle port

2.3.1 Definition of signal

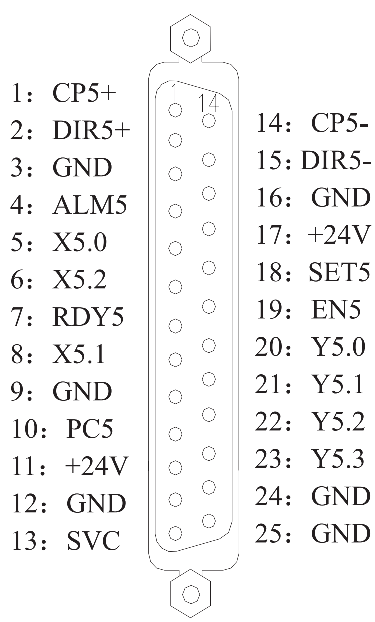


Fig.2-14 CN15 Spindle Prot

CP5+, CP5-	Spindle pulse signal
DIR5+, DIR5-	Spindle direction signal
ALM5	Spindle alarm signal
RDY5	Spindle is ready
PC5	Spindle zero signal
SVC	Output of voltage
SET5	Spindle disable signal
EN5	Spindle enable signal
X5.0~X5.2	PLC Address,only For these,Lower voltage is valid
Y5.0~Y5.3	PLC address

2.3.2 Spindle zero signal

Except for the PC5 signal, other fixed signals of the spindle interface are the same as that of the X,Y,Z, 4th axes. the PC5 interface circuit is shown as follows:

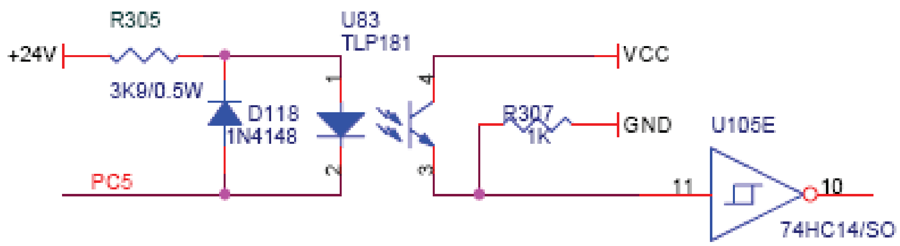


Fig.2-15 Spindle zero signal interface circuit

2.3.3 Linear axis

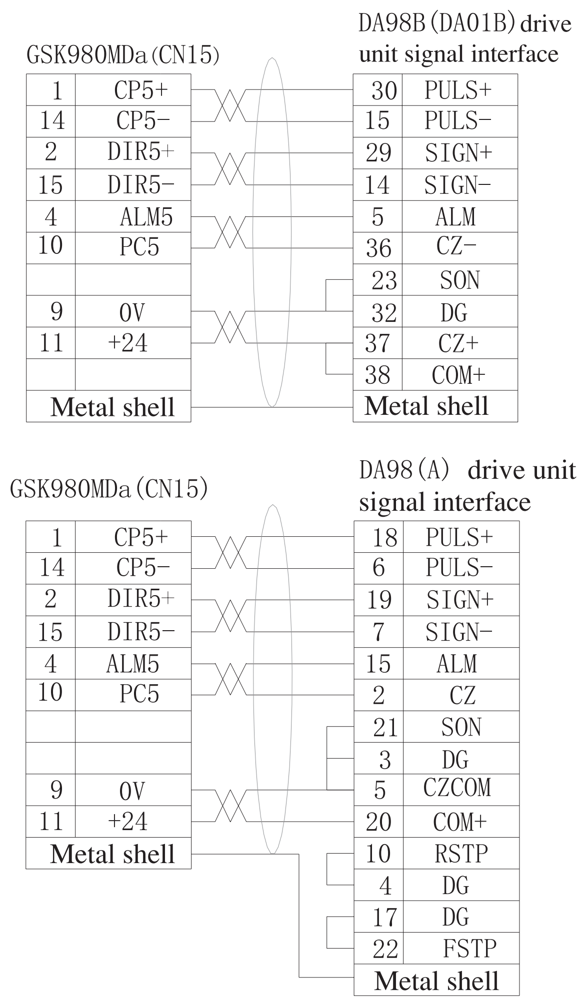


Fig.2-16 Connection of spindle interface to drive unit

2.3.4 Connected with inverter

The connection of GSK980MDa with convertor is shown in Fig. 2-17:

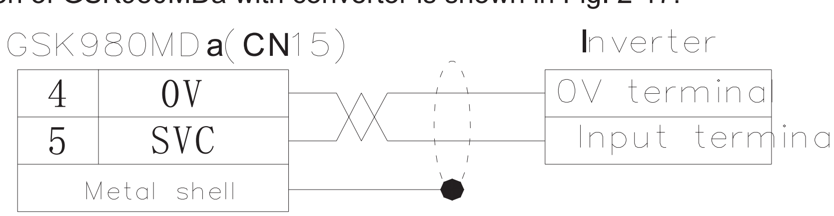


Fig.2-17 Connection of GSK980MDa to inverter

2.3.5 Connection of spindle interface as rotary axis

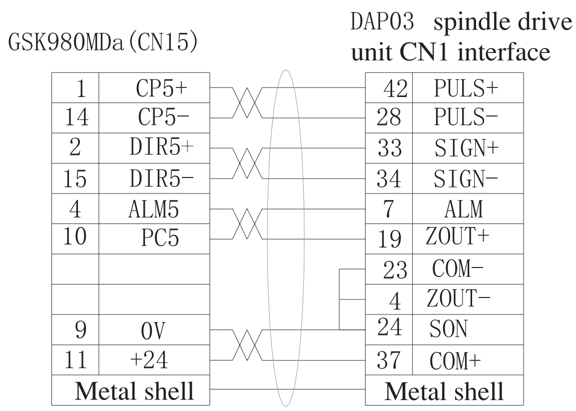


Fig.2-18 Connection of spindle to DAP03

2.3.6 Connection of spindle interface as "CS" axis

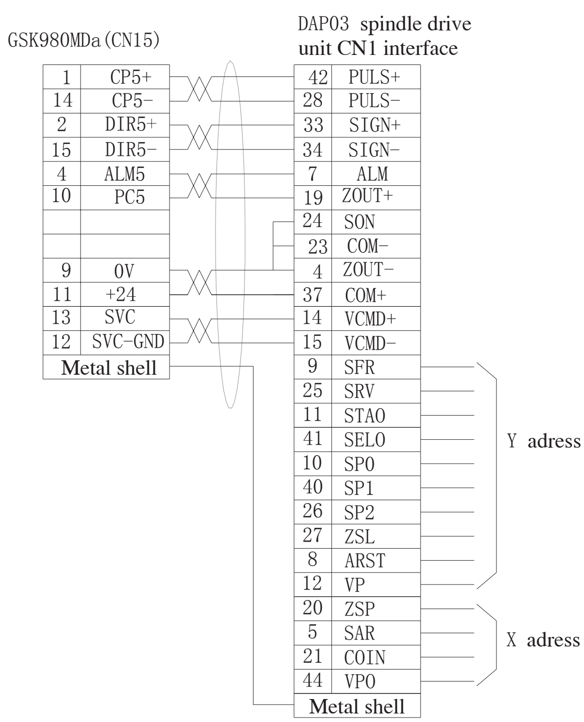


Fig.2-19 Connection of spindle to DAP03

2.3.7 SVC Signal explanation

The analog spindle interface SVC can output 0~10V voltage, its interior signal circuit is shown in Fig. 2-20:

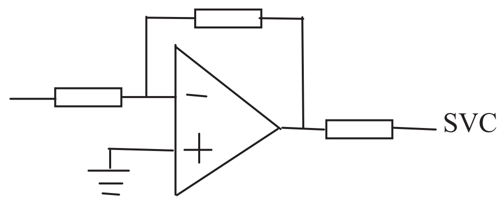
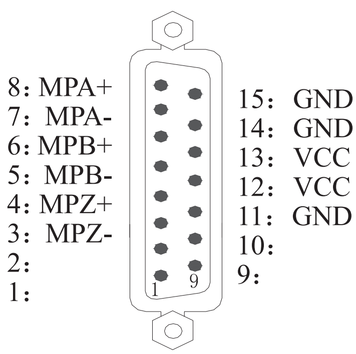


Fig 2-20 SVC Signal circuit

2.4 Connection to Spindle Encoder

2.4.1 Spindle encoder interface definition



Name	Explanation
MPA-/MPA+	Encode A phase pulse
MPB-/MPB+	Encode B phase pulse
MPZ-/MPZ+	Encode Z phase pulse

Fig.2-21 CN21 Encode interface
（DB15 male socket）

2.4.2 Signal Explanation

MPZ-/MPZ+, MPB-/MPB+, MPA-/MPA+ are the encoder Z, B, A phase differential input signals respectively, which are received by 26LS32; MPB-/MPB+, MPA-/MPA+ are normal square wave of phase shift 90°with the maximum signal frequency less than 1MHz; the encoder pulses for GSK980MDa are set by data parameter No.109, whose range is from 100 to 5000.

Its interior connection circuit is shown in Fig. 2-22: (n=A, B, C)

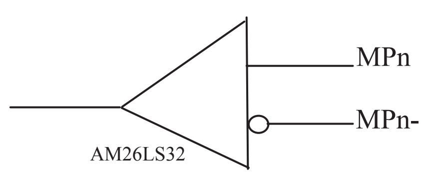


Fig.2-22 Encode signal circuit

2.4.3 Connection of spindle encoder interface

The connection of GSK980MDa to spindle encoder is shown in Fig. 2-23, twisted pair cables are used to connection.

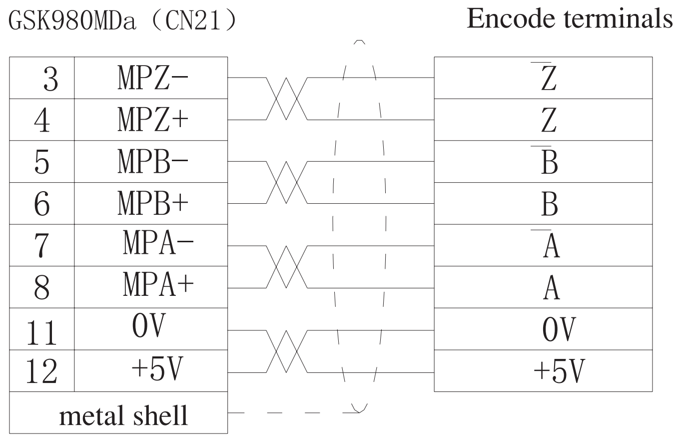


Fig.2-23 Connection of GSK980MDa to encoder

2.5 Connection to Handwheel

2.5.1 Handwheel interface definition

13：GND	26：
12：GND	25：
11：GND	24：
10：GND	23：X6.5
9：X6.3	22：X6.4
8：X6.2	21：
7：	20：
6：X6.1	19：
5：X6.0	18：+24V
4：HB-	17：+24V
3：HB+	16：+5V
2：HA-	15：+5V
1：HA+	14：+5V

Signal	Explanation
HA+, HA-	Handwheel A phase signal
HB+, HB-	Handwheel B phase signal
X6.0~X6.5	PLC adress
+24V	Direct current
VCC, GND	

Fig.2-24 CN31 handwheel interface
（3-line DB26 male socket）

2.5.2 Signal explanation

"HA+", "HA-", "HB+", "HB-" are the input singals of handwheel A and B phases. Its interior connection circuit is shown in Fig. 2-25:

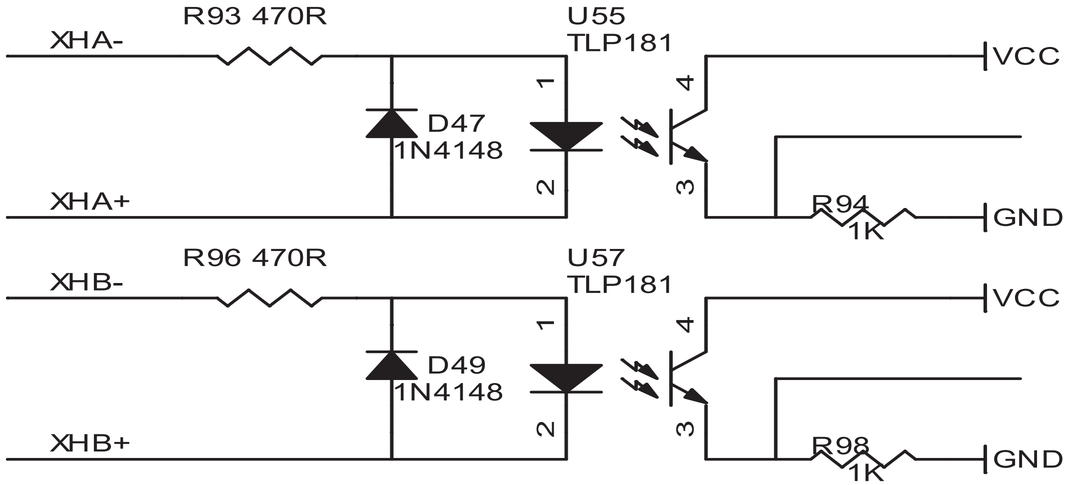


Fig.2-25 Handwheel signal circuit

The connection of GSK980MDa to handwheel is shown in Fig. 2-26：

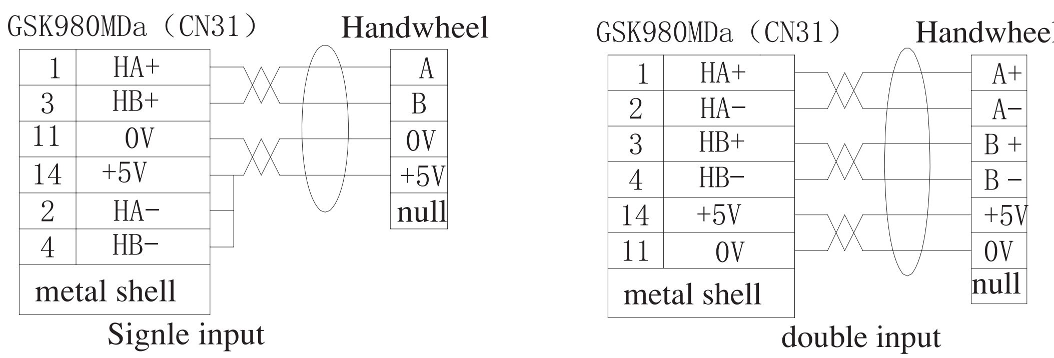
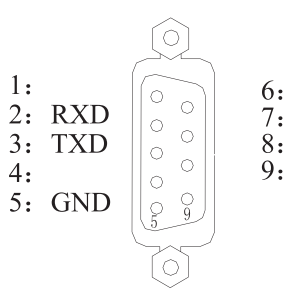


Fig.2-26 Connection of GSK980MDa to handwheel

2.6 Connection of GSK980MDa to PC

2.6.1 Communication interface definition



Signal	Explanation
RXD	For date reception
TXD	For date transmiting
GND	For signal grounding

Fig.2-27 CN51 communication interface
(DB9 female socket)

2.6.2 Communication interface connection

The communication between GSK980MDa and PC can be done via RS232 interface (GSK980MDa communication software needed), The connection of them is shown in Fig.2-28

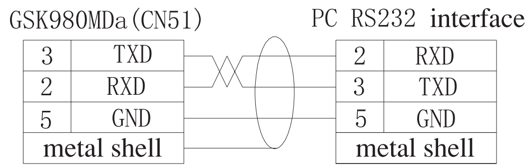


Fig.2-28 Connection of GSK980MDa to PC

The communication of a GSK980MDa to another GSK980MDa can be made via their CN51 interfaces, and the connection of them is shown in Fig.2-29:

