

GSK980MD MILLING CNC SYSTEM

User Manual



GSK

广州数控设备有限公司
GSK CNC EQUIPMENT CO., LTD.

Warning!

- Before installation, programming and operation, read this manual and the machine builder manual carefully and perform the relevant operations strictly by the instructions in the manuals. Or else it may result in product and machine damage, workpiece scrap, even serious personal hurt.

NOTE !

- The functions, technical specifications (such as precision, speed) introduced in this manual are only suitable for this product and those machines fixed with this product. The actual function configurations and technical performance are depended on the machine builder's design. And the CNC machine function configuration and technical specifications are defined by machine builder's manual.
- Though this system is employed with integrated operator panel, the functions of the keys on the panel are defined by PLC program (ladder). It should be noted that the keys functions are narrated for the standard PLC program in this manual.
- For the panel key functions and significance, please refer to the machine builder's manual.

The content of this manual is subject to change without further notice.

CAUTIONS

■ Transportation and Storage

- The product package box stacking should not exceed 6 layers.
- Don't crawl, stand or place heavy object on the product package box.
- Don't draw or move this product by the cables connected with it.
- Don't collide to or scratch panel and displayer.
- The product package box should be prevent from moisture, insolation and drenching.

■ Check by opening box

- Whether the product is the ordered one after opening the package.
- Whether the product is damaged during transiting.
- Whether the parts are complete or damaged by ordering sheet.
- If the product model doesn't correspond to the ordered one, subsidiaries are lacking or damaged in transiting, please contact us in time.

■ Wiring

- **Wiring and check should be done by the qualified technicians.**
- **This product must be securely grounded with a grounding resistance less than $4\ \Omega$, and the grounding wire can't be replaced by a neutral wire (zero wire)**
- **The wiring must be correct and secure to protect against product fault or unexpected result.**
- **The surge diode connected with this product must be joint by the specified direction or this product may be damaged.**
- **The power supply of this product must be cut off prior to plugging or opening the product cabinet.**

■ Reparation

- **Cut off the power supply before reparation or component replacement.**
- **If short-circuit or overloading occurs, check the fault first, then restart after the fault is eliminated.**
- **Don't switch on or off power frequently, the interval should be at least 1 minute for the repowering after power off.**

PART 1

PROGRAMMING

CHAPTER 1	PROGRAMMING	I -1
1.1	GSK980MD Brief	I -1
1.2	Execution of the Program	I -7
1.2.1	Order of the program execution	I -7
1.2.2	Execution order of command word within block.....	I -8
CHAPTER 2	MSTF COMMAND	II -1
2.1	M Command (Miscellaneous Function)	II -1
2.1.1	EP (End of program) M02.....	II -1
2.1.2	End-of-run M30	II -2
2.1.3	Subprogram call M98	II -2
2.1.4	Return from subprogram M99	II -2
2.1.5	Macro program call M9000~M9999.....	II -3
2.1.6	M command defined by standard PLC ladder diagram	II -4
2.1.7	Program stop M00.....	II -4
2.1.8	Spindle CCW, CW, stop control M03, M04 and M05	II -4
2.1.9	Coolant control M08, M09	II -4
2.1.10	Lubricant control M32, M33.....	II -5
2.2	Spindle Function	II -6
2.2.1	Spindle speed switching value control.....	II -6
2.2.2	Spindle speed analog voltage control.....	II -6
2.2.3	Spindle override	II -7
2.3	Tool Function	II -8
2.4	Feeding Function.....	II -8
2.4.1	Cutting feed (G94/G95, F command)	II -8
2.4.2	Manual feed	II -10
2.4.3	MPG/ Step feed	II -10
2.4.4	Automatic acceleration or deceleration	II -10
CHAPTER 3	G COMMAND.....	III-1
3.1	Brief	III-1
3.1.1	Modal, non-modal and initial.....	III-3
3.1.2	Examples.....	III-3
3.1.3	Related definition.....	III-3
3.1.4	Address definition	III-4
3.2	Rapid Positioning G00.....	III-6
3.3	Linear Interpolation G01.....	III-7
3.4	Arc and Helical Interpolation G02, G03	III-8
3.5	Dwell G04	III-12
3.6	Plane Selection Command G17, G18 and G19.....	III-13
3.7	Conversion of Inch and Metric G20 and G21	III-14
3.8	Reference Point Return G28	III-15
3.9	Return From Reference point G29	III-16
3.10	The 2nd, 3rd and 4th Reference Point Return G30.....	III-17
3.11	Skip Function G31	III-19
3.12	Tool Radius Compensation C (G40, G41 and G42)	III-21
3.13	Tool Length Compensation (G43, G44, G49).....	III-23

3.14	Workpiece Coordinate system G54~G59	III-26
3.15	Compound Cycle Command.....	III-28
3.15.1	Brief for canned cycle.....	III-28
3.15.2	Description for canned cycle	III-32
3.15.3	Cautions for canned cycle.....	III-53
3.15.4	Examples for modal data specified in canned cycle	III-55
3.15.5	Examples for canned cycle and tool length compensation	III-56
3.16	Absolute and Incremental Commands G90 and G91	III-58
3.17	Workpiece Coordinate System Setting G92	III-58
3.18	Feed per min. G94, Feed per rev. G95	III-59
3.19	G98、G99.....	III-60
3.20	Chamfering Function	III-60
3.20.1	Linear chamfering.....	III-60
3.20.2	Circular chamfering	III-62
3.20.3	Special.....	III-64
3.21	Macro Command	III-65
3.21.1	Macro Variable.....	III-65
3.21.2	Operation and transfer command G65.....	III-66
CHAPTER 4 CUTTER RADIUS COMPENSATION		IV-1
4.1	Application for Cutter Radius Compensation.....	IV-1
4.1.1	Brief	IV-1
4.1.2	Compensation value setting	IV-2
4.1.3	Command format	IV-2
4.1.4	Compensation direction	IV-2
4.1.5	Caution.....	IV-3
4.1.6	Example for application.....	IV-4
4.2	Offset Path Explanation for Cutter Radius Compensation	IV-5
4.2.1	Conception for inner side or outer side.....	IV-5
4.2.2	Tool movement in start-up.....	IV-5
4.2.3	Tool movement in offset mode	IV-7
4.2.4	Tool operation in offset cancellation mode.....	IV-12
4.2.5	Interference check.....	IV-13
4.2.6	Command of compensation vector cancel temporarily	IV-15
4.2.7	Exceptional case	IV-16

- ✓ Full screen parts program editing, 22MB program capacity.
- ✓ Parameter backup and data communication.
- ✓ Integrated multilingual display interface chosen by the parameter.
- ✓ Multilevel operation password function convenient for the equipment administration.
- ✓ Bidirectional communication between CNC and CNC, CNC and PC; the CNC software and the PLC program can be upgraded by communication.