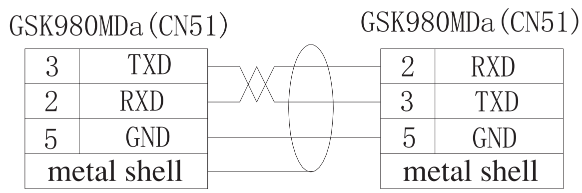


Fig.2-29 Communication connection of GSK980MDa to GSK980MDa

2.7 Connection of Power Interface

GSK-PB2 power box is applied in this GSK980MDa, which involves 4 groups of voltage: +5V（3A）, +12V（1A）, -12V（0.5A）, +24V（0.5A）, and its commom terminal is COM（0V）. The connection of GSK-PB2 power box to GSK980MDa CN1 interface has been done for its delivery from factory, and the user only need to connect it to a 220V AC power in using:

The interface definition of GSK980MDa CN1 is shown below:

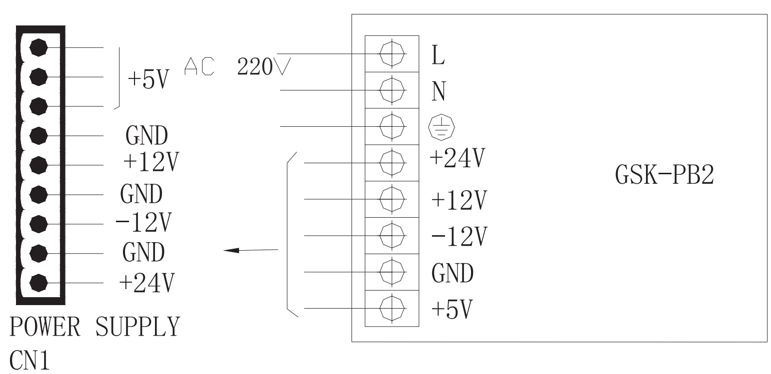


Fig.2-30

2.8 I/O Interface Definition：

CN61：44-core (3-line) male socket

NO.	Address	NO.	Address	NO.	Address	NO.	Address
1	X0.0	12	X1.3（DECZ）	23	GND	34	X2.5(DEC5)
2	X0.1	13	X1.4	24	GND	35	X2.6
3	X0.2	14	X1.5	25		36	X2.7
4	X0.3（DECX）	15	X1.6	26		37	X3.0
5	X0.4	16	X1.7	27		38	X3.1
6	X0.5（ESP）	17		28		39	X3.2
7	X0.6	18		29	X2.0	40	X3.3
8	X0.7	19		30	X2.1	41	X3.4
9	X1.0	20		31	X2.2	42	X3.5（SKIP）
10	X1.1	21	GND	32	X2.3（DECY）	43	X3.6
11	X1.2	22	GND	33	X2.4（DEC4）	44	X3.7

CN62：44-core (3-line) female socket

NO.	Address	NO.	Address	NO.	Address	NO.	Address
1	Y0.0	12	Y1.3	23	+24V	34	Y2.5
2	Y0.1	13	Y1.4	24	+24V	35	Y2.6
3	Y0.2	14	Y1.5	25	+24V	36	Y2.7
4	Y0.3	15	Y1.6	26	GND	37	Y3.0
5	Y0.4	16	Y1.7	27	GND	38	Y3.1
6	Y0.5	17	GND	28	GND	39	Y3.2
7	Y0.6	18	GND	29	Y2.0	40	Y3.3
8	Y0.7	19	GND	30	Y2.1	41	Y3.4
9	Y1.0	20	+24V	31	Y2.2	42	Y3.5
10	Y1.1	21	+24V	32	Y2.3	43	Y3.6
11	Y1.2	22	+24V	33	Y2.4	44	Y3.7

Note 1: The I/O function of GSK980MDa drilling and milling CNC is defined by ladder diagram;

Note 2:If output function is valid, the output signal is on to 0V. If output function is invalid, the output signal is cut off by high impendance;

Note 3: If input function is valid, the input signal is on to 24V. If input function is invalid, the input signal is cut off with it;

Note 4: The effectiveness of +24V, 0V is equal to GSK980MD power box terminals that have the same name;

Note 5: XDEC, YDEC, ZDEC, DEC4, DEC5, ESP, SKIP are fixed signals that can't be altered.

2.8.1 Input Signal

Input signal means the signal from machine to CNC, when this signal is on with +24V, the input is valid; when it is off with +24V, the input is invalid. The contact point of input signal at machine side should meet the following conditions:

The capacity of the contact point: DC30V, 16mA above

Leakage current between contact points in open circuit: 1mA below

Voltage drop between contact points in closed circuit: 2V below (current 8.5mA, including cable voltage drop)

There are two external input types for input signals: one type is input by trigger point switch whose signals are from keys, stroke switch and contacts of relay at machine side, as is shown in Fig 2-31:

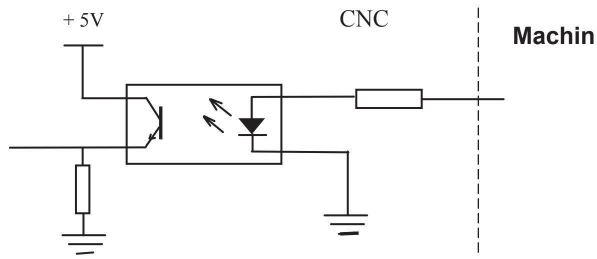


Fig.2-31

The other type is input by switch with no contacts (transistor), as is shown in Fig. 2-32, 2-33

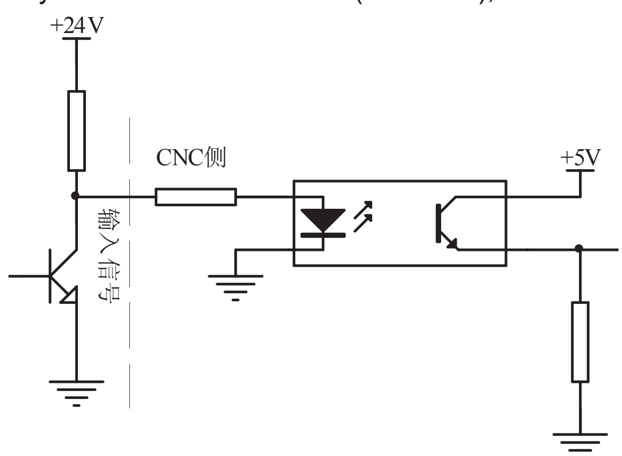


Fig.2-32 Connection of NPN

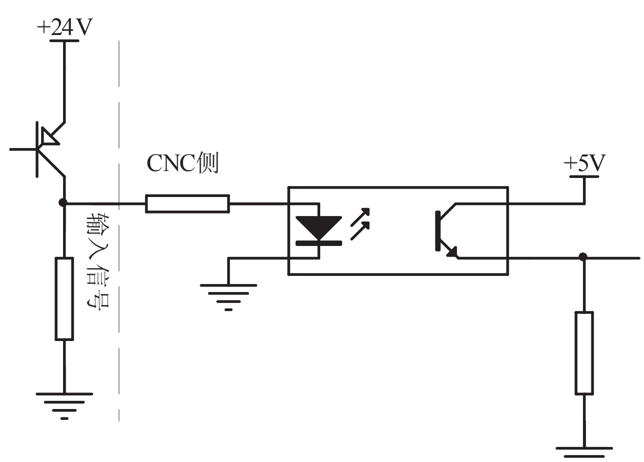


Fig.2-33 Connection of PNP

2.8.2 Output signal

The output signal is used for the machne relay and indicator, if it is on with 0V, the output function is valid; if it is off with 0V, the output function is invalid. There are total 36 digital volume outputs in I/O interface that they all have the same structure as is shown in Fig.2-34:

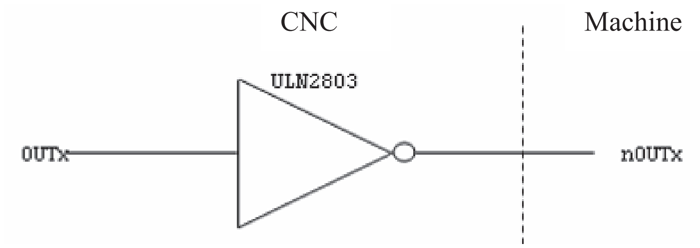


Fig.2-34 Circuit for digital volume output module

The logic signal OUTx output from the main board is sent to the input terminal of inverter (ULN2803) via a connector. And there are 2 output types for nOUTx: output with 0V, or high impedance. Its typical application is shown in follows:

- To drive LED

  A serial resistance is needed to limit the current (usually 10mA) that goes through the LED by using ULN2803 output to drive LED, which is shown in Fig.2-35

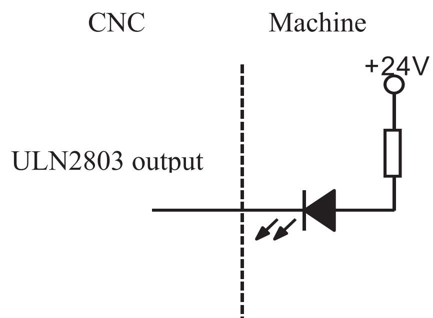


Fig.2-35

- To drive filament indicator

  An external preheat resistance is needed to decrease the current impact at power on by using ULN2803 output to drive filament indicator, and this resistance value should be within a range that the indicator cann't light up. It is shown in Fig.2-36:

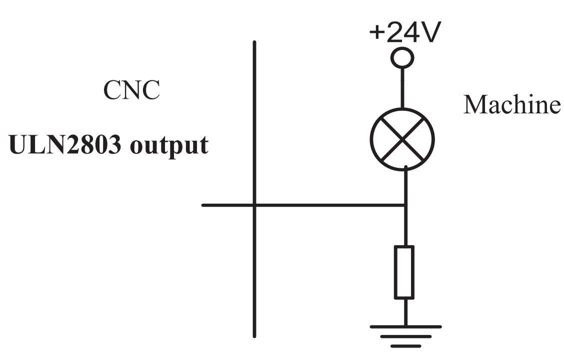


Fig. 2-36

- To drive inductive load (relay etc.)

To use ULN2803 output to drive an inductive load, it requires to connect a freewheeling diode near the coil to protect output circuit and deduce interference. It is shown in Fig.2-37:

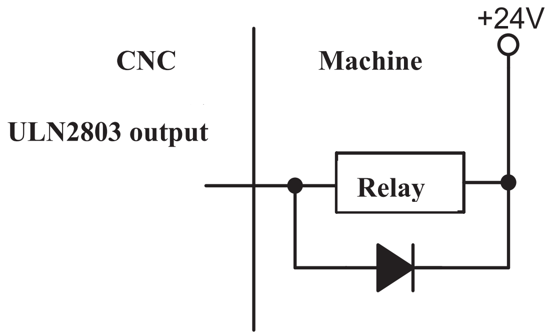


Fig.2-37

2.9 Machine Zero

- Relative signal

DECX	X axis deceleration signal		PCX	X axis zero signal
DECY	Y axis deceleration signal		PCY	Y axis zero signal
DECZ	Z axis deceleration signal		PCZ	Z axis zero signal
DEC4	4th axis deceleration signal		PC4	4th axis zero signal
DEC5	5th axis deceleration signal		PC5	5th axis zero signal

- CNC diagnosis

0 0 0				DEC5	DEC4	DECZ	DECY	DECX
Corresponding pin-out				CN61.34	CN61.33	CN61.12	CN61.32	CN61.4
PLC address				X2.5	X2.4	X1.3	X2.3	X0.3

0 0 8				PC5	PC4	PCZ	PCY	PCX
Corresponding pin-out				CN15.10	CN14.3	CN13.3	CN12.3	CN11.3

- Bit parameter

0 0 4			DECI					

DECI =1: Deceleration signal is on with 24V for deceleration when machine zero return is performed

=0: Deceleration signal is off 24V for deceleration when machine zero return is performed

0 0 6				ZM5	ZM4	ZMZ	ZMY	ZMX

ZMX =1：X axis machine zero return type C;

=0：X axis machine zero return type B.

ZMY =1：Y axis machine zero return type C;

=0：Y axis machine zero return type B.

ZMZ =1：Z axis machine zero return type C;

=0：Z axis machine zero return type B.

ZM4 =1：4th axis machine zero return type C;

=0：4th axis machine zero return type B.

ZM5 =1：5th axis machine zero return type C;

=0：5th axis machine zero return type B.

0	0	7					ZC5	ZC4	ZCZ	ZCY	ZCX

ZCX =1：The deceleration signal （DECX）and one-rotation signal （PCX）of X axis are in parallel connection during machine zero return（a proximity switch acting as both the deceleration signal and zero signal）;

=0：The deceleration signal（DECX）and one-rotation signal（PCX）of X axis are connected independently during machine zero return（the indepent deceleration signal and zero signal are required）.

ZCY =1：The deceleration signal （DECY）and one-rotation signal （PCY）of Y axis are in parallel connection during machine zero return（a proximity switch acting as both the deceleration signal and zero signal）;

=0：The deceleration signal（DECY）and one-rotation signal（PCY）of Y axis are connected independently during machine zero return (the indepent deceleration signal and zero signal are required）.

ZCZ =1：The deceleration signal（DECZ） and one-rotation signal （PCZ）of Z axis are in parallel connection during machine zero return（a proximity switch acting as both the deceleration signal and zero signal）;

=0：The deceleration signal（DECZ） and one-rotation signal（PCZ）of Z axis are connected independently during machine zero return（the indepent deceleration signal and zero signal are required）.

ZC4 =1：The deceleration signal（DEC4） and one-rotation signal （PC4）of 4th axis are in parallel connection during machine zero return（a proximity switch acting as both the deceleration signal and zero signal）;

=0：The deceleration signal（DEC4） and one-rotation signal （PC4）of 4th axis are connected independently during machine zero return（the indepent deceleration signal and zero signal are required）.

ZC5 =1：The deceleration signal（DEC5） and one-rotation signal （PC5）of 5th axis are in parallel connection during machine zero return（an proximity switch acting as both the deceleration signal and zero signal）;

=0：The deceleration signal（DEC5） and one-rotation signal（PCZ）of 5th axis are connected

Volume Ⅲ Installation

independently during machine zero return (the indepent deceleration signal and zero signal are required) .

0	1	1

					ZNIK		

ZNLK =1：The direction keys are locked as machine zero return is performed,by pressing the direction key once,it moves to the machine zero automatically and stops,By pressing the key at the machine zero return,the motion stops immediately;

=0：The direction keys are not locked as machine zero return is performed, but the direction keys should be pressed and held on

0	1	2

							ISOT

ISOT =1：Manual rapid traverse valid prior to machine zero return;

=0：Manual rapid traverse invalid prior to machine zero return.

0	1	4

			ZRS5	ZRS4	ZRSZ	ZRSY	ZRSX

ZRSZ, ZRSX, ZRSY, ZRS4, ZRS5 =1: To select machine zero return type B, C, which have machine zero, it needs to detect deceleration and zero signals in machine zero return;

=0: To select machine zero return type A, which has no machine zero, it does not detect deceleration and zero signals in machine zero return.