The Technical Specification Table

Operation control	Controllable axes: three axes (X, Y and Z); simultaneous control axes (interpolation axes): three axes (X, Y and Z)
	Interpolation functions: X, Y and Z axes linear, helical and optional two axes circular interpolation.
	Position command range: -9999.999~9999.999mm; minimum command unit: 0.001mm
	Electronic gear ratio: command multiplier 1~32767, command frequency divisor 1~32767
	Rapid traverse speed: maximum 30000mm/min
	Rapid override: F0, 25%, 50%, 100% four levels real-time adjustment.
	Cutting feedrate: maximum 15000mm/min or 500mm/rev. (feed per revolution)
	Feedrate override: 0~150% sixteen-level real-time adjustment
	Manual feedrate: 0~1260mm/min sixteen-level real-time adjustment
	MPG feed: 0.001, 0.01, 0.1mm three gears
	Acceleration or deceleration: the rapid traverse by S acceleration or deceleration, the cutting feed by exponential acceleration or deceleration.
	The automatic chamfering function
G command	62 kinds of G codes: G00, G01, G02, G03, G04, G10, G17, G18, G19, G20, G21, G28, G29, G30, G31, G40, G41, G42, G43, G44, G49, G54, G55, G56, G57, G58, G59, G65, 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. 27 kinds of arithmetic, logical operation and skip can be achieved by macro command G65.
Operation mode	Seven operation modes: Edit, Auto, MDI, DNC, machine zero return, MPG/increment and Manual operation.
Tapping	Tapping function; pitch: 0.001~500mm or 0.06~25400 teeth/inch
	Spindle encoder: encoder linear number can be set (0~5000p/r)
	The drive ratio between encoder and spindle: (1~255): (1~255)
Precision compensation	Backlash compensation: (X, Y and Z axes) 0~2.000mm
	Pitch error compensation: X, Y and Z axes, each of them have 255 compensation points, the compensation amount of each point: -0.255~0.255mm
	Tool compensation: 32 groups tool length compensation, tool nose radius compensation (compensation type C)
M command	Special M commands (redefinition is not allowed): M02、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	Up to 32 tool number (T01~T32), the tool change time sequence is achieved by PLC program.
	Tool life management function

Spindle speed control	The control mode of speed switching value: S □□ command is defined or deposited 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.
	The control mode of speed analog voltage: the spindle speed per minute is commanded by S code, output 0~10V voltage to spindle converter, the spindle stepless shift supporting 4 gears spindle mechanical gear.
PLC function	9 kinds of basic command and 23 kinds of function command, 2-level PLC program that has 5000 steps and the processing time is 2μs for each step. The first level program refresh cycle is 8ms; it can offer the edit software for the ladder and PLC program communication download.
	Integrated machine panel: 41 points input (key), 42 points output (LED) Basic I/O: 32 points input/ 32 points output
Display interface	Displayer: 320×240 lattice, 5.7" mono-color LCD, CCFL back light
	Display mode: multilingual interface set by parameters, which can display the machining path.
Program edit	Program capacity: 22MB, it supports the calling of the user macro, and the subprogram 4 level nesting.
	Edit mode: full-screen editing, support the relative, absolute and mixed coordinates.
Communication	Bilateral program, parameter transmission between CNC and PC, CNC and CNC, supports the system software and the download upgrade of the PLC program serial port, DNC communication between CNC and PC
Suited drive	DA98 series digital AC servo or DY3 series step drive equipment by using the pulse+direction signal input.

G Command Table

G code	Command function	G code	Command function
G00	Positioning (rapid traverse)	G81	Drilling cycle (point-drilling cycle)
*G01	Linear interpolation (cutting feed)	G82	Drilling cycle (counterbore cycle)
G02	circular/helical interpolation by CW	G83	Peck drill cycle
G03	circular/helical interpolation by CCW	G84	Tapping cycle
G04	dwell, exact stop	G85	Boring cycle
G10	offset setting	G86	Drilling cycle
*G17	XY plane selection	G88	Boring cycle
G18	ZX plane selection	G89	Boring cycle
G19	YZ plane selection	*G90	Absolute programming
G20	Inch input	G91	Incremental programming
G21	Metric input	G92	Coordinate system setting
G28	Reference point return	G94	Feeding per minute
G29	Return from reference point	G95	Feeding per revolution
G30	Return from reference point (the 2nd, 3rd, 4th reference points)	G98	Return to the initial plane in canned cycle
G31	Skip function	G99	Return to the R (point) plane in canned cycle
*G40	Tool radius compensation cancellation	G110	Round groove inner rough mill in CCW
G41	Tool radius compensation left	G111	Round groove inner rough mill in CW
G42	Tool radius compensation right	G112	Whole-circle inner finish mill in CCW

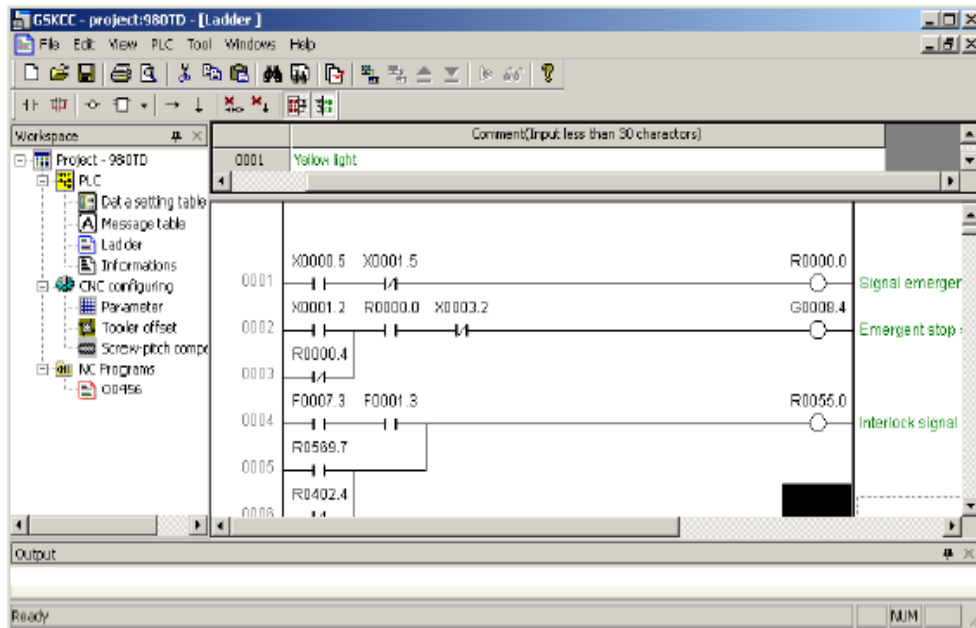
G43	Tool length offset positively	G113	Whole-circle inner finish mill in CW
G44	Tool length offset negatively	G114	Excircle finish mill in CCW
*G49	Tool length offset concellation	G115	Excircle finish mill in CW
*G54	Workpiece coordinate 1	G134	Rectangle groove rough mill in CCW
G55	Workpiece coordinate 2	G135	Rectangle groove rough mill in CW
G56	Workpiece coordinate 3	G136	Rectangle groove inner finish mill in CCW
G57	Workpiece coordinate 4	G137	Rectangle groove inner finish mill in CW
G58	Workpeice coordinate 5	G138	Rectangle outer finish mill in CCW
G59	Workpiece coordinate 6	G139	Rectangle outer finish mill in CW
G65	Macro command	G140	Rectangle path series punch in CW
G73	High-speed peck drill cycle	G141	Rectangle path series punch in CCW
G74	Left-hand tapping cycle	G142	Circular path series punch in CW
*G80	Canned cycle cancellation	G143	Circular path series punch in CCW

PLC Command Table

Basic command	Function	Function command	Function
LD	Read normally open contact	TMRB	Timer
LDI	Read normally closed contact	CODB	Binary system (Bit) transfer
OUT	Output loop	ROTB	Binary system(Bit) spin control
AND	Normally open contact series connection	MOVN	Data copy
ANI	Normally closed contact series connection	DECB	Binary system(Bit) decoding
OR	Normally open contact parallel connection	JMPB	Program skip
ORI	Normally closed contact parallel connection	SP	Subprogram numbering
ORB	Parallel connection of the series circuit block	SPE	Subprogram end
ANB	Series connection of the parallel circuit block	ADDB	Binary (Bit) data addition
		SUBB	Binary (Bit) data subtraction
Function command	Function	ALT	Alternative output
END1	First level program ending	DIFU	Up setting
END2	Second program ending	DIFD	Down setting
SET	Setting	MOVE	Logical AND
RST	Resetting	PARI	Parity check
CMP	Comparison setting	LBL	Program skip numbering
CTRC	Counter	CALL	Subprogram call

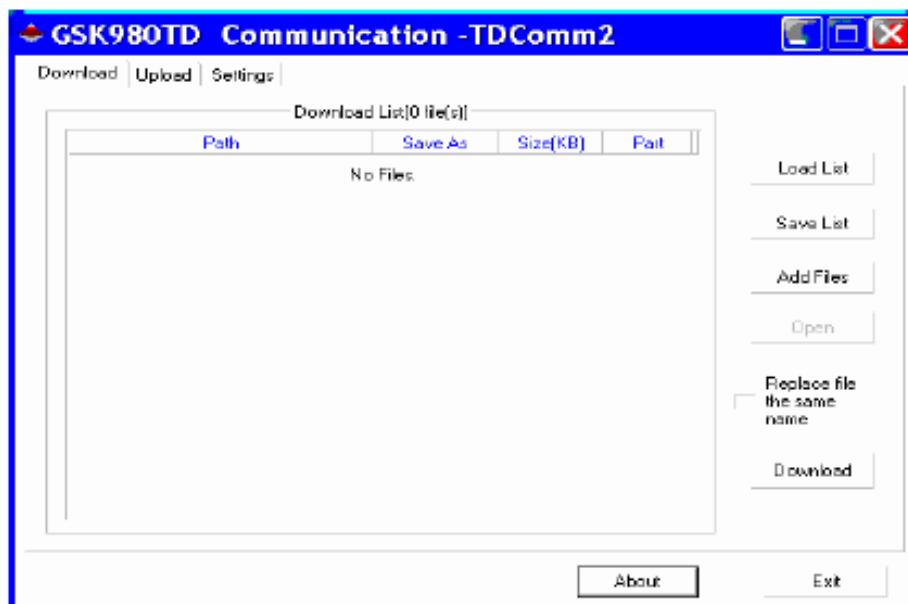
GSK980MD Configuration Software: GSKCC

The GSKCC is run in the condition of the WIN98/2000/XP operation system, so, the machine manufacturer can perform the GSK980MD ladder diagram, parts program, parameter, pitch error compensation data and tool compensation data on the PC to finish the upload and download of the files between PC and GSK980MD system.



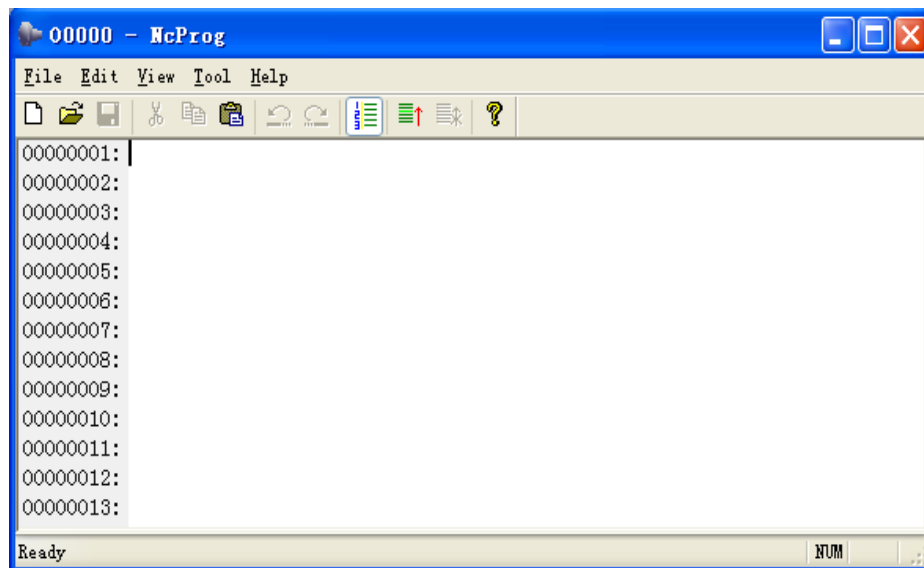
GSK980MD Communication Software: TDComm

The TDComm is run in the condition of the WIN98/2000/XP, which is provided to the end user to finish the bidirectional transmission of the parts program, the parameter, pitch error compensation data and tool compensation data between PC and CNC.



GSK980MD DNC Communication Software: GcodeEdit


The GcodeEdit is run in the condition of the WIN98/2000/XP, which is provided to the manufacturer or the end user to finish the DNC program transmission between PC and CNC.









1.2 Execution of the Program

1.2.1 Order of the program execution

The GSK980MD can not open two or more programs at the same time; it can run the current opened program in Auto mode. So, the GSK980MD can perform only one program at any time. The cursor is located at the beginning of the row of the first block when a program is opened, and the cursor can be moved in Edit mode. Start the execution of

the program from the block the current cursor is located at by using the cycle start signal ( key on the panel or external cycle start key) in the halt state of Auto mode, usually, the blocks are executed one by one in the order of the blocks editing, and the execution is stopped till the M02 or M03 code is executed. The cursor is moved along with the program execution. The program execution sequence or state will be changed in the following conditions:

- The program execution will be stopped if the  key or the Emergency Stop button is pressed;
- The program execution will be stopped if the CNC alarm or PLC alarm is generated;
- The operation mode is switched to the MDI or Edit mode when the program is being executed; or the program is started from the block the cursor is located at when switching to Auto mode by single block stop (The program pauses after the current block executed), and then the  key is pressed or external cycle start signal is ON;
- The operation mode is switched to Manual, MPG, Step and Machine zero mode when the program being executed. The program is held on, then switching to Auto mode, and then the communication is switched on when the  key is pressed or external cycle start signal is ON, the program is operated from the halt position;
- The program pauses when pressing  key or external pause signal is cut off, the program is operated from stopped position when pressing  key or external cycle start signal is ON;
- The program pauses after each block is executed when the single block switch is opened, pressing  key or switching on external cycle start signal is needed, program is executed continuously from the next block;
- The skip switch of the block is opened which it is skipped or inexecuted when the “/” is in front of the block;
- The skip object block is turned to when the G65 skip command is performed
- When M98 or M9000~M9999 command is performed, to call corresponding subprogram or macro run; the subprogram or macro run is ended, when the M99 command is to be performed, to call the next block run as returning to the mainprogram (If M99 command specifies the return object block number, then run by skipping to object block);
- When the M99 command is performed in main program (its run is not started up because others program calls), then returning to the program first stage to continue run, so the current program will be run circularly.

1.2.2 Execution order of command word within block

R, M, S and T, most command words M, S and T are explained by NC before sending to PLC for processing, the other command words are processed by NC directly. M98, M99, M9000~M9999, as well as S command word for spindle speed by rev/min, m/min units are all processed by NC directly.

When G command shares a same block with M00, M01, M02 and M30, the NC performs M command after finishing G command, and then sending the corresponding M signal to PLC for processing.