0	2	2

			MZR5	MZR4	MZRZ	MZRY	MZRX

MZRX, MZRZ, MZRY, MZR4, MZR5 =1：The direction of zero return is negative for X, Z, Y ,4th,5th axes;

=0：The direction of zero return is positive for X, Z, Y,4th ,5th axes

- Date parameter

089	Low speed of machine zero return of X axis
090	Low speed of machine zero return of Y axis
091	Low speed of machine zero return of Z axis
092	Low speed of machine zero return of 4th axis
093	Low speed of machine zero return of 5th axis

094	High speed of machine zero return of X axis
095	High speed of machine zero return of Y axis
096	High speed of machine zero return of Z axis
097	High speed of machine zero return of 4th axis
098	High speed of machine zero return of 5th axis

130	X axis machine zero offset (0.001)
131	Y axis machine zero offset (0.001)
132	Z axis machine zero offset (0.001)
133	The 4th axis machine zero offset (0.001)
134	The 5th axis machine zero offset (0.001)

145	X machine coordinate of the 1st reference point (0.001mm)
146	Y machine coordinate of the 1st reference point (0.001mm)
147	Z machine coordinate of 1st reference point (0.001mm)
148	4th machine coordinate of the 1st reference point (0.001mm)
149	5th machine coordinate of the 1st reference point (0.001mm)

150	X machine coordinate of the 2nd reference point (0.001mm)
151	Y machine coordinate of the 2nd reference point (0.001mm)
152	Z machine coordinate of the 2nd reference point (0.001mm)
153	4th machine coordinate of the 2nd reference point (0.001mm)
154	5th machine coordinate of the 2nd reference point (0.001mm)

155	X machine coordinate of the 3rd reference point (0.001mm)
156	Y machine coordinate of the 3rd reference point (0.001mm)
157	Z machine coordinate of the 3rd reference point (0.001mm)
158	4th machine coordinate of the 3rd reference point (0.001mm)
159	5th machine coordinate of the 3rd reference point (0.001mm)

160	X machine coordinate of the 4th reference point (0.001mm)
161	Y machine coordinate of the 4th reference point (0.001mm)
162	Z machine coordinate of the 4th reference point (0.001mm)
163	4th machine coordinate of the 4th reference point (0.001mm)
164	5th machine coordinate of the 4th reference point (0.001mm)

- **Signal connection**

  The interior wiring circuit of deceleration signal is shown in Fig.2-37

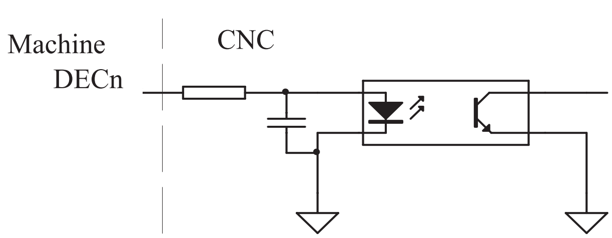
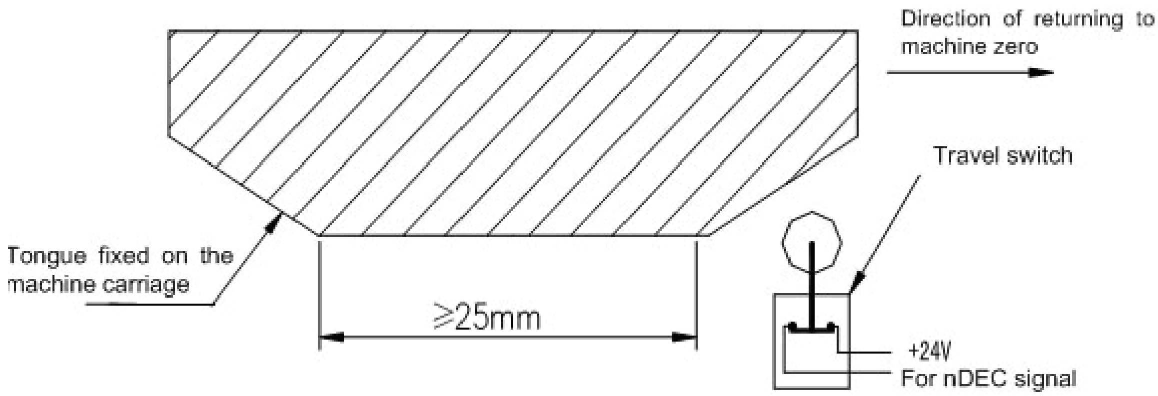


Fig.2-37

- **achine zero return type B by regarding servo motor one-rotation signal as zero signal**

  ①Its sketch map is shown in follows:



② The circuit of deceleration signal (for three axes)

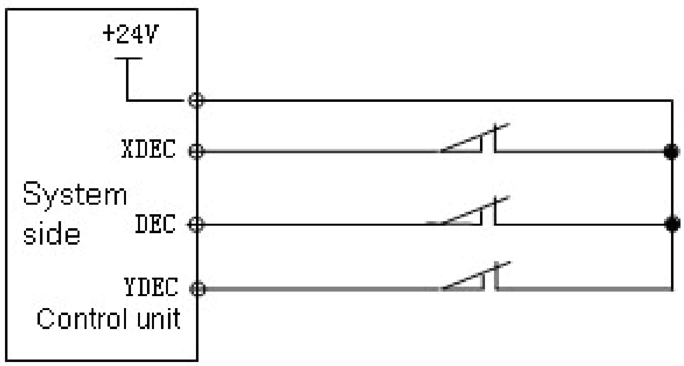


Fig.2-40

③ Action time sequence of machine zero return

When ZMn(n is X,Y,Z,4th,5th axis) of the bit parameter No.006, ZCn(n=X, Y, Z, 4th, 5th) of bit parameter No.007 and the BIT5（DECI）of the bit parameter No.004 are all set to 0, the deceleration signal low level is valid. The action time sequence of machine zero return is shown in follows

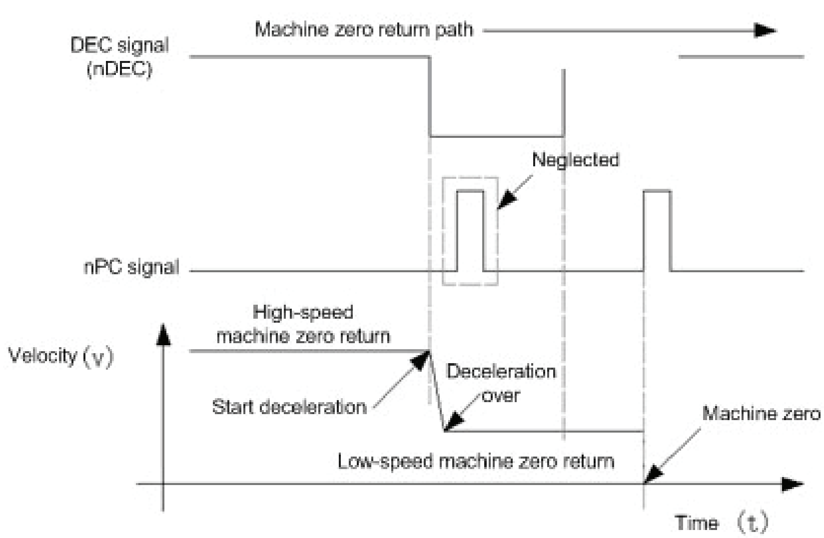


Fig.2-41

④Machine zero return process

A：Select machine zero return mode, press the manual positive or negative feed key(machine zero return direction is set by bit parameter No.022), the corresponding axis moves to the machine zero by a rapid traverse speed. As the axis press down the deceleration switch to cut off deceleration signal, the feed slows down immediately, and it continues to run in a fixed low speed.

B：When the deceleration switch is released, the deceleration signal contact point is closed again. And CNC begins to detect the encoder one-rotation signal, if the signal level changes, the motion will be stoped. And the corresponding zero indicator on the operator panel lights up for machine zero return completion

- **Machine zero return type B as an proximity switch is taken as both deceleration and zero signals**

① Its sketch map is shown in follows:

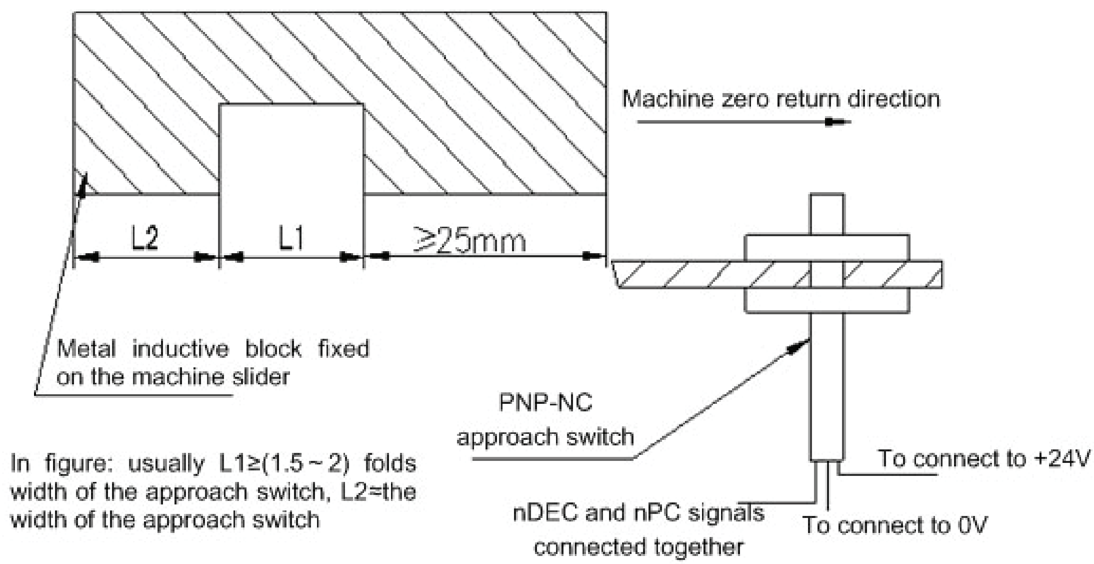


Fig.2-42

② Wiring of the deceleration signal

See details in Section 2.1.6 of this chapter

③ Action time sequence of machine zero return

When ZMn (n is X,Y,Z,4th ,5th axis)of the bit parameter No.006 and the BIT5（DECI）of the bit parameter No.004 are all set to 0, ZCn (n is X,Y,Z,4th ,5th axis)of the bit parameter No.007 is set to 1, the deceleration signal low level is valid . The action time sequence of zero return is shown in follows:

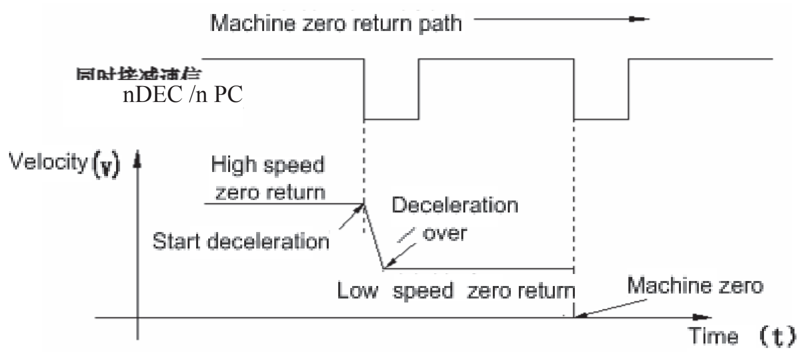


Fig.2-43 the action time sequence of zero return

④ Machine zero returns process

A：Select the Machine Zero mode, press manual positive or negative (zero return direction set by bit parameter No.183) feed key, the corresponding axis will move to the zero at a traverse speed.

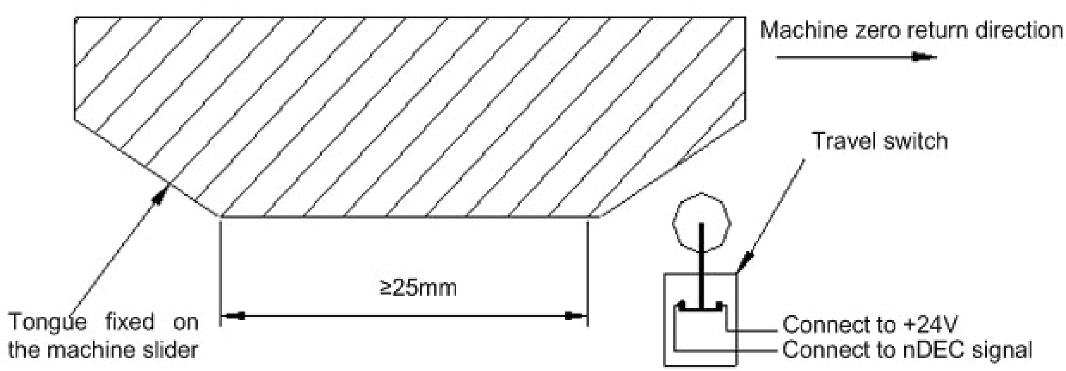
B：As the approach switch touches the tongue for the first time, the deceleration signal is valid and it slows down immediately to run in a low speed.

C：As the approach switch detaches the tongue, the deceleration signal is invalid, it moves at a fixed low speed after deceleration and starts to detect zero signal (PC).

D：As the approach switch touches the tongue for the second time, the zero signal is valid and the movement stops. The indicator for zero return on the panel lights up.

- **Machine zero return type C as servo motor one-rotation signal taken as zero signal**

① Its sketch map is shown below:



② Circuit of the deceleration signal

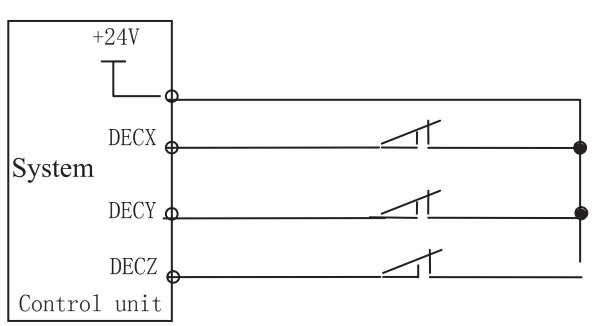


Fig.2-45

③ Action time sequence of machine zero return

When ZMn (n is X,Y,Z,4th ,5th axis) of the bit parameter No.006 are all set for 1, ZCn (n is X,Y,Z,4th ,5th axis)of the bit parameter No.007 are all set for 0, the BIT5（DECI）of the bit parameter No.004 is set for 0, and the deceleration signal low level is valid. The action time sequence of machine zero return is shown in follows

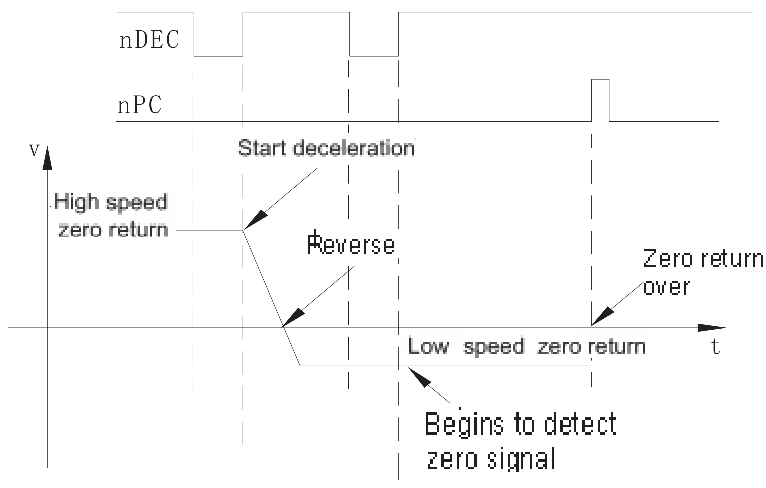


Fig.2-46

④ Machine zero returns process

A : Select the Machine Zero mode, press manual positive or negative (zero return direction set by bit parameter №022) feed key, the corresponding axis will move to the machine zero at a traverse speed. Then it touches the tongue and presses down the deceleration switch, and moves forward. When the tongue detaches the deceleration switch, the axis slows down to zero, then moves reversely and accelerates to a fixed low speed for continuous moving

B：As the tongue touches the deceleration switch for the second time, it moves on till the tongue detaches the deceleration switch. And it begins to detect the zero signals. If the zero signal level changes, the movement stops. Then zero return indicator of the corresponding axis on the panel lights up and machine zero operation is finished.