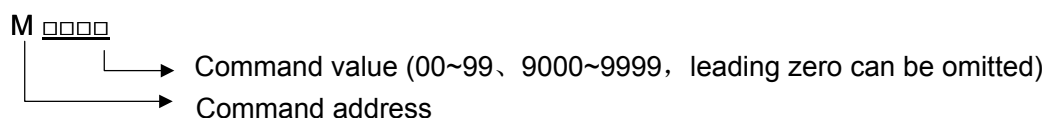
When the G command shares a same block with the M98, M99, M9000~M9999, these M command words are performed by NC after it finishes the G command (the M signal not sent to PLC).

As the M, S and T command words processed by PLC and the G command words are sharing the same block, the M, S and T command words are performed with the G command words at the same time which they are determined by PLC program (ladder diagram), or the M, S and T command words are performed after the G command is finished. As for the performance order of the command words, refer to the explanation of the machine tool manufacturer.

CHAPTER 2 MSTF COMMAND

2.1 M Command (Miscellaneous Function)

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



M98, M99 and M9000~M9999 are independently processed by NC, and the M code is not output to PLC.

The M02 and M30 are defined as program END command by NC, at the same time it also gives the M code to PLC for using the I/O control (close spindle, close cooling etc.).

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

One block only has one M command. The CNC alarm occurs when two or more M commands are displayed in one block.

Table 2-1 M command table for program execution

Commands	Functions
M02	End-of-Run
M30	End-of-Run
M98	Subprogram call
M99	Return from the Subprogram; the program will be circularly executed if the command M99 is used for main program ending (namely, the current program is not called by other programs).
M9000~M9999	Call macro program (Program No. more than 9000)

2.1.1 EP (End of program) M02

Format: M02

Command function: The M02 command is executed in the Auto mode. The automatic run is ended when the other commands of current block are executed; now in order to not return to the program beginning, the cursor is stop at block which the M02 located. If the program is executed again the cursor should be stopped at the beginning of the program.

The function of command M02 also can be defined by the PLC ladder diagram other than the abovementioned functions which are processed by NC. The standard ladder diagram can be defined as: the current input state of CNC is not change after the command M02 is executed.

2.1.2 End-of-run M30

Format: M30

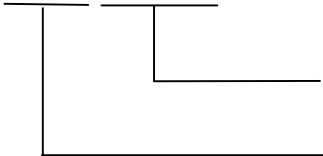
Command function: If M30 command is executed in the Auto mode, the automatic run is ended when the other commands of current block are executed; the system cancels the nose radius compensation and the cursor returns to the beginning of the program when the machine pieces number is added by 1 (It is up to parameter if the cursor returns to the beginning).

The cursor is not return to the beginning of the program when the BIT4 of parameter No.005 is set to 0 in CNC; when it is set to 1, the program is finished, so the cursor returns to the beginning of the program at once.

The function of command M30 can be defined by the PLC ladder diagram other than the abovementioned functions processed by NC. The standard ladder diagram can be defined as: to close the M03, M04 or M08 signal output after the M30 command is executed, at the same time the M05 signal is given.

2.1.3 Subprogram call M98

Format: M98 P○○○○□□□□




The called subprogram No. (0000~9999) .The leading zero of subprogram can be omitted when the call frequency are not given; the subprogram No. should be 4 digits when the calling frequency is given;
Calling frequency (1-9999), calling for once, the input can be omitted

Command function: when the M98 command is executed in the Auto mode, CNC calls and executes the subprogram specified by P, which can be performed 9999 times at most, when the other commands of current block are executed. The M98 command is disabled in MDI.

2.1.4 Return from subprogram M99

Format: M99 P○○○○



The block No. (0000~9999) to be executed when a mainprogram is returned, the leading zero can be omitted.

Command 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 mainprogram; 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).

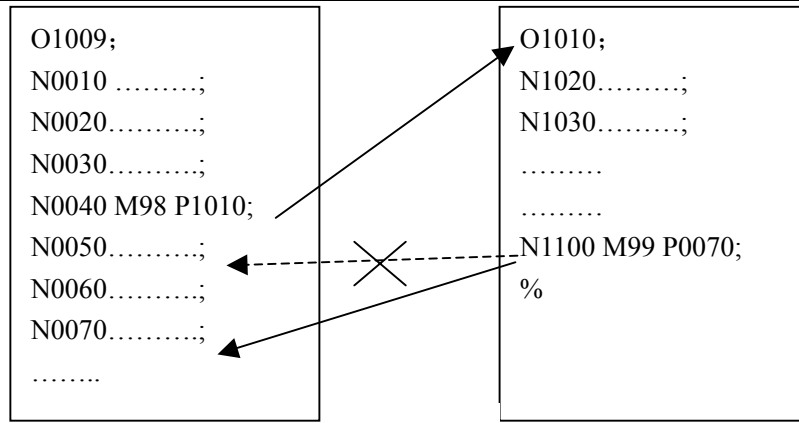


Fig. 2-1

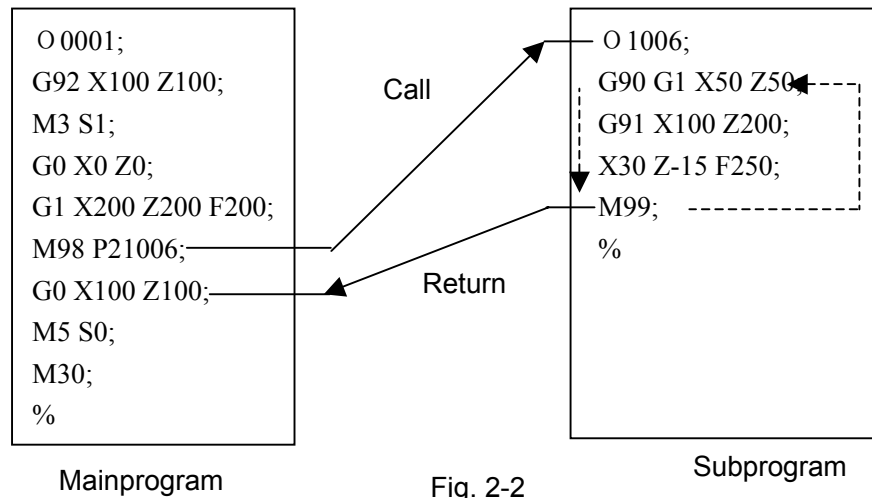


Fig. 2-2

This GSK980MD can calls quadruple subprogram, namely, the other subprogram can be called from the subprogram. (See Fig. 2-3)

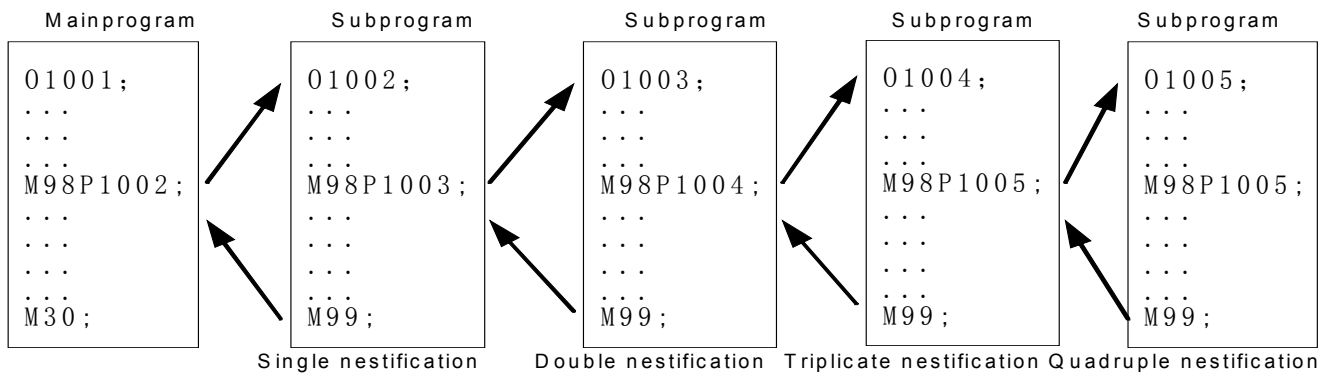


Fig. 2-3 Subprogram nestifications

2.1.5 Macro program call M9000~M9999

Format: M□□□□
 └─ 9000~9999

Command function: Call the macro program which is corresponded by the command value (O9000~O9999).