- **Machine zero return type C as an proximity switch is taken as both deceleration and zero signals**

① Its sketch map is shown below:

Volume Ⅲ Installation

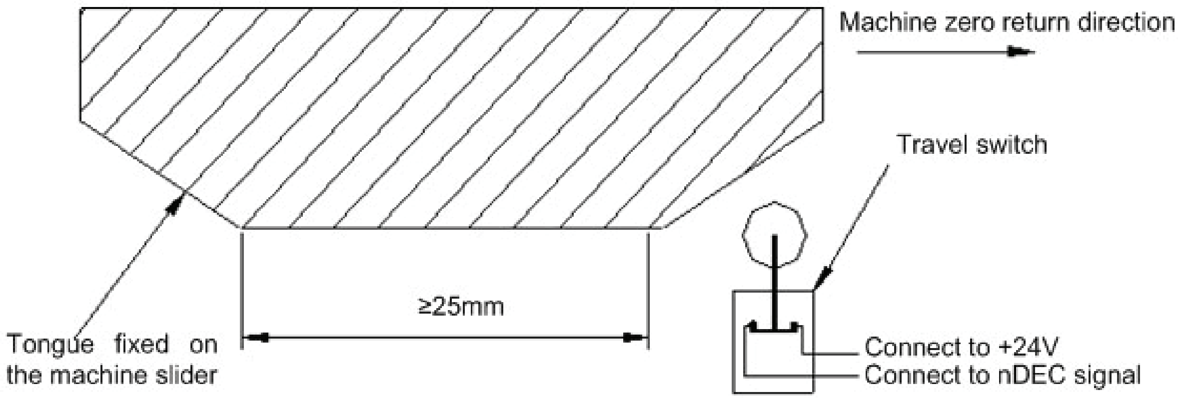


Fig.2-47

② Circuit of the deceleration signal

See details in Section 2.1.6 of this chapter

③ Action time sequence of machine zero return

When ZMn (n is X,Y,Z,4th ,5th axis) of the bit parameter No.006 and ZCn (n is X,Y,Z,4th ,5th axis)of the bit parameter No.007 are all set to 1, the BIT5（DECI）of the bit parameter No.004 is set to 0, the deceleration signal low level is valid. The action time sequence of machine zero return is shown in follows:

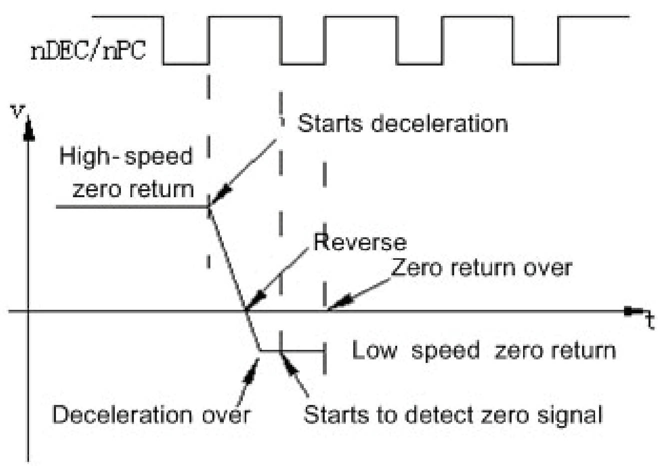


Fig.2-48

④ Machine zero returns process

A：Select the Machine Zero mode, press manual positive or negative (zero return direction is set by bit parameter No.183) feed key, the corresponding axis will move to the machine zero at a traverse speed. Then it touches the tongue and presses down the deceleration switch, and moves forward. When the tongue detaches the deceleration switch, the axis slows down to zero speed, then moves reversely and accelerates to a fixed low speed for continuous moving

B：As the tongue touches the deceleration switch for the second time, it begins to detect the zero signal. It moves on till the tongue detaches the deceleration switch, the movement stops immediately. Then zero return indicator of the corresponding axis on the panel lights up and machine zero return operation is finished.

CHAPTER 3 PARAMETER

In this chapter the CNC bit and data parameters are introduced. Various functions can be set by these parameters.

3.1 Parameter Description (by sequence)

3.1.1 Bit parameter

The expression of bit parameter is shown in follows：

Parameter NO.	BIT7	BIT6	BIT5	BIT4	BIT3	BIT2	BIT1	BIT0

0 0 1	***	***	***	ACS	HWL	***	***	***

ACS =1: Analog voltage control of spindle speed;
=0: Switching control of spindle speed.
HWL =1: MPG mode;
=0: Step mode.

0 0 2	***	***	***	LIFJ	MDITL	LIFC	NRC	TLIF

LIFJ =1: Tool life management group skip valid;
=0: Tool life management group skip invalid.
MDITL =1: Tool life management valid in MDI mode;
=0: Tool life management invalid in MDI mode.
LIFC =1: Tool life counting type 2, by times;
=0: Tool life counting type 1, by times.
NRC =1: Tool nose radius compensation valid;
=0: Tool nose radius compensation invalid.
TLIF =1: Tool life management valid;
=0: Tool life management invalid.

0 0 3	***	***	PCOMP	***	***	***	D/R	***

PCOMP =1: Screw-pitch error compensation valid;
=0: Screw-pitch error compensation invalid.
D/R =1: Tool offset D is diameter value;
=0: Tool offset D is radius value.

0 0 4	***	RDRN	DECI	***	PROD	***	***	SCW

RDRN =1：In G00 dry run mode, speed=feedrate × speed of dry run;
=0：G00 speed = rapid override × rapid tranverse speed.
DECI =1：Deceleration signal high level for machine zero return;
=0：Deceleration signal low level for machine zero return.

PROD =1：Relative coordinate displayed in POSITION page is programming position;

=0：Relative coordinate displayed in POSITION page involving tool compensation.

SCW =1：Inch output(inch system)valid after repower;

=0：Metric output(metric system)valid after repower

The functions of metric and inch system

There are two kinds of input and output units for CNC numerical control system: metric unit, millimeter (mm) and English unit (inch).

Output increement unit is set by Bit0（SCW）of bit parameter №004 in GSK980MDa system. SCW=0 indicates that minimum command increment, parameter and screw–pitch values are in metric units; SCW=1 indicates that minimum command increment, parameter and screw–pitch values are in inches units. The setting of this parameter depends on machine tool.

G code: By selecting G20/G21 code, it is able to set whether minimum input increment values are in inch or in metric. Executing G21 indicates that minimum input increment values are in metric; and executing G20 indicates that values are in inch,

0	0	5	***	***	SMAL	M30	***	***	PPD	PCMD

SMAL =1：Spindle manual gear shift for S command;

=0：Spindle auto gear shift for S command.

M30 =1：Cursor returns to beginning after M30 execution;

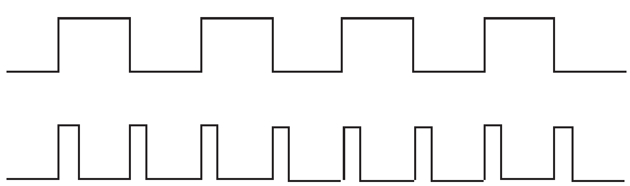
=0：Cursor not to beginning after M30 execution.

PPD =1：Relative coordinate set by G92;

=0：Relative coordinate not set by G92.

PCMD =1：Axial output wave form is pulse;

=0：Axial output wave form is square.



Square output，max. output frequency 266KPPS

Pulse output，max. output frequency 266KPPS，Pulse width 1 μs.

0	0	6	***	***	***	ZM5	ZM4	ZMZ	ZMY	ZMX

ZM5 =1：5th zero return type C;

=0：5th zero return type B.

ZM4 =1：4th zero return type C;

=0：4th zero return type B.

ZMZ =1：Z zero return type C;

=0：Z zero return type B.

ZMY =1：Y zero return type C;

=0：Y zero return type B.

ZMX =1：X zero return type C;

=0：X zero return type B.

0 0 7	AVGL	***	SMZ	ZC5	ZC4	ZCZ	ZCY	ZCX

On the condition that blocks smoothing transition is valid, more smooth velocity link and better machining quality will be obtained during the path transition from line to line or from line to arc by properly changing the linear feedrate.

So the actual output speed may be different to the programming speed when using this function. And it may also differ as regard to the linear segment with the same programming speed. The deviation is not more than 15mm/min between the actual output speed and the programming speed on the condition that the programming speed F is less than 1200mm/min

AVGL =1：When SMZ=0 linear smoothing is valid,i.e. smoothing transition function is valid;
=0：Linear smoothing transition function is invalid.

SMZ =1：To execute next block till all moving blocks executed;
=0：For smooth transition between blocks.

ZC5 =1：Deceleration signal (DEC5)and one-rotation signal (PC5) of 5th axis are in parallel connection(a proximity switch taken as both deceleration signal and zero signal) during machine zero return;
=0：Deceleration signal (DEC5) and one-rotation signal (PC5) of 5th axis are connected independently (independent deceleration signal and zero signal are required) during machine zero return.

ZC4 =1：Deceleration signal (DEC4)and one-rotation signal (PC4) of 4th axis are in parallel connection (a proximity switch taken as both deceleration signal and zero signal) during machine zero return;
=0：Deceleration signal (DEC4) and one-rotation signal (PC4) of 4th axis are connected independently (independent deceleration signal and zero signal are required) during machine zero return.

ZCZ =1：Deceleration signal (DECZ) and one-rotation signal (PCZ) of Z axis are in parallel connection a proximity switch taken as both deceleration signal and zero signal) during machine zero return;
=0：Deceleration signal (DECZ) and one-rotation signal (PCZ) of Z axis are connected independently (independent deceleration signal and zero signal are required) during machine zero return.

ZCY =1：Deceleration signal (DECY) and one-rotation signal (PCY) of Y axis are in parallel connection a proximity switch taken as both deceleration signal and zero signal) during machine zero return;
=0：Deceleration signal (DECY) and one-rotation signal (PCY) of Y axis are connected independently (independent deceleration signal and zero signal are required) during machine zero return.

ZCX =1：Deceleration signal (DECX)and one-rotation signal (PCX) of X axis are in parallel connection a proximity switch taken as both deceleration signal and zero signal) during machine zero return;

=0：Deceleration signal (DECX) and one-rotation signal (PCX) of X axis are connected independently (independent deceleration signal and zero signal are required) during machine zero return.

0	0	8		DISP	***	***	DIR5	DIR4	DIRZ	DIRY	DIRX

DISP =1：Enter absolute page after power on;

=0：Enter relative page after power on.

DIR5 =1：Direction signal (DIR)is high level as 5^{th} axis moves positively;

=0：Direction signal (DIR)is low level as 5^{th} axis moves negatively.

DIR4 =1：Direction signal (DIR)is high level as 4^{th} axis moves positively;

=0：Direction signal (DIR)is low level as 4^{th} axis moves negatively.

DIRZ =1：Direction signal (DIR)is high level as Z axis moves positively;

=0：Direction signal (DIR)is low level as Z axis moves negatively.

DIRY =1：Direction signal (DIR)is high level as Y axis moves positively;

=0：Direction signal (DIR)is low level as Y axis moves negatively.

DIRX =1：Direction signal (DIR)is high level as X axis moves positively;

=0：Direction signal (DIR)is low level as X axis moves negatively.

0	0	9		***	***	***	ALM5	ALM4	ALMZ	ALMY	ALMX

ALM5 =1：5^{th} axis low level alarm signal (ALM5);

=0：5^{th} axis high level alarm signal (ALM5).

ALM4 =1：4^{th} axis low level alarm signal (ALM4);

=0：4^{th} axis high level alarm signal (ALM4).

ALMZ =1：Z axis low level alarm signal (ALMZ);

=0：Z axis high level alarm signal (ALMZ).

ALMY =1：Y axis low level alarm signal (ALMY);

=0：Y axis high level alarm signal (ALMY).

ALMX =1：X axis low level alarm signal (ALMX);

=0：X axis high level alarm signal (ALMX).

0	1	0		CPF7	CPF6	CPF5	CPF4	CPF3	CPF2	CPF1	CPF0

CPF0～CPF7： Setting values of backlash compensation pulse frequency.

Set frequency =（2^7×CPF7+2^6×CPF6+2^5×CPF5+2^4×CPF4+2^3×CPF3+2^2×CPF2+2^1×CPF1+CPF0）Kpps

0	1	1		BDEC	BD8	***	***	***	ZNIK	***	***

BDEC =1：Backlash compensation type B, the compensation data are output by ascending type and the set frequency is invalid.;

=0：Backlash compensation type A, the compensation data are output by the set frequency (by bit parameter No.010) or 1/8 of it.

BD8 =1：Backlash compensation is done by the 1/8 of the set frequency;

=0：Backlash compensation is done by the set frequency.

Volume Ⅲ Installation

ZNIK =1：Direction keys locked during zero return, homing continues to end by pressing direction key once;

=0：Direction keys unlocked but should be held on during zero return.

0	1	2

***	***	***	TMANL	***	***	EBCL	ISOT

TMANL =1：Manual tool change for T code;

=0：Auto tool change for T code.

EBCL =1：Program end sign EOB displays “;”(semicolon);

=0：Program end sign EOB displays “*”(asterisk).

ISOT =1：Prior to machine zero return after power on, manual rapid traverse valid;

=0：Prior to machine zero return after power on, manual rapid traverse invalid.

0	1	3

SCRD	G01	RSCD	***	***	***	SKPI	G31P

SCRD =1：Coordinate system holding on at power down;

=0：Coordinate system not holding on at power down, G54 coordinate system is set after power on.

G01 =1：G01 status when power on;

=0：G00 status when power on.

RSCD =1：G54 coordinate system when reset 4;

=0：Coordinate system not changed when reset.

SKPI =1：High level valid for skip signal;

=0：Low level valid for skip signal.

G31P =1：G31 immediately stops when skip signal is valid;

=0：G31 slows down to stop when skip signal is valid.

0	1	4

***	***	***	ZRS5	ZRS4	ZRSZ	ZRSY	ZRSX

ZRS5 =1: There are machine zero point in 5th axis, it detects deceleration signal and zero signal when performing machine zero return;

=0: There are no machine zero point in 5th axis, it returns to machine zero without detecting deceleration signal and zero signal when performing machine zero return.

ZRS4 =1: There are machine zero point in 4th axis, it detects deceleration signal and zero signal when performing machine zero return;

=0: There are no machine zero point in 4th axis, it returns to machine zero without detecting deceleration signal and zero signal when performing machine zero return.

ZRSZ =1: There are machine zero point in Z axis, it detects deceleration signal and zero signal when performing machine zero return;

=0: There are no machine zero point in Z axis, it returns to machine zero without detecting deceleration signal and zero signal when performing machine zero return.

ZRSY =1: There are machine zero point in Y axis, it detects deceleration signal and zero signal when performing machine zero return;

=0: There are no machine zero point in Y axis, it returns to machine zero without detecting deceleration signal and zero signal when performing machine zero return.

ZRSX =1: There are machine zero point in X axis, it detects deceleration signal and zero signal when performing machine zero return;

=0: There are no machine zero point in X axis, it returns to machine zero without detecting deceleration signal and zero signal when performing machine zero return.