Macro program: Program 09000~09999 is special space obligated for the machine tool manufacturer for using editing and achieving speical function subprogram, which is called marco program. Two-level operation authority is

needed when editing the program 09000~09999, the user can not modify or operate the macro program but the macro calling command if his authority is 3~5 level. So the M9000~M9999 command operates invalidly in MDI.

2.1.6 M command defined by standard PLC ladder diagram

The M commands other than the abovementioned commands (M02, M03, M98, M99, M9000~M9999) are defined by PLC. The M commands are defined by standard PLC hereinafter. This GSK980MD milling machine is used for machine control. About the function, meaning, control time sequence and logic etc. of the M command, refer to the manual issued by the machine tool builder.

M command specified by standard PLC ladder diagram

Commands	Functions	Remarks
M00	Program pause	
M03	Spindle CCW	Function interlock, state hold
M04	Spindle CW	
*M05	Spindle stop	
M08	Coolant on	Function interlock, state hold
*M09	Coolant off	
M32	Lubrication on	Function interlock, state hold
*M33	Lubrication off	

Notes: The command with “*” specified by standard PLC is valid when the power is turned on.

2.1.7 Program stop M00

Format: M00

Command function: the program is stopped after executing the M00 command, the “pause” is displayed; the program will continue when the key of Cycle Start is pressed.

2.1.8 Spindle CCW, CW, stop control M03, M04 and M05

Format: M03;

M04;

M05;

Command function: M03: spindle forward rotation (CCW);

M04: spindle reverse rotation (CW);

M05: spindle stop.

Note: The control time sequence and logic of M03, M04 and M05 are specified by standard PLC program, refer to the Appendix of this manual.

2.1.9 Coolant control M08, M09

Format: M08;

M09;

Command function: M08: cooling on;

M09: cooling off.

Note: The control time sequence and logic of M08 and M09 are specified by standard PLC program, refer to the Appendix of this manual.

2.1.10 Lubricant control M32, M33

Format: M32;

M33;

Command function: M32: lubrication on;

M33: lubrication off.

Note: The control time sequence and logic of M08 and M09 are specified by standard PLC program; refer to the Appendix of this manual.

2.2 Spindle Function

The spindle speed is controlled by S command, there are two ways to control spindle speed for GSK980MD.

Spindle speed switching value control mode: the S□□ (2-digit command value) command is processed by PLC program for exporting the switching value signal to machine, so that the step speed change of the spindle is achieved.

Spindle speed analog voltage control mode: the actual spindle speed is specified by the S□□□□ (4-digit command value), the NC outputs the 0~10V analog voltage signal to the spindle servo device or transducer for achieving the stepless speed regulating of the spindle.

2.2.1 Spindle speed switching value control

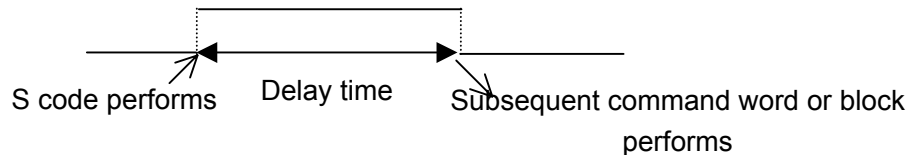
The spindle speed is on switching value control when the BIT4 of bit parameter NO.001 is set to 0. One block only has one S command. The CNC alarm occurs when there are two or more S commands displayed in block.

When the S command shares the same block with the command word, the performance sequence is defined by PLC program. For details, refer to the manual issued by the machine tool builder.

This GSK980MD milling machine is used for machining control when the spindle speed switching value is controlled. The time sequence and logic for S command should be referred by the manual issued by the machine tool builder. The following S command is defined by GSK980MD standard PLC, for reference only.

Command format: S□□
 └─ 00~04 (the leading zero can be omitted): 1~4 gears spindle speed switching value control.

In spindle speed switching value control mode, the FIN signal is returned after the set time of data parameter No.081 is delayed after the code signal of S command is sent to PLC. Now the time is called execution time of S code.



The S01, S02, S03 and S04 output states are invariable when the CNC is reset.

The S1~S4 commands are ineffective output when the CNC is switched on. An arbitrary command is performed from S01, S02, S03 and S04, the corresponding S signal output is effective and held on, at the same time the other 3 S signal output are cancelled. The S1~S4 output are cancelled when performing the S00 command, only one of S1~S4 is effective in the meantime.

2.2.2 Spindle speed analog voltage control

The spindle speed is analog voltage control when the BIT4 of current bit parameter is set to 1

Format: S OOOO
 └─ 0000~9999 (leading zero can be omitted): Spindle speed analog voltage control

Command function: The CNC outputs 0~10V analog voltage to control the spindle servo or transducer for achieving the stepless speed regulating of the spindle when the spindle speed is set. The S command value is not memorized when the power is turned off; and then the parameter recovers to 0 when the power is turned on.

The CNC owns four mechanical spindle shifts function. Counting the corresponding analog voltage value specified by the speed based upon the current set value (corresponding to data parameter No.037~No.040) of the top speed (output analog voltage is 10V) of the spindle shift when the S command is performed, then output the voltage value to spindle servo or transducer, so that the consistency of actual speed and required speed of the spindle are controlled.

The analog voltage output is 0V when the CNC is switched on. The output analog voltage value is invariable (Unless the cutting feed in constant linear speed control and the absolute value of X axis absolute coordinate value are changed) after the S command is executed. The analog voltage output is 0V when the command S0 is executed. And the analog voltage output value is invariable when the CNC is reset or at urgent stop.

The parameter related to spindle speed analog voltage control:

Data parameter No.021: the output voltage offset for spindle top speed (the output analog voltage is 10V);

Data parameter No.043: the voltage offset for the zero spindle speed (the output analog voltage is 0V);

Data parameter No.037~No.040: The top speed for spindle 1~4 shifts (the output analog voltage is 10V);

2.2.3 Spindle override

The spindle actual speed can be modified by using spindle override when the spindle speed analog voltage control is effective, the actual speed modified by spindle override is limited by the top speed of current spindle shift, and also it is controlled by the lowest spindle limitation value and the top spindle limitation value in constant linear speed control mode.

This NC offers 8-level spindle override (50%~120%, the change is 10% per level). The actual level and the modificative mode of the spindle override are defined by PLC ladder diagram. Refer to the manual issued by the machine tool builder when attempting to use it. The following is function description is GSK980MD standard PLC ladder diagram, for reference only.

The spindle override defined by GSK980MD standard PLC ladder diagram has 8 levels. The spindle actual real-time speed can be adjusted by using the spindle override key in the command speed range of 50%~120%, the spindle override will be memorized when the power is turned off. Refer to the OPERATION of this manual for modification operation of the spindle override.