0	1	5		LPTK	RPTK	NAT	BRCH	***	***	***	***

LPTK =1：Hole locating is done by cutting feed on line continuous drilling;

=0：Hole locating is done by rapid feed on line continuous drilling;

RPTH =1: Hole locating is cutting path in circle and rectangle continuous drilling;

=0：Hole locating is rapid path in circle and rectangle continuous drilling;

NAT =1 Define the range of user macro program asin, atan;

=0：Not define the range of user macro program asin, atan;

BRCH =1：Plane returning is selected by G98 and G99 in continous drilling;

=0：Plane returning is selected by G99 in continous drilling

0	1	7		***	MST	MSP	MOT	MESP	***	***	***

MST =1：External cycle start signal (ST) invalid，

=0：External cycle start signal (ST) valid.

MSP =1：External stop signal (SP) invalid，

=0：External stop signal (SP) valid with external stop switch connected, otherwise CNC shows “stop” .

MOT =1：Not detect software stroke limit;

=0：Detect software stroke limit.

MESP =1：Emergency stop invalid;

=0：Emergency stop valid.

0	1	8		***	***	***	ESCD	***	***	***	***

ESCD =1：S code off at emergency stop;

=0：S code not off at emergency stop.

0	1	9		KEY1	***	***	HNG5	HNG4	HNGZ	HNGY	HNGX

KEY1 =1：Prog. switch ON after power on;

=0：Prog. switch OFF after power on.

HNG5 =1：5th MPG:ccw:+,cw:-;

=0：5th MPG:ccw:-,cw:+.

HNG4 =1：4th MPG:ccw:+,cw:-;

=0：4th MPG:ccw:-,cw:+.

HNGZ =1：Z MPG:ccw:+,cw:-;

=0：Z MPG:ccw:-,cw:+.

HNGY =1：Y MPG:ccw:+,cw:-;
=0：Y MPG:ccw:-,cw:+.
HNGX =1：X MPG:ccw:+,cw:-;
=0：X MPG:ccw:-,cw:+.

0	2	0		SPFD	SAR	THDA	VAL5	VAL4	VALZ	VALY	VALX

SPFD =1：Cutting feed stops if spindle stops;
=0：Cutting feed not stop after spindle stop.
SAR =1：Detect spindle SAR signal prior to cutting;
=0：Not detect spindle SAR signal prior to cutting.
THDA =1：Thread machining adopts exponential acceleration and deceleration;
=0：Thread machining adopts linear acceleration and deceleration.
VAL5 =1：For 5th axis move key,↑ is positive，↓is negative;
=0：For 5th axis move key, ↓is positive，↑is negative.
VAL4 =1：For 4th axis move key,↑ is positive，↓is negative;
=0：For 4th axis move key, ↓is positive，↑is negative.
VALZ =1：For Z axis move key,↑ is positive，↓is negative;
=0：For Z axis move key, ↓is positive，↑is negative.
VALY =1：For Y axis move key,↑ is positive，↓is negative;
=0：For Y axis move key, ↓is positive，↑is negative.
VALX =1：For X axis move key, →is positive，←is negative;
=0：For X axis move key, ←is positive，→is negative

0	2	2		CALH	SOT	***	MZR5	MZR4	MZRZ	MZRY	MZRX

CALH =1：Length offset not cancelled in reference point return;
=0：Length offset cancelled in reference point return.
SOT =1：Software limit is valid after zero return at power on;
=0：Software limit is valid once power on.
MZR5 =1：Machine zero return in negative 5th axis;
=0：Machine zero return in positive 5th axis.
MZR4 =1：Machine zero return in negative 4th axis;
=0：Machine zero return in positive 4th axis.
MZRZ =1：Machine zero return in negative Z axis;
=0：Machine zero return in positive Z axis.
MZRY =1：Machine zero return in negative Y axis;
=0：Machine zero return in positive Y axis.
MZRX =1：Machine zero return in positive X axis;
=0：Machine zero return in negative X axis.

0	2	5		RTORI	***	RTPCP	***	***	RTCRG	***	***

RTORI=1：Spindle performs zero return when M29 is executed;
=0：Spindle does not perform zero return when M29 is executed.

Volume III Installation

RTPCP=1：Rigid tapping is the high-speed deep hole cycle(G73 mode);

=0：Rigid tapping is the high-speed deep hole cycle (G83 mode).

RTCRG=1：Do not wait for G61.0 to be 1 as excuting next program block after rigid tapping cancelled;

=0：Do wait for G61.0 to be 1 as excuting next program block after rigid tapping cancelled.

0	2	6	A4IS1	A4IS0	***	RCS4	***	***	ROS4	ROT4

RCS4 =1：4th Cs function is valid(power on);

=0：4th Cs function is invalid(power on).

Note: Only when the rotary axis function is valid (ROT4=1), can the RCS4 be set valid.

ROS4, ROT4：Set the type of 4th;

	Linear	**Rotary A**	**Rotary B**	**invalid**
ROT4	0	1	1	0
ROS4	0	0	1	1

A4IS1, A4IS0:Selecte increment system of 4th.

A4IS1	**A4IS0**	**Increment System of 4TH**
0	0	Same to the X, Y, Z
0	1	IS-A
1	0	IS-B
1	1	IS-C

0	2	7	***	RRT4	***	***	***	RRL4	RAB4	ROA4

RRT4 =1：Zero mode D is used on 4th rotary axis (power on);

=0：Zero mode A,B,C are used on 4th rotary axis (power on).

RRL4 =1：4th rel.coor.cycle func.is valid (power on);

=0：4th rel.coor.cycle func.is invalid(power on).

RAB4 =1：4th rotates according to symbol direction;

=0：4th rotates according to nearby rotation.

ROA4 =1：4th abs.coor.cycle func.is valid (power on);

=0：4th abs.coor.cycle func.is invalid(power on).

Note 1: Parameter ROA4 is valid for only rotary axis (ROT4=1),

Note 2: Only parameter ROA4 =1, is RAB4 valid

Note 3: Only parameter ROA4 =1, is RRL4 valid

0	2	8	A5IS1	A5IS0	***	RCS5	***	***	ROS5	ROT5

RCS5 =1：5th Cs function is valid(power on);

=0：5th Cs function is invalid(power on).

Note: Only rotary axis function is valid (ROT5=1), is RCS5 valid.

ROS5, ROT5：Set the type of 5th;

	Linear	Rotary A	Rotary B	invalid
ROT5	0	1	1	0
ROS5	0	0	1	1

A5IS1, A5IS0: Selecte increment system of 5th..

A5IS1	A5IS0	Increment System of 5TH
0	0	Same to the X, Y, Z
0	1	IS-A
1	0	IS-B
1	1	IS-C

0	2	9

***	RRT5	***	***	***	RRL5	RAB5	ROA5

RRT5 =1：Zero mode D is used on 5th rotary axis (power on);
=0：Zero mode A,B,C are used on 5th rotary axis (power on).
RRL5 =1：5th rel.coor.cycle func.is valid (power on);
=0：5th rel.coor.cycle func.is invalid(power on).
RAB5 =1：5th rotates according to symbol direction;
=0：5th rotates according to nearby rotation.
ROA5 =1：5th abs.coor.cycle func.is valid (power on);
=0：5th abs.coor.cycle func.is invalid(power on).

Note1: ROA5 is valid to only rotary axis (ROT5=1);
Note2: Only when parameter ROA4 =1, is RAB4 valid;
Note3: Only when parameter ROA4 =1, is RRL4 valid;

0	3	8

ISC	***	***	***	***	***	***	***

ISC =1：Minimum increment system is IS-C(need restart);
=0：Minimum increment system is IS-B(do not need restart).

0	3	9

***	***	***	ABP5	ABP4	ABPZ	ABPY	ABPX

ABPx =1：Output axis pulse by two right-angle intersection phases(need restart);
=0：Output axis pulse by pulse and direction (do not need restart).

0	4	0

***	***	***	***	***	L2	L1	L0

L2, L1, L0：Interface language selection:

Language	L2	L1	L0
Chinese	0	0	0
English	0	0	1
Frence	0	1	0
Spanish	0	1	1
Germen	1	0	0
Italian	1	0	1
Russian	1	1	0
Korean	1	1	1

3.1.2 Data parameter

0	4	9	CMRX：X axis multiplier coefficient
0	5	0	CMRY：Y axis multiplier coefficient
0	5	1	CMRZ：Z axis multiplier coefficient
0	5	2	CMR4：4th axis multiplier coefficient
0	5	3	CMR5：5th axis multiplier coefficient

Setting range： 1～32767

0	5	4	CMDX：X axis frequency division coefficient
0	5	5	CMDY：Y axis frequency division coefficient
0	5	6	CMDZ：Z axis frequency division coefficient
0	5	7	CMD4：4th axis frequency division coefficient
0	5	8	CMD5：5th axis frequency division coefficient

Setting range： 1～32767

setting range： 1～32767

Electronic gear ratio formula： $$\frac{CMR}{CMD}=\frac{S\times 360}{\alpha\times L}\times\frac{Z_M}{Z_D}$$

S：min. command output unit

Z_M：belt wheel teeth of lead screw

α: motor rotation angle for a pulse

Z_D：Wheel teeth of motor belt

L：Screw lead

0	5	9	X axis max. rapid traverse speed
0	6	0	Y axis max. rapid traverse speed
0	6	1	Z axis max. rapid traverse speed
0	6	2	4th axis max. rapid traverse speed
0	6	3	5th axis max. rapid traverse speed

Setting range：10～99999999（Unit：mm/min）

0 6 4	Acceleration&deceleration time constant of X axis rapid traverse (ms)
0 6 5	Acceleration&deceleration time constant of Y axis rapid traverse (ms)
0 6 6	Acceleration&deceleration time constant of Z axis rapid traverse (ms)
0 6 7	Acceleration&deceleration time constant of 4th axis rapid traverse (ms)
0 6 8	Acceleration&deceleration time constant of 5th axis rapid traverse (ms)

Setting range：10～4000（Unit：ms）

0 6 9	Rapid traverse speed when rapid override is F0

Setting range：6～4000（Unit：mm/min）

0 7 0	Axes top feedrate of cutting

Setting range：10～4000（Unit：mm/min）

0 7 1	Exponential acceleration start speed and deceleration end speed in cutting feed

Setting range：0～8000（Unit：mm/min）

0 7 2	Exponential acceleration&deceleration time constant of cutting

Setting range：10～4000（Unit：ms）

0 7 3	Start speed in manual feed.

Setting range：0～8000（Unit：mm/min）

0 7 4	Exponential acceleration&deceleration time constant of manual feed

Setting range：10～4000（Unit：ms）

0 7 5	Threading axes start speed

Setting range：6～8000（Unit：mm/min）

0 7 7	Initial speed of acc.&dec.speed of CS axis

Setting range：0～5000（Unit：deg/min）

0 7 8	Acc.&dec.time constant of CS axis

Setting range：10～10000（Unit：ms）

0	8	1	Initial speed of linear acceleration/deceleration in rigid tapping

Setting range：0～5000（Unit：mm/min）

0	8	2	Linear acc.&dec. time constant in rigid tapping tool infeed

Setting range：10～10000（Unit：ms）

0	8	3	Linear acc.&dec. time constant in rigid tapping tool retract

Setting range：0～4000（Unit：ms）, 082 setting value is used when it is set to 0.

0	8	4	Override value in rigid tapping tool retract(0: override is set to 100%)

Setting range：0～200, 0: override is set to 100%

0	8	5	Tool retract amount in deep hole rigid tapping(high-speed, standard)

Setting range：0～32767000（Unit：0.001mm）

0	8	9	Low speed of X axis machine zero return
0	9	0	Low speed of Y axis machine zero return
0	9	1	Low speed of Z axis machine zero return
0	9	2	Low speed of 4th axis machine zero return
0	9	3	Low speed of 5th axis machine zero return

Setting range：10～1000（Unit：mm/min）

0	9	4	High speed of X axis machine zero return
0	9	5	High speed of Y axis machine zero return
0	9	6	High speed of Z axis machine zero return
0	9	7	High speed of 4th axis machine zero return
0	9	8	High speed of 5th axis machine zero return

Setting range：10～921571875（Unit：mm/min）

0	9	9	Voltage compensation for 0V analog voltage output

Setting range：-1000～1000（Unit：mV）

1	0	0	Voltage offset value when spindle max. speed analog voltage 10V output

Setting range：-2000～2000（Unit：mV）

1	0	1	Max spindle speed of 1st gear when analog voltage output is 10V
1	0	2	Max.spindle speed of 2nd gear when analog voltage output is 10V
1	0	3	Max.spindle speed of 3rd gear when analog voltage output is 10V
1	0	4	Max.spindle speed of 4th gear when analog voltage output is 10V

Setting range：10～9999（Unit：r/min）

1	0	7	Spindle speed resches to signal detection delay time

Setting range：0～4080（Unit：ms）

1	0	8	Max. spindle speed fluctuation allowed by system

Setting range：50～1000（Unit：r/min）

1	0	9	spindle encoder pulses

Setting range：0～5000（Unit：p/r）, It is drilling holes when 0 indicates G74 and G84 cycle.

1	1	0	Transmission ratio of encoder and spindle- spindle gear teeth
1	1	1	Transmission ratio of encoder and spindle- encoder gear teeth

Setting range：1～255

1	1	5	X axis backlash offset
1	1	6	Y axis backlash offset
1	1	7	Z axis backlash offset
1	1	8	4th axis backlash offset
1	1	9	5th axis backlash offset

Setting range：0～2000(Unit:0.001mm)

1	2	0	Interval of X axis screw-pitch error compensation
1	2	1	Interval of Y axis screw-pitch error compensation
1	2	2	Interval of Z axis screw-pitch error compensation
1	2	3	Interval of 4th axis screw-pitch error compensation
1	2	4	Interval of 5th axis screw-pitch error compensation

Setting range：10000～999999 (Unit:0.001mm)

1	2	5	Screw-pitch error compensation position number of X axis machine zero
1	2	6	Screw-pitch error compensation position number of Y axis machine zero
1	2	7	Screw-pitch error compensation position number of Z axis machine zero
1	2	8	Screw-pitch error compensation position number of 4th axis machine zero
1	2	9	Screw-pitch error compensation position number of 5th axis machine zero

Setting range：0～255

1	3	0	X axis machine zero offset
1	3	1	Y axis machine zero offset
1	3	2	Z axis machine zero offset
1	3	3	4th axis machine zero offset
1	3	4	5th axis machine zero offset

Setting range：-99999～99999 (Unit:0.001mm)

1	3	5	Max. X coordinate value of software limit
1	3	6	Max. Y coordinate value of software limit
1	3	7	Max. Z coordinate value of software limit
1	3	8	Max. 4th coordinate value of software limit
1	3	9	Max. 5th coordinate value of software limit
1	4	0	Min. X coordinate value of software limit
1	4	1	Min. Y coordinate value of software limit
1	4	2	Min. Z coordinate value of software limit
1	4	3	Min. 4th coordinate value of software limit
1	4	4	Min. 5th coordinate value of software limit

Setting range：-9999999～+9999999 (Unit:0.001mm)

1	4	5	X machine coordinate of 1st reference point
1	4	6	Y machine coordinate of 1st reference point
1	4	7	Z machine coordinate of 1st reference point
1	4	8	4th machine coordinate of 1st reference point
1	4	9	5th machine coordinate of 1st reference point
1	5	0	X machine coordinate of 2nd reference point
1	5	1	Y machine coordinate of 2nd reference point
1	5	2	Z machine coordinate of 2nd reference point
1	5	3	4th machine coordinate of 2nd reference point
1	5	4	5th machine coordinate of 2nd reference point
1	5	5	X machine coordinate of 3rd reference point
1	5	6	Y machine coordinate of 3rd reference point
1	5	7	Z machine coordinate of 3rd reference point
1	5	8	4th machine coordinate of 3rd reference point
1	5	9	5th machine coordinate of 3rd reference point
1	6	0	X machine coordinate of 4th reference point
1	6	1	Y machine coordinate of 4th reference point
1	6	2	Z machine coordinate of 4th reference point
1	6	3	4th machine coordinate of 4th reference point
1	6	4	5th machine coordinate of 4th reference point

Setting range：-9999999～+9999999 (Unit:0.001mm)

1	7	2	Initial value of cutting feedrate when power on

Setting range：10～15000 (Unit:mm/min)

1	7	4	Feedrate of dry run

Setting range：10～99999999 (Unit:mm/min)

1	7	5	Arc radius error limit

Setting range：0～1000 (Unit:0.001mm), On arc code （G02,G03）, if error exceeds the difference excuting limit between initial point radius and end point radius, alarm will be issued.

1	7	6	Retraction amount of G73 high deep hole drilling cycle

Setting range：0～32767000 (Unit:0.001mm),

1	7	7	Cutting initial point of G83 high deep hole drilling cycle

Setting range：0～32767000 (Unit:0.001mm),

1	7	8	G110,G111,G134,G135 Lead of helical tool infeed

Setting range：0～999999（unit 0.001mm）

If setting value is less than 10, helical feeding is invalid for rough milling command G110, G111, G134, G135, and it feeds by linear type.

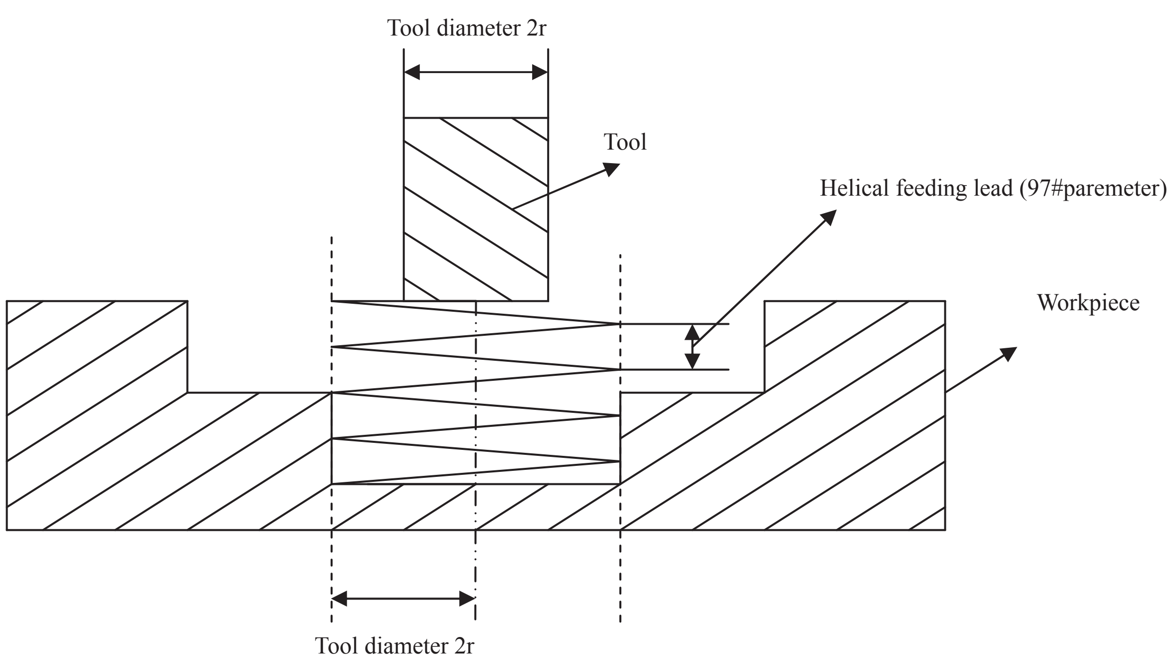
If setting value is more than or equal to 10, it feeds by helical type for rough milling command G110, G111, G134, G135.

Rough milling command（G110,G111,134,G135）helical feed function:

Namely, for Z axis depth cutting of rough milling command G110, G111, 134, G135, the tool feeds not by linear type, but by helical type. So the workpiece with no groove may be rough milled directedly.

Note 1 when the Z axis cutting depth is less than 10μm each time, the helical feeding is invalid.
Note 2 when the tool radius is less than 1mm, the helical feeding is also invalid.

The helical feeding path is shown in follows:



1	8	9	Movement per rotation of the 4th axis
1	9	0	Movement per rotation of the 5th axis

Setting range：1～9999999（unit：0.001deg）

2	0	1	Allowded valid ey number at the same time

Setting range：2～5

2	0	2	Define the name of the 4th axis(A:65, B:66, C:67)
2	0	3	Define the name of the 5th axis(A:65, B:66, C:67)

Setting range：65～67　65-A，66-B，67-C

2	1	3	Total tool number selection

Setting range：1～32

2	1	4	Reset output time

Setting range：16～4080（unit：ms）

2	1	5	Serial communication baudrate

Setting range：1200, 2400, 4800, 9600, 19200, 38400, 57600, 115200（unit：bit/s）

2	1	6	Block No. increment for block No.auto insertion

Setting range：1～100

3.2 Parameter description (by function sequence)

3.2.1 Axis control logic

0	0	8	DISP	***	***	**DIR5**	**DIR4**	**DIRZ**	**DIRY**	**DIRX**

DIR5 =1：Direction signal (DIR)is high level as the 5th axis moves positively;
=0：Direction signal (DIR)is low level as the 5th axis moves negatively.
DIR4 =1：Direction signal (DIR)is high level as the 4th axis moves positively;
=0：Direction signal (DIR)is low level as the 4th axis moves negatively.
DIRZ =1：Direction signal (DIR)is high level as Z axis moves positively;
=0：Direction signal (DIR)is low level as Z axis moves negatively.
DIRY =1：Direction signal (DIR)is high level as Y axis moves positively;
=0：Direction signal (DIR)is low level as Y axis moves negatively.
DIRX =1：Direction signal (DIR)is high level as X axis moves positively;
=0：Direction signal (DIR)is low level as X axis moves negatively.

0	0	9

***	***	***	**ALM5**	**ALM4**	**ALMZ**	**ALMY**	**ALMX**

ALM5 =1：the 5th axis low level alarm signal (ALM5);
=0：the 5th axis high level alarm signal (ALM5).
ALM4 =1：the 4th axis low level alarm signal (ALM4);
=0：the 4th axis high level alarm signal (ALM4).
ALMZ =1：Z axis low level alarm signal (ALMZ);
=0：Z axis high level alarm signal (ALMZ).
ALMY =1：Y axis low level alarm signal (ALMY);
=0：Y axis high level alarm signal (ALMY).
ALMX =1：X axis low level alarm signal (ALMX);
=0：X axis high level alarm signal (ALMX).

0	1	9

KEY1	***	***	**HNG5**	**HNG4**	**HNGZ**	**HNGY**	**HNGX**

HNG5 =1：the 5th MPG:ccw:+,cw:-;
=0：the 5th MPG:ccw:-,cw:+.
HNG4 =1：the 4th MPG:ccw:+,cw:-;
=0：the 4th MPG:ccw:-,cw:+.
HNGZ =1：Z MPG:ccw:+,cw:-;
=0：Z MPG:ccw:-,cw:+.
HNGY =1：Y MPG:ccw:+,cw:-;
=0：Y MPG:ccw:-,cw:+.
HNGX =1：X MPG:ccw:+,cw:-;
=0：X MPG:ccw:-,cw:+.

0	2	0

SPFD	SAR	THDA	**VAL5**	**VAL4**	**VALZ**	**VALY**	**VALX**

VAL5 =1：For the 5th axis move key,↑ is positive，↓is negative;
=0：For the 5th axis move key, ↓is positive，↑is negative.
VAL4 =1：For the 4th axis move key,↑ is positive，↓is negative;
=0：For the 4th axis move key, ↓is positive，↑is negative.
VALZ =1：For Z axis move key,↑ is positive，↓is negative;
=0：For Z axis move key, ↓is positive，↑is negative.
VALY =1：For Y axis move key,↑ is positive，↓is negative;
=0：For Y axis move key, ↓is positive，↑is negative.
VALX =1：For X axis move key, →is positive，←is negative;
=0：For X axis move key, ←is positive，→is negative

Volume Ⅲ Installation

0	4	9	CMRX：X axis multiplier coefficient
0	5	0	CMRY：Y axis multiplier coefficient
0	5	1	CMRZ：Z axis multiplier coefficient
0	5	2	CMR4：4th axis multiplier coefficient
0	5	3	CMR5：5th axis multiplier coefficient

Setting range: 1～32767

0	5	4	CMDX：X axis frequency division coefficient
0	5	5	CMDY：Y axis frequency division coefficient
0	5	6	CMDZ：Z axis frequency division coefficient
0	5	7	CMD4：4th axis frequency division coefficient
0	5	8	CMD5：5th axis frequency division coefficient

Setting range： 1～32767

Electronic gear ratio formula：$\frac{CMR}{CMD}=\frac{S\times 360}{\alpha \times L}\times\frac{Z_M}{Z_D}$

S：Min. command output unit

Z_M：belt wheel teeth of lead screw

α: motor rotation angle for a pulse

Z_D：Wheel teeth of motor belt

L：Screw lead

Volume Ⅲ Installation

3.2.2 Acceleration & deceleration control

0	0	4	***	**RDRN**	DECI	***	PROD	***	***	SCW

RDRN =1：G00 rapid traverse， speed = federate ×dry run speed;

=0：G00 speed = rapid override × rapid tranverse speed .

0	1	2	***	***	***	TMANL	***	***	EBCL	**ISOT**

ISOT =1：Prior to machine zero return after power on, manual rapid traverse valid;

=0：Prior to machine zero return after power on, manual rapid traverse invalid.

0	5	9	X axis max. rapid traverse speed
0	6	0	Y axis max. rapid traverse speed
0	6	1	Z axis max. rapid traverse speed
0	6	2	4th axis max. rapid traverse speed
0	6	3	5th axis max. rapid traverse speed

Setting range:10～1843143750（unit：mm/min）

0	6	4	Acceleration&deceleration time constant of X axis rapid traverse (ms)
0	6	5	Acceleration&deceleration time constant of Y axis rapid traverse (ms)
0	6	6	Acceleration&deceleration time constant of Z axis rapid traverse (ms)
0	6	7	Acceleration&deceleration time constant of 4th axis rapid traverse (ms)
0	6	8	Acceleration&deceleration time constant of 5th axis rapid traverse (ms)

Setting range:10～4000(unit：ms)

0	6	9	Rapid traverse speed when rapid override is F0

Setting range:6～4000（unit：mm/min）

0	7	0	Axes top feedrate of cutting

Setting range:10～15000（unit:mm/min）

0	7	1	Exponential acceleration start speed and deceleration end speed in cutting feed

Setting range:0～8000（unit:mm/min）

0	7	2	Exponential acceleration&deceleration time constant of cutting

Setting range:10～4000（unit：ms）

0	7	3	Start speed in manual feed.

Setting range:0～8000（unit:mm/min）

0	7	4	Exponential acceleration&deceleration time constant of manual feed

Setting range:10～4000（unit：ms）

3.2.3 Machine protection

0	1	7	***	**MST**	**MSP**	**MOT**	**MESP**	***	***	***

MST =1：External cycle start signal (ST) invalid，

=0：External cycle start signal (ST) valid.

MSP =1：External stop signal (SP) invalid，

=0：External stop signal (SP) valid with external stop switch connected, otherwise CNC shows “stop” .

MOT =1：Not detect software stroke limit;

=0：Detect software stroke limit.

MESP =1：Emergency stop invalid;

=0：Emergency stop valid

0	1	8	***	***	***	**ESCD**	***	***	***	***

ESCD =1：S code off at emergency stop;

=0：S code not off at emergency stop

Volume III Installation

0	2	2		CALH	**SOT**	***	MZR5	MZR4	MZRZ	MZRY	MZRX

SOT =1：Software limit valid after zero return at power on;

=0：Software limit valid after power on.

1	3	5	Max. X coordinate value of software limit
1	3	6	Max. Y coordinate value of software limit
1	3	7	Max. Z coordinate value of software limit
1	3	8	Max. 4th coordinate value of software limit
1	3	9	Max. 5th coordinate value of software limit
1	4	0	Min. X coordinate value of software limit
1	4	1	Min. Y coordinate value of software limit
1	4	2	Min.Z coordinate value of software limit
1	4	3	Min. 4th coordinate value of software limit
1	4	4	Min. 5th coordinate value of software limit

Setting range：-9999999～+9999999（unit：0.001mm）

3.2.4 Thread function

0	2	0		SPFD	SAR	**THDA**	VAL5	VAL4	VALZ	VALY	VALX

THDA =1：Threading machining adopts exponential acceleration and deceleration;

=0：Threading machining adopts linear acceleration and deceleration.

0	7	5	Threading axes start speed

Setting range：6～8000（unit:mm/min）

3.2.5 Spindle control

0	0	1		***	***	***	**ACS**	HWL	***	***	***

ACS =1: Analog voltage control of spindle speed;

=0: Switching control of spindle speed.

0	9	9	Voltage compensation for 0V analog voltage output

Setting range：-1000～1000 （unit:mV）

1	0	0	Voltage offset value when spindle max. speed analog voltage 10V output

Setting range：-2000～2000（unit：mV）

1 0 1		Max spindle speed of 1st gear when analog voltage output is 10V
1 0 2		Max.spindle speed of 2nd gear when analog voltage output is 10V
1 0 3		Max.spindle speed of 3rd gear when analog voltage output is 10V
1 0 4		Max.spindle speed of 4th gear when analog voltage output is 10V

Setting range：10～9999 （unit:r/min）

1 0 7	Delay of spindle speed in-position signal detection

Setting range：0～4080 （unit:ms）

1 0 8	Max. spindle speed fluctuation allowed by system

Setting range：50～1000（unit:r/min）

1 0 9	spindle encoder pulses/rev

Setting range：0～5000 （unit：p/r）0: Not detect spindle encoder in G74, G84 tapping.

1 1 0	Transmission ratio of encoder and - spindle gear teeth
1 1 1	Transmission ratio of encoder and - encoder gear teeth

Setting range:1～255

3.2.6 Tool function

0 0 2	***	***	***	**LIFJ**	**MDITL**	**LIFC**	**NRC**	**TLIF**

LIFJ =1: Tool life management group skip valid;
=0: Tool life management group skip invalid.
MDITL =1: Tool life management valid in MDI mode;
=0: Tool life management invalid in MDI mode.
LIFC =1: Tool life counting type 2 by times;
=0: Tool life counting type 1 by times.
NRC =1: Tool nose radius compensation valid;
=0: Tool nose radius compensation invalid.
TLIF =1: Tool life management valid;
=0: Tool life management invalid

0 1 2	***	***	***	**TMANL**	***	***	EBCL	ISOT

TMANL =1：Manual tool change for T code;
=0：Auto tool change for T code.

2 1 3	Total tool number selection

Setting range：1～32

Volume Ⅲ Installation

3.2.7 Edit and Display

0	0	4		***	RDRN	DECI	***	**PROD**	***	***	SCW

PROD =1：Relative coordinate displayed in POSITION page is programming position;
=0：Relative coordinate displayed in POSITION page is position involving tool offset.

0	0	8		**DISP**	***	***	DIR5	DIR4	DIRZ	DIRY	DIRX

DISP =1：Enter absolute page after power on;
=0：Enter relative page after power on.