2.3 Tool Function

There is no tool function in this 980MD system.

2.4 Feeding Function

2.4.1 Cutting feed (G94/G95, F command)

Format: G94F_; (F0001~F8000, leading zero can be omitted, for feed speed per minute, mm/min)

Command function: The cutting feedrate is specified by mm/min, G94 is modal G command. If the current mode is G94 that it needs no G94 any more.

Format: G95F_; (F0.0001~F500, leading zero can be omitted)

Command function: The cutting feedrate is offered by the unit of mm/rev., G95 is modal G command. The G95 command can be omitted if the current mode is G95. When the CNC performs G95 F_, the cutting feedrate is controlled by feedrate command based on the multiplication of F command value (mm/rev) and current spindle speed (rev/min). The actual feedrate varies with the spindle speed. The spindle cutting feedrate per revolution is specified by G95 F_, the even cutting line can be formed on the face of workpiece. It is necessary to install spindle encoder when the G95 mode is operated.

The G94 and G95 are modal G commands at the same group, one of them is available only. The G94 is initial state G command, so, it defaults the G94 when the CNC is switched on. The following below shows the conversion formula of feed value per rev. and feed value per min:

$$F_m = F_r \times S$$

Thereinto: F_m : feed value per minute (mm/min);

F_r : feed value per revolution (mm/r);

S : spindle speed (r/min).

The feedrate value is set by the CNC bit parameter No.053 when the CNC is switched on, the F value is invariable after the F command is executed. The feedrate is 0 after F0 is executed. The F value is invariable when CNC is reset or at urgent stop.

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 transducer.

Cutting feed: The CNC makes tool movement path and the path (linear or circular arc) defined by command into consistency (The circular interpolation can be performed by two axes in selected plane when it is circular arc, the helical interpolation is formed by the third axis linear interpolation linkage), by which, the CNC controls three directions movement for X axis, Y axis and Z axis at the same time. The instantaneous speed of movement path in a tangential direction is consistent with the F command value, so this is called CUTTING FEED or INTERPOLATION. The cutting feedrate is supplied by F command, which it is disassembled to each interpolation axis according to the programming path when the CNC performs the interpolation command (cutting feed).

Linear interpolation: The CNC can control the instantaneous speed in the directions of X axis, Y axis and Z axis, so the vector resultant speed in these three directions are equal to the F command value.

$$f_x = \frac{d_x}{\sqrt{d_x^2 + d_y^2 + d_z^2}} \cdot F$$

$$f_y = \frac{d_y}{\sqrt{d_x^2 + d_y^2 + d_z^2}} \cdot F$$

$$f_z = \frac{d_z}{\sqrt{d_x^2 + d_y^2 + d_z^2}} \cdot F$$

F is vector resultant speed for the instantaneous speed in X, Y and Z axis directons

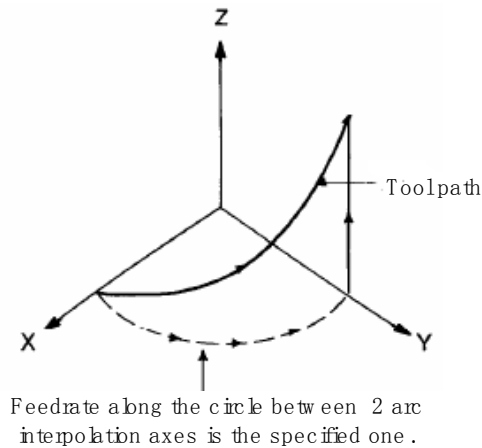
The d_x is instantaneous increment of the X axis, the f_x is instantaneous speed of X axis.

The d_y is instantaneous increment of Y axis, the f_y is instantaneous speed of Y axis.

The d_z is instantaneous increment of Z axis, the f_z is instantaneous speed of Z axis.

Circular interpolation (helical interpolation): Performing the arc interpolation in selected plane, the third axis performs linear interpolation, so the F value is circular interpolation speed. An interpolation of linear and circular arc has the following relation when the linear interpolation speed is f:

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



There are 16 levels feedrate override (0~150%, 10% per level) are offered by NC. The actual feedrate series, the memory performed or not when the power is turned off and the method of overriding are defined by PLC ladder diagram. Refer to the manual issued by the machine tool builder. The function description of GSK980MD standard PLC ladder diagram is as follows, for reference only.

By using the feedrate override key of the machine panel or external override switch it can performs real-time modification for the cutting feedrate. The actual cutting feedrate can be adjusted in the range of command speed 0~150%, here, the feedrate is memorized when the power is turned off. How to operate the cutting feedrate adjustment, refer to Chapter 3 OPERATION of this manual.

Related parameter:

CNC parameter No. 029: the exponential acceleration or deceleration time constant of cutting feed and manual feed.

CNC parameter No.030: the initial (terminal) speed of exponential acceleration or deceleration for cutting feed.

CNC parameter No.031: the upper limit value (X axis, Y axis and Z axis are same) of the cutting feedrate.

2.4.2 Manual feed

Manual feed: This GSK980MD can perform positive/negative movement of X, Y or Z axis by the current manual feedrate in the Manual mode. X axis, Y axis and Z axis can be moved at one time.

This NC offers 16 levels (0~150%, 10% each time) manual feedrate (override), see the following table 2-2. The actual feedrate series and modification mode or the like in manual feeding, are defined by PLC ladder diagram. Refer to the manual issued by the machine tool builder. The function description of GSK980MD standard PLC ladder diagram is as follows, for reference only.

Table 2-2

Feedrate override(%)	0	10	20	30	40	50	60	70	80	90	100	110	120	130	140	150
Manual feedrate (mm/min)	0	2.0	3.2	5.0	7.9	12.6	20	32	50	79	126	200	320	500	790	1260

Note: The manual feedrate of X axis is diameter variation per minute; the feedrate defined by GSK980MD standard PLC ladder diagram is memorized when the power is turned off.

Related parameter: Data parameter No.029: for exponential acceleration or deceleration time constant in manual feed.

Data parameter No.041: for speed lower limit of acceleration or deceleration in manual feed.

2.4.3 MPG/ Step feed

MPG feed: This GSK980MD can move positively or negatively in X, Y or Z axis by current increment in the MPG mode. Only one of the axes can be moved at one time.

Step feed: This GSK 980MD can move positively or negatively for X, Y or Z axis by current increment in the Step mode. One of the axes can be moved only at one time.

Only one mode is effective for the MPG or step mode at one time, it is up to Bit3 of CNC bit parameter No.001.

This NC offers 4 steps (0.001mm, 0.01mm, 0.1mm and 1mm) MPG/ step increment. The actual MPG/ step increment series, the selection of increment and current effective axis or the like, are defined by PLC ladder diagram. Refere to the manual issued by the machine tool builder.

Related parameter: Data parameter No.029: for exponential acceleration or deceleration time constant of cutting feed and manual feed.

Data parameter No.041: for initial or terminal speed of exponential acceleration or deceleration in manual feed.