0	1	2		***	***	***	TMANL	***	***	**EBCL**	ISOT

EBCL =1：Program end sign EOB displays “;”(semicolon);
=0：Program end sign EOB displays “*”(asterisk).

0	4	0		***	***	***	***	***	**L2**	**L1**	**L0**

L2, L1, L0：Interface language selection;

Language	L2	L1	L0
Chinese	0	0	0
English	0	0	1
Frence	0	1	0
Spanish	0	1	1
Germen	1	0	0
Italy	1	0	1
Russian	1	1	0
Korean	1	1	1

2	1	6		Block No. increment for block No.auto insertion

Setting range：1～100

3.2.8 Precision compensation

0	0	3		***	***	**PCOMP**	***	***	***	**D/R**	***

PCOMP =1: Screw-pitch error compensation valid;
=0: Screw-pitch error compensation invalid.
D/R =1: Tool offset D value is diameter input;
=0: Tool offset D value is radius input.

0	1	0		**CPF7**	**CPF6**	**CPF5**	**CPF4**	**CPF3**	**CPF2**	**CPF1**	**CPF0**

CPF0～CPF7： Setting values of backlash compensation pulse frequency.
The set frequency =
（2^7×CPF7+2^6×CPF6+2^5×CPF5+2^4×CPF4+2^3×CPF3+2^2×CPF2+2^1×CPF1+CPF0）Kpps

0	1	1		BDEC	BD8	***	***	***	ZNIK	***	***

BDEC =1：Backlash compensation type B, the compensation data are output by ascending or decending type and the set frequency is invalid.;

=0：Backlash compensation type A, the compensation data are output by the set frequency (set by bit parameter No.010) or 1/8 of it.

BD8 =1：Backlash compensation is done by the 1/8 of the set frequency;

=0：Backlash compensation is done by the set frequency.

0	2	2		CALH	SOT	***	MZR5	MZR4	MZRZ	MZRY	MZRX

CALH =1：Length offset not cancel in reference point return;

=0：Length offset cancel in reference point return.

1	1	5	X axis backlash offset
1	1	6	Y axis backlash offset
1	1	7	Z axis backlash offset
1	1	8	4th axis backlash offset
1	1	9	5th axis backlash offset

Setting range：0～2000（unit:0.001mm）

1	2	0	Interval of X axis screw-pitch error compensation
1	2	1	Interval of Y axis screw-pitch error compensation
1	2	2	Interval of Z axis screw-pitch error compensation
1	2	3	Interval of 4th axis screw-pitch error compensation
1	2	4	Interval of 5th axis screw-pitch error compensation

Setting range： 1000～999999（unit：0.001mm ）

1	2	5	Screw-pitch error compensation number of X axis machine zero
1	2	6	Screw-pitch error compensation number of Y axis machine zero
1	2	7	Screw-pitch error compensation number of Z axis machine zero
1	2	8	Screw-pitch error compensation number of the 4th axis machine zero
1	2	9	Screw-pitch error compensation number of the 5th axis machine zero

Setting range： 0～255

3.2.9 Communication setting

2	1	5		Serial communication baudrate

Setting range：1200, 2400, 4800, 9600, 19200, 38400, 57600, 115200 (unit:bit/s)

Volume Ⅲ Installation

3.2.10 Machine zero return

0	0	4		***	RDRN	**DECI**	***	PROD	***	***	SCW

DECI =1：Deceleration signal high level for machine zero return;
=0：Deceleration signal low level for machine zero return.

0	1	1		BDEC	BD8	***	***	***	**ZNIK**	***	***

ZNIK =1：Direction keys locked during zero return, homing continues to end by pressing direction key once;
=0：Direction keys unlocked but should be held on during zero return

0	0	6		***	***	***	**ZM5**	**ZM4**	**ZMZ**	**ZMY**	**ZMX**

ZM5 =1：5th zero return type C;
=0：5th zero return type B.
ZM4 =1：4th zero return type C;
=0：4th zero return type B.
ZMZ =1：Z zero return type C;
=0：Z zero return type B.
ZMY =1：Y zero return type C;
=0：Y zero return type B.
ZMX =1：X zero return type C;
=0：X zero return type B.

0	0	7		AVGL	***	SMZ	**ZC5**	**ZC4**	**ZCZ**	**ZCY**	**ZCX**

ZC5 =1：The deceleration signal (DEC5) and one-rotation signal (PC5) of 5th axis in parallel connection (a proximity switch acting as both the deceleration signal and zero signal) during machine zero return;
=0：The deceleration signal (DEC5) and one-rotation signal (PC5) of 5th axis are connected independently (the indepent deceleration signal and zero signal are required) during machine zero return.

ZC4 =1：The deceleration signal (DEC4) and one-rotation signal (PC4) of 4th axis in parallel connection (a proximity switch acting as both the deceleration signal and zero signal) during machine zero return;
=0：The deceleration signal (DEC4) and one-rotation signal (PC4) of 4th axis are connected independently (the indepent deceleration signal and zero signal are required) during machine zero return.

ZCZ =1：The deceleration signal (DECZ) and one-rotation signal (PCZ) of Z axis in parallel connection (a proximity switch acting as both the deceleration signal and zero signal)

Volume III Installation

during machine zero return;

=0：The deceleration signal DECZ) and one-rotation signal (PCZ) of Z axis are connected independently (the indepent deceleration signal and zero signal are required) during machine zero return.

ZCY =1：The deceleration signal (DECY) and one-rotation signal (PCY) of Y axis in parallel connection (a proximity switch acting as both the deceleration signal and zero signal) during machine zero return;

=0：The deceleration signal (DECY)and one-rotation signal PCY) of Y axis are connected independently (the indepent deceleration signal and zero signal are required) during machine zero return.

ZCX =1：The deceleration signal (DECX) and one-rotation signal (PCX) of X axis in parallel connection (a proximity switch acting as both the deceleration signal and zero signal) during machine zero return;

=0：The deceleration signal (DECX) and one-rotation signal (PCX) of X axis are connected independently (the indepent deceleration signal and zero signal are required) during machine zero return.

0	1	4

***	***	***	ZRS5	ZRS4	ZRSZ	ZRSY	ZRSX

ZRS5 =1: There are machine zero point in the 5th axis, it detects deceleration signal and zero signal when performing machine zero return;

=0: There are no machine zero point in the 5th axis, it returns to machine zero without detecting deceleration signal and zero signal when performing machine zero return.

ZRS4 =1: There are machine zero point in the 4th axis, it detects deceleration signal and zero signal when performing machine zero return;

=0: There are no machine zero point in the 4th axis, it returns to machine zero without detecting deceleration signal and zero signal when performing machine zero return.

ZRSZ =1: There are machine zero point in Z axis, it detects deceleration signal and zero signal when performing machine zero return;

=0: There are no machine zero point in Z axis, it returns to machine zero without detecting deceleration signal and zero signal when performing machine zero return.

ZRSY =1: There are machine zero point in Y axis, it detects deceleration signal and zero signal when performing machine zero return;

=0: There are no machine zero point in Y axis, it returns to machine zero without detecting deceleration signal and zero signal when performing machine zero return.

ZRSX =1: There are machine zero point in X axis, it detects deceleration signal and zero signal when performing machine zero return;

=0: There are no machine zero point in X axis, it returns to machine zero without detecting deceleration signal and zero signal when performing machine zero return.

0	2	2		CALH	SOT	***	MZR5	MZR4	MZRZ	MZRY	MZRX

CALH =1：Length offset not cancel in reference point return;

=0：Length offset cancel in reference point return.

MZR5 =1：Machine zero return in negative the 5th axis;

=0：Machine zero return in positive the 5th axis.

MZR4 =1：Machine zero return in negative the 4th axis;

=0：Machine zero return in positive the 4th axis.

MZRZ =1：Machine zero return in negative Z axis;

=0：Machine zero return in positive Z axis.

MZRY =1：Machine zero return in negative Y axis;

=0：Machine zero return in positive Y axis.

MZRX =1：Machine zero return in positive X axis;

=0：Machine zero return in negative X axis.

0	8	9	Low speed of X axis machine zero return
0	9	0	Low speed of Y axis machine zero return
0	9	1	Low speed of Z axis machine zero return
0	9	2	Low speed of the 4th axis machine zero return
0	9	3	Low speed of the 5th axis machine zero return

Setting range：10～1000（unit：mm/min）

0	9	4	High speed of X axis machine zero return
0	9	5	High speed of Y axis machine zero return
0	9	6	High speed of Z axis machine zero return
0	9	7	High speed of the 4th axis machine zero return
0	9	8	High speed of the 5th axis machine zero return

Setting range：10～921571875 (unit:mm/min)

1	3	0	X axis machine zero offset
1	3	1	Y axis machine zero offset
1	3	2	Z axis machine zero offset
1	3	3	The 4th axis machine zero offset
1	3	4	The 5th axis machine zero offset

Setting range：-99999～99999(unit：0.001mm)

Volume III Installation

			Description
1	4	5	X machine coordinate of the 1st reference point
1	4	6	Y machine coordinate of the 1st reference point
1	4	7	Z machine coordinate of the 1st reference point
1	4	8	The 4th machine coordinate of the 1st reference point
1	4	9	The 5th machine coordinate of the 1st reference point
1	5	0	X machine coordinate of the 2nd reference point
1	5	1	Y machine coordinate of the 2nd reference point
1	5	2	Z machine coordinate of the 2nd reference point
1	5	3	The 4th machine coordinate of the 2nd reference point
1	5	4	The 5th machine coordinate of the 2nd reference point
1	5	5	X machine coordinate of the 3rd reference point
1	5	6	Y machine coordinate of the 3rd reference point
1	5	7	Z machine coordinate of the 3rd reference point
1	5	8	The 4th machine coordinate of the 3rd reference point
1	5	9	The 5th machine coordinate of the 3rd reference point
1	6	0	X machine coordinate of the 4th reference point
1	6	1	Y machine coordinate of the 4th reference point
1	6	2	Z machine coordinate of the 4th reference point
1	6	3	The 4th machine coordinate of the 4th reference point
1	6	4	The 5th machine coordinate of the 4th reference point

Setting range：-99999999～99999999 (unit:0.001mm)

3.2.11 Rotary axis function

0	2	5	**RTORI**	***	**RTPCP**	***	***	**RTCRG**	***	***

RTORI =1：M29 is executed,Spindle need to return zero;

=0：M29 is executed,Spindle need not to return zero.

RTPCP =1：Rigid tapping is the high-speed deep hole cycle(G73);

=0：Rigid tapping is the high-speed deep hole cycle (G83).

RTCRG =1：Do not wait for G61.0 to be 1 as excuting next program block after rigid tapping cancelled;

=0：Do wait for G61.0 to be 1 as excuting next program block after rigid tapping cancelled.

0	2	6	***	***	***	**RCS4**	***	***	**ROS4**	**ROT4**

RCS4 =1：Cs function of 4th axis is valid(power on);

=0：Cs function of 4th axis is invalid(power on).

ROS4, ROT4：Set the type of 4th axis;

	Linear	Rotary A	Rotary B	invalid
ROT4	0	1	1	0
ROS4	0	0	1	1

0	2	7	***	**RRT4**	***	***	***	**RRL4**	**RAB4**	**ROA4**

RRT4 =1：Zero mode D is used on the 4th rotary axis (power on);

=0：Zero mode A,B,C are used on the 4th rotary axis (power on).

RRL4 =1：the 4th rel.coor.cycle func.is valid (power on);

=0：the 4th rel.coor.cycle func.is invalid(power on).

RAB4 =1：the 4th rotates according to symbol direction;

=0：the 4th rotates according to nearby rotation.

ROA4 =1：the 4th abs.coor.cycle func.is valid (power on);

=0：the 4th abs.coor.cycle func.is invalid(power on).

0	2	8	***	***	***	**RCS5**	***	***	**ROS5**	**ROT5**

RCS5 =1：Cs function of the 5th axis is valid(power on);

=0：Cs function of the 5th axis is invalid(power on).

ROS5, ROT5：Set the type of 5th;

	Linear	Rotary A	Rotary B	invalid
ROT5	0	1	1	0
ROS5	0	0	1	1

0	2	9	***	**RRT5**	***	***	***	**RRL5**	**RAB5**	**ROA5**

RRT5 =1：Zero mode D of the 5th axis (power on) ;

=0：Zero mode A, B, C of the 5th axis (power on) .

RRL5 =1：the 5th rel.coor.cycle func.is valid (power on);

=0：the 5th rel.coor.cycle func.is invalid(power on).

RAB5 =1：the 5th rotation according to symbol direction;

=0：the 5th rotation according to nearby direction.

ROA5 =1：the 5th abs.coor.cycle func.is valid (power on);

=0：the 5th abs.coor.cycle func.is invalid(power on).

RRT4 =1：Zero mode D is used on the 5th rotary axis (power on);

=0：Zero mode A,B,C are used on the 5th rotary axis (power on).

RRL4 =1：the 5th rel.coor.cycle func.is valid (power on);

=0：the 5th rel.coor.cycle func.is invalid(power on).

RAB4 =1：5th rotates according to symbol direction;

=0：5th rotates according to nearby rotation.

ROA4 =1：the 5th abs.coor.cycle func.is valid (power on);

=0：the 5th abs.coor.cycle func.is invalid(power on).

0 7 7	Initial speed of acc.&dec in using CS funciton

Setting range： 0～5000（Unit:deg/min）

0 7 8	Acc.&dec.time constant in using CS function

Setting range： 10～10000（Unit:ms）

0 8 1	Initial speed of linear acceleration/deceleration in rigid tapping

Setting range： 0～5000（Unit:mm/min）

0 8 2	Linear time constant in rigid tapping tool infeed

Setting range： 10～10000（Unit:ms）

0 8 3	Time constant in rigid tapping tool retract

Setting range： 0～4000（Unit:ms）, 082 setting value is used when it is set to 0.

0 8 4	Override value in rigid tapping tool retract(0: override is set to 100%)

Setting range： 0～200, 0: override is set to 100%

0 8 5	Tool retract amount in deep hole rigid tapping(high-speed, standard)

Setting range：0～32767000,（Unit:0.001mm）

1 8 9	One-rotaton increment of the 4th axis
1 9 0	One-rotaton increment of 5th axis

Setting range：1～9999999,（Unit:0.001deg）

2 0 1	Amount of valid keys pressed simultaneously

Setting range：2～5

2 0 2	Define the name of the 4th axis (A:65, B:66, C:67)
2 0 3	Define the name of the 5th axis (A:65, B:66, C:67)

Setting range：65～67 65-A，66-B，67-C

Volume III Installation

CHAPTER 4 MACHINE DEBUGGING METHODS AND STEPS

The trial run methods and steps at initial power on for this GSK980MDa are described in this chapter. The corresponding operation can be performed after the debugging by the following steps.

4.1 Emergency Stop and Stroke Limit

This GSK980MDa system has software limit function, it is suggested that the stroke limit switches are fixed in the positive or negative axes for hardware limit. The connection is shown in follows:（The chart is designed for X, Y, Z axes）

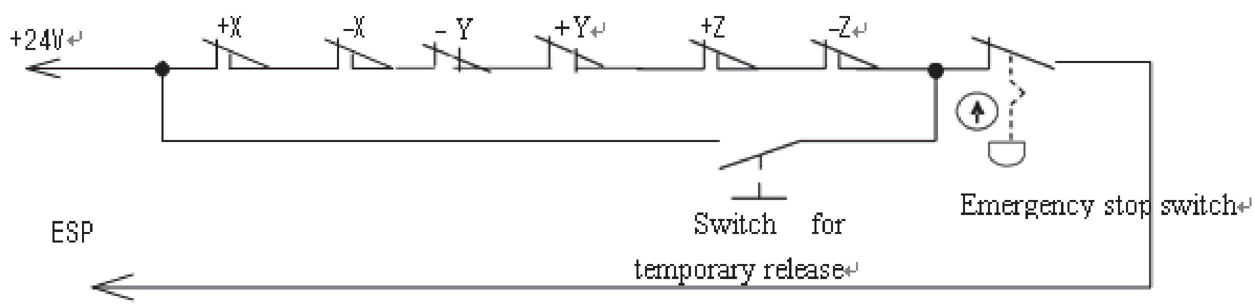


Fig.4-1

So the MESP of bit parameter No.17should be set to 0.