2.4.4 Automatic acceleration or deceleration

This GSK980MD performs automatically acceleration or deceleration in order to achieve the smooth transition of the speed at the beginning of the axis movement or before the movement stops; this will diminish the impact when the movement is start or stop. This GSK980MD adopts kinds of acceleration or deceleration as follows:

Rapid traverse: linear type front acceleration or deceleration

Cutting feed: exponential type rear acceleration or deceleration

Manual feed: exponential type rear acceleration or deceleration

MPG feed: exponential type rear acceleration or deceleration

Step feed: exponential type rear acceleration or deceleration

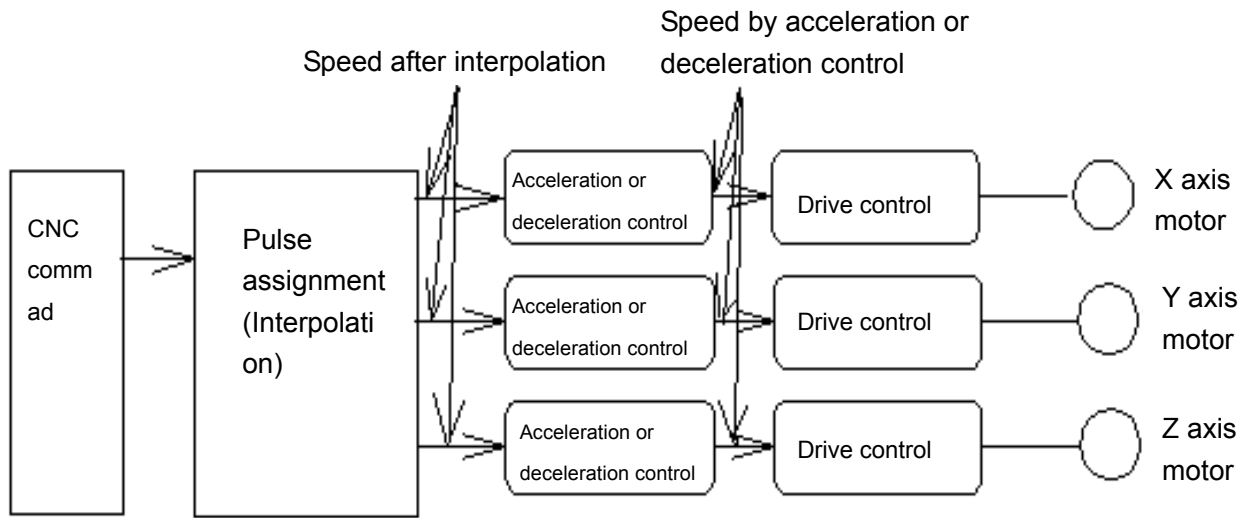


Fig. 2-9

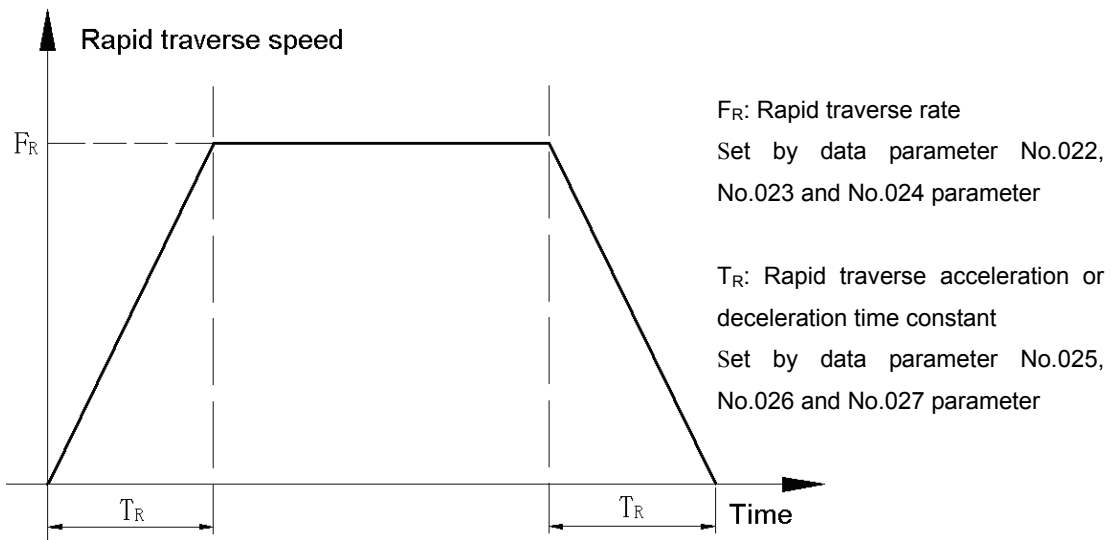
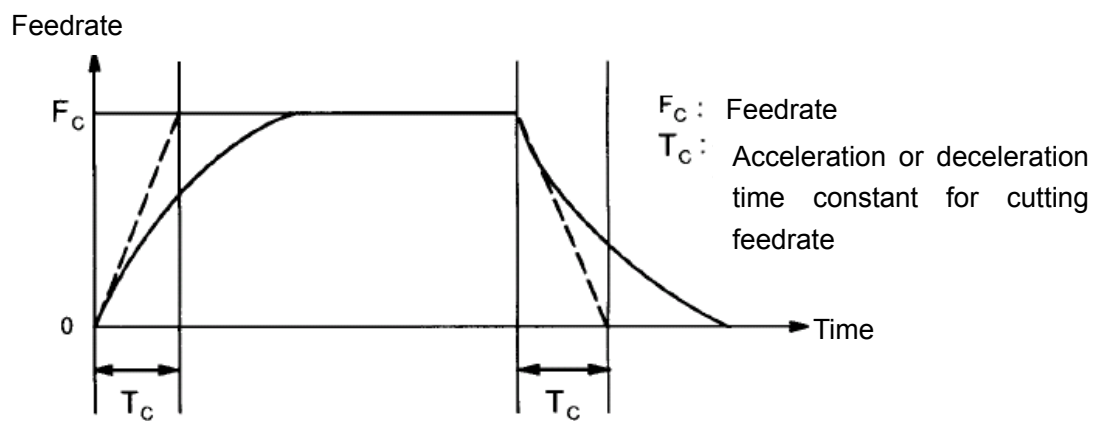


Fig. 2-10 Curve for rapid traverse



F_C : feedrate
 T_C : The acceleration or deceleration time constant of cutting feedrate
(Data parameter No.029)

Fig. 2-11 Curves for cutting and manual feedrate

When the cutting feed is performed, this GSK980MD adopts exponential rear acceleration or deceleration, an arc transition will be formed for the acceleration or deceleration at the meeting point of the path for the adjacent two cutting feed blocks, when the BIT3 of the bit parameter No.007 is set to 0. A contour error exists between the actual tool path and the programmed path when the positioning is not enough accurate at the meeting point of the two paths. In order to avoid this kind of error, the exact stop command (G04;) can be inserted between the two blocks or the BIT3 of the CNC bit parameter No.007 is set to 1. Now, the previous block is decelerated to zero speed and it is positioned to the end of the block, and then the next cutting feed block is performed. The following block can be performed because each block is accelerating from the initial speed and then decelerating to zero at last. If the program time is increasing, it may cause the lower machining efficiency.

The BIT3 of bit parameter No.007 is set to 0, the transition between two adjacent blocks is processed according to the table 2-3.

Table 2-3

Next block \ Previous block			
	Rapid positioning	Cutting feed	Without move
Rapid positioning	X	X	X
Cutting feed	X	O	X
Without move	X	X	X

Note: X: The subsequent block is performed after the previous block is accurately positioning at the end of the block.