And the CNC diagnostic message ESP can monitor the state of emergency stop input signal.

In Manual or MPG mode, slowly move the axes to test the validity of stroke limit switch, correctness of alarm display, validity of overtravel release button.When the overtravel occurs or Emergency Stop button is pressed,“emergency stop” alarm will be issued by CNC system. The alarm can be cancelled by pressing down the Overtravel button and moving reversely.

4.2 Drive unit Unit Setting

Set BIT4～BIT0 of bit parameter No.009 according to alarm logic level of drive unit. The BIT4～BIT0 of bit parameter No.009 for our drive unit are all set for 1 .

If the machine moving direction is not consistent with the moving command, modify the BIT4～BIT0 of bit parameter No.008，BIT4～BIT0 of bit parameter No.019, BIT4～BIT0 of bit parameter No.20.

4.3 Gear Ratio Adjustment

The data parameter No.049～No.058 can be modified for electronic gear ratio adjustment to meet the different mechanical transmission ratio if the machine travel distance is not consistent with the displacement distance displayed by the CNC coordinate.

Calculation formula:

$$\frac{CMR}{CMD} = \frac{\delta \times 360}{\alpha \times L} \times \frac{Z_M}{Z_D}$$

CMR: command multiplier coefficient (data parameter №049, №050, №051, №052, №053)

CMD: command frequency division coefficient (data parameter №054, №055, №056, №057, №058)

α :: pulse volume, motor rotation angle for a pulse

L: lead

δ: min. input command unit of CNC (0.0001 for all axes of GSK980MDa)

ZM: gear teeth of lead screw

ZD: gear teeth of motor

If the electronic gear ratio numerator is greater than the denominator, the allowed CNC max. speed will decrease. For example: the data parameter No.051（CMRZ）=2，№056（CMDZ）=1, so the allowed Z axis max. speed is 8000mm/min.

If the electronic gear ratio numerator is not equal to the denominator, the allowed CNC positioning precision will decrease. For example: when the data parameter No.051（CMRZ）=1 and №056（CMDZ）=5, the pulse is not output as the input increment is 0.004, but a pulse is output if the input increment is up to 0.005.

In order to ensure the CNC positioning precision, speed index and match with digit servo with electronic gear ratio function, it is suggested that the CNC electronic gear ratio is set for 1:1 or the electronic gear ratio calculated is set to the digital servo.

When matching with the step drive, choose the drive unit with step division function as far as possible, and properly select mechanical transmission ratio. The 1:1 electronic gear ratio should be ensured to avoid the too large difference between the numerator and the denominator of this CNC gear ratio.

Example:

Match GSK980MDa with DA98B, take X axis for example: set command multiplier coefficient and command frequency division coefficient to 1. Calculation formula is shown below.

CNC：

$$\frac{CMR}{CMD} = \frac{\delta \times 360}{\alpha \times L} \times \frac{Z_M}{Z_D} = \frac{1}{1}$$

The following conclusions can be reached:

$$\alpha = \frac{\delta \times 360}{L} \times \frac{Z_M}{Z_D} \text{ (deg/pulse)}$$

Drive unit：

Parameters 12, 13 of drive unit correspond to position command pulse frequency division molecule and denominator. Calculation formula of drive unit gear ratio is shown as follows:

$$P \times G = 4 \times N \times C$$

P：Correspondence between required pulse volume for motor rotates 3600 and CNC end：$P = 360/\alpha$

G： Electronic gear ratio of drive unit, G= position command pulse frequency division molecule/ position command pulse frequency division denominator

N： Set motor rev number to 1

C： Wire number of feedback encoder: DA98B is 2500p/r.

The following conclusions can be reached:

$$G = \frac{4 \times N \times C}{P} = 4 \times N \times C \times \frac{\alpha}{360} = \frac{4 \times N \times C}{360} \times \frac{\delta \times 360}{L} \times \frac{Z_M}{Z_D} =$$

$$= \frac{10 \times Z_M}{L \times Z_D}$$

Set molecule and denominator of caculated ratio to drive unit 12, 13 separately.

4.4 Acceleration&deceleration Characteristic Adjustment

Adjust the relative CNC parameters according to the factors such as the drive unit, motor characteristics and machine load:

Data parameter №059～№063：X, Y, Z, 4th, 5th axis rapid traverse rate;

Data parameter №064～№068: linear acceleration & deceleration time constant of X, Y, Z, 4th, 5th axis rapid traverse rate;

Data parameter №069: rapid traverse speed when rapid override is F0

Data parameter №070: upper limit of axes cutting feedrate;

Data parameter №071: Start/end speed of exponential acceleration & deceleration in cutting feeding;

Data parameter №072: Exponential acceleration & deceleration time constant of cutting feeding;

Data parameter№073：Start/end speed of exponential acceleration & deceleration in MPG/Step feedrate;

Data parameter№074 ： Exponential acceleration & deceleration time constant of MPG/STEP/manual feed;

Data parameter№075：Start/end speed in thread cutting of each ax;

Data parameter№077：Initial feedrate of acc.&dec in CS axis;

Data parameter№078：Acc.&dec.time constant in CS axis;

Data parameter№081：Initial speed of linear acceleration/deceleration in rigid tapping;

Data parameter№082：Linear acceleration/deceleration time constant in rigid tapping tool infeed;

Data parameter№083：Linear acceleration/deceleration time constant in rigid tapping tool retraction;

Data parameter№084：Override value in rigid tapping tool retract;

Data parameter№172：Initial feedrate when power on;

Data parameter№174：Feedrate of DRY run;

SMZ of bit parameter №007: for validity of smoothing transition between blocks

The larger the acceleration&deceleration time constant is, the slower tacceleration&deceleration is, the smaller the machine movement impact and the lower the machining efficiency is.And vice versa.

If acceleration&deceleration time constants are equal, the higher the acceleration & deceleration start/end speed is, the faster the acceleration & deceleration is, the bigger the machine movement impact and the higher the machining efficiency is. And vice versa.

The principle for acceleration&deceleration characteristic adjustment is to properly reduce the acceleration & deceleration time constant and increase the acceleration&deceleration start/end speed to improve the machining efficiency on the condition that there is no alarm, motor out-of-step and obvious machine impact. If the acceleration&deceleration time constant is set too small, and the start/end speed is set too large, it is easily to cause drive unit alarm, motor out-of-step or machine vibration.

When the bit parameter №007 BIT3（SMZ）=1, the feedrate drops to the start speed of the acceleration&deceleration at the cutting path intersection, then it accelerates to the specified speed of the adjacent block to obtain an accurate positioning at the path intersection, but this will reduce the machining efficiency. When SMZ=0, the adjacent cutting path transits smoothly by the acceleration&deceleration. The feedrate does not always drop to the start speed when the previous path is finished and a circular transition (non-accurate positioning) will be formed at the path intersection. The machining surface by this path transition has a good finish and a higher machining efficiency. When the stepper motor drive unit is applied, the SMZ of the bit parameter №007 should be set to 1 to avoid the out-of-step.

When the stepper motor drive unit is applied to this system, the out-of-step may occur if rapid traverse speed is too large, acceleration&deceleration time constant is too small, acceleration&deceleration start/end speed is too large. The suggested parameter setting is shown in follows (the electronic gear ratio is 1:1):

Data parameter №059~№063≤5000 Data parameter №064~№068≥350 Data parameter №071≤50

Data parameter №072≥150 Data parameter №073≤50 Data parameter№074≥150

Data parameter№075≤100

When AC servo motor drive unit is applied to this system, the machining efficiency can be improved by a larger start speed and smaller ACC&DEC time constant setting. If optimum ACC&DEC characteristics are required, the ACC&DEC time constant may be set to 0，which can be got by adjusting the AC servo ACC&DEC parameters. The suggested parameter settings are as follows (electronic gear ratio is 1:1).

Data parameter №059~№063 set higher properly

Data parameter №064~№068≤60

Data parameter №071≥50

Data parameter №072≤50

Data parameter №073≥50

Data parameter №074≤50

Data parameter №075≤500

The parameter settings above are recommended for use, refer to the actual conditions of the drive unit, motor characteristic and machine load for its proper setting.

Volume Ⅲ Installation

4.5 Machine Zero Adjustment

Adjust the relevant parameters based on the valid level of the connection signal, zero return type or direction applied:

（DECI）of the bit parameter №004: valid level of deceleration signal as machine zero return

(ZM5~ZMX) of the bit parameter №006: return and initial backlash direction of X, Y, Z，4th, 5th axes machine zeroes at deceleration.

(ZC5~ZCX) of the bit parameter №007: it is able to set whether an approach switch taken as both deceleration and zero signals when X, Y, Z, 4th, 5th axes return to machine zero point.

（ZNLK）of the bit parameter №011: for direction keys lock when performing zero return

(ZRS5~ZRSX) of the bit parameter №014: for deceleration and zero signals detection of X, Y, Z axes in machine zero return.

(MZR5~MZRX）of the bit parameter №22: for positive or negative zero turn of X, Y, Z, 4th, 5th axes

Data parameter №089~№093: low speed of X, Y, Z, 4th, 5th axes in machine zero return

Data parameter №094~№098: high speed of X, Y, Z, 4th, 5th axes in machine zero return

RRT4 of bit parameter №027 and RRT5 of №029 set the machine zero return type of the 4th and the 5th axis separately.

Machine zero return can be done after the validity of overtravel limit swithch is confirmed.Machine zero return types A, B, C can be selected for basic axes (X, Y, Z). Machine zero return types A, B, C, D can be selected for additional axes (4th, 5th).

The machine zero is usually fixed at the max. travel point, and the effective stoke of the zero return touch block should be more than 25mm to ensure a sufficient deceleration distance for accurate zero return. The more rapid the machine zero return is, the longer the zero return touch block should be. Or the moving carriage will rush through the block which may influence the zero return precision because of the insufficient deceleration distance.

Usually there are 2 types of machine zero return connection:

1 The connection to AC servo motor: schematic diagram of using a travel switch and a servo motor one-rotation signal separately

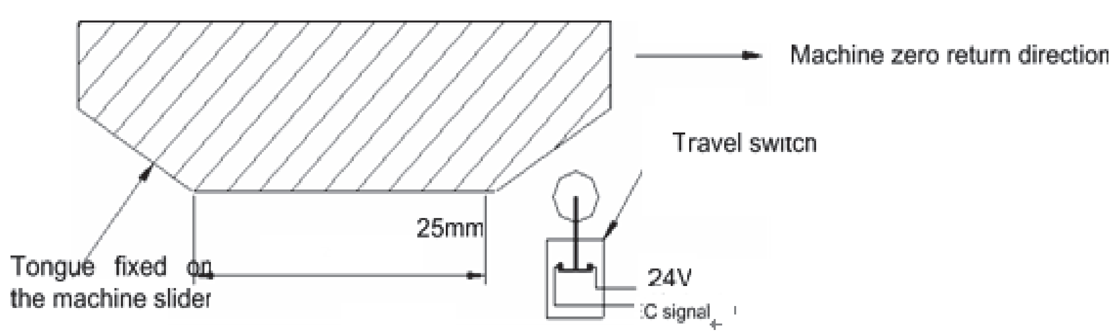


Fig. 4-2

Volume III Installation

By this connection type, when the deceleration switch is released in machine zero return, the one-rotation signal of encoder should be avoided to be at a critical point after the travel switch is released.In order to improve the zero return precision，it should be ensured the motor reaches the one-rotation signal of encoder after it rotates for half circle.And the moving distance for motor half circle rotation is the motor gear teeth/(2×lead screw gear teeth)

2 The connection to stepper motor: the schematic diagram of using a proximity switch taken as both deceleration signal and zero signal

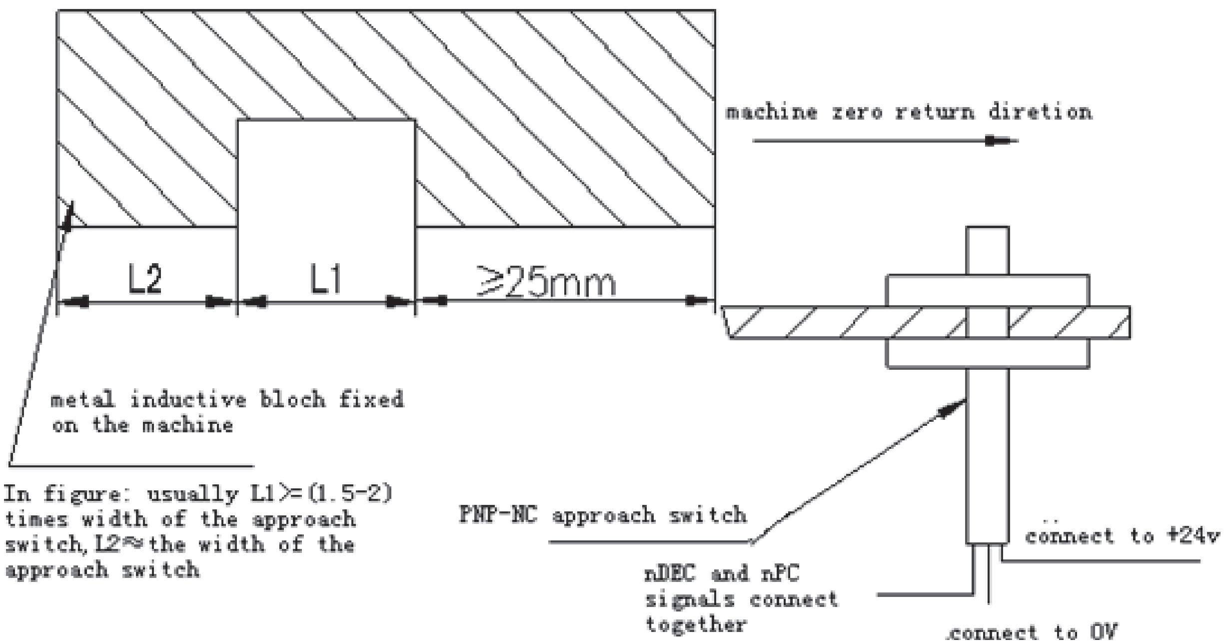


Fig 4-3

4.6 Spindle Adjustment

4.6.1 Spindle encoder

Encoder with the linear number 100~5000p/r is needed to be installed on the machine for threading. The linear number is set by data parameter No.109. The transmission ratio(spindle gear teeth/encoder gear teeth) between encoder and spindle is 1/255～255. The spindle gear teeth are set by CNC data parameter No.110, and the encoder gear teethare set by data parameter No.111. Synchronous belt transmission should be applied for it (no sliding transmission).

The DGN.011 and DNG.012 of CNC diagnosis messages are used to check the validity of threading signal from the spindle encoder.

4.6.2 Spindle brake

After spindle stop is executed, proper spindle brake time should be set to stop the spindle promptly in order to enhance the machining efficiency. If the brake is employed with energy consumption type, too long braking time may damage the motor. So the brake time is set by PLC.

Volume Ⅲ Installation