O: Each axis speed is transitted according to the acceleration or deceleration between the adjacent blocks; an arc transition is formed at the meeting point of the tool path. (Inaccurate positioning)

Example (The BIT3 of the bit parameter is set to 0)

G91 G01 *-100; (X axis move negatively)

Z-200; (Z axis move negatively)

Y-300; (Y axis move negatively)

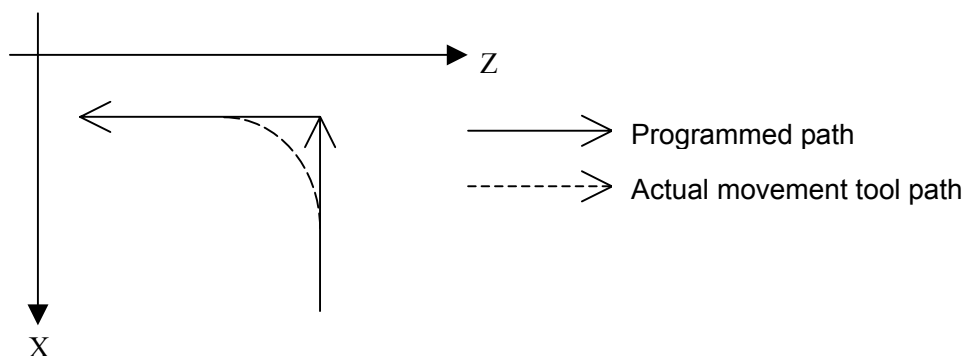
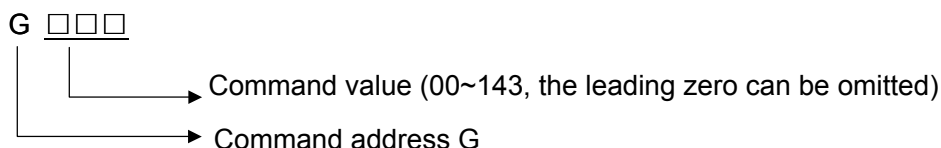


Fig. 2-12

CHAPTER 3 G COMMAND

3.1 Brief

The G command is composed by the command address G and the 1 to 3 digits command value after the command G. Many kinds of operations are specified such as tool movement relative to workpiece, coordinate set, etc. See Table 3-1 for G commands.



The G command words can be classified to 11 groups such as 00, 01, 02, 03, 05, 06, 07, 08, 09, 10 and 14. They share the same block except for 01 and 00 groups, different groups G commands can be defined at the same block. The last G command is valid when two or more same group G commands are introduced at the same block. Different G command groups without common parameter (command word) can be defined at the same block, and their functions are simultaneously valid regardless of sequence. If the G command or the optional G command other than Table 3-1 is employed, alarm occurs.

Table 3-1 G command word list

Command words	Groups	Functions	Remarks
G04	00	Dwell, exact stop	Non-modal G command
G28		Machine zero return	
G29		Return from reference point	
G30		2nd, 3rd and 4th reference point return	
G31		Skip function	
G92		Coordinate system set	
G65		Macro	
G00 (initial G command)	01	Rapid traverse	Modal G command
G01		Linear interpolation	
G02		Circular interpolation (CW)	
G03		Circular interpolation (CCW)	
G73		Peck drilling cycle	
G74		Left-hand (counter) tapping cycle	
G80 (initial G command)		Canned cycle cancellation	
G81		Drilling cycle (spot drill cycle)	
G82		Drilling cycle (counterbore cycle)	
G83		Peck drilling cycle	
G84		Tapping cycle	
G85		Boring cycle	
G86		Drilling cycle	
G88		Boring cycle	

Command words	Groups	Functions	Remarks
G89		Boring cycle	
G110		Circular groove inner rough-milling CW	
G111		Circular groove inner rough-milling CCW	
G112		Circular groove inner fine-milling CW	
G113		Circular groove inner fine-milling CCW	
G114		Excircle finish-milling CW	
G115		Excircle finish-milling CCW	
G134		Rectangle groove rough-milling CW	
G135		Rectangle groove rough-milling CCW	
G136		Rectangle groove inner finish-milling CW	
G137		Rectangle groove inner finish-milling CCW	
G138		Rectangle outter finish-milling CW	
G139		Rectangle outter finish-milling CCW	
G17 (initial G command)	02	XY plane selection	Modal G command
G18		ZX plane selection	
G19		YZ plane selection	
G90 (initial G command)	03	Absolute programming	Modal G command
G91		Relative programming	
G94 (initial G command)	05	Feed per minute	Modal G command
G95		Feed per revolution	
G20	06	Data input in inch	Modal power down memorize
G21		Data input in metric	
G40 (initial G command)	07	Tool nose radius compensation cancellation	Modal G command
G41		Tool nose radius compensation left	
G42		Tool nose radius compensation right	
G43	08	Tool length offset in + direction	Modal G command
G44		Tool length offset in - direction	
G49 (initial G command)		Tool length offset cancellation	
G140	09	Rectangle path serially punch CW	Non-modal G command
G141		Rectangle path serially punch CCW	
G142		Arc path serially punch CW	
G143		Arc path serially punch CCW	
G98 (initial G command)	10	Return to initial level in canned cycle	Modal G command
G99		Return to R level in canned cycle	
G54 (initial G command)	14	Workpiece coordinate system 1	Modal G command
G55		Workpiece coordinate system 2	
G56		Workpiece coordinate system 3	
G57		Workpiece coordinate system 4	
G58		Workpiece coordinate system 5	
G59		Workpiece coordinate system 6	

3.1.1 Modal, non-modal and initial

The G commands can be set to 11 groups such as 00, 01, 02, 03, 05, 06, 07, 08, 09, 10 and 14. Thereinto, G commands of 00 group are non-modal G commands, that of other G group are modal commands. G00, G80, G40, G49 and G94 are initial G commands.

After the G command is executed, the function defined or status is valid until it is changed by other G command where in the same group, this kind of command is called **modal G command**. After this G command is performed and before the function defined or status is changed, this G command need not be input again when the next block performs this G command.

After the G command is performed, the function defined or status is valid for once, the G command word should be input again while every time the G command is performed, this kind of command is called **non-modal G command**.

The modal G command is valid without performing its function or state after the system is powered on, this is called **initial G command**. If the G command is not introduced after the power is turned on, then the initial G command is executed. The initial commands of GSK980MD are G00, G80, G40, G49 and G94.

3.1.2 Examples

Example 1

O0001;

G17 G0 X100 Y100; (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 power is turned on)

O0003;

G90 G94 G01 X100 Y100 F500; (G94 feed per minute, feedrate is 500mm/min)

G91 G95 G01 X10 F0.01; (G95 feed per revolution, input the F value again)

G90 G00 X80 Y50;

M30;

3.1.3 Related definition

The words or characters which are not specially described in this manual are as follows:

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 G90, the incremental value of X axis against current point for G91;

Y: the absolute coordinate of Y axis at the end for G90, the incremental value of Y axis against current point for G91;

Z: the absolute coordinate of Z axis at the end for G90, the incremental value of Z axis against current point for G91;

F: Cutting feedrate.

3.1.4 Address definition

Usage of the address in system is as follows:

Address	Function	Value range	Rounding
A	Punching number for rectangle serial punch (G140/G141) in side 1 and side 3.	0~9999	Decimal part omitted
B	Punching number for rectangle serial punch (G140/G141) in side 2 and side 4.	0~9999	Decimal part omitted
	Radius for arc serially punch (G142/143)	-9999.999~9999.999	Round-off
C	Radius of arc chamfer	-9999.999~9999.999 Absolute value for negative	Round-off
	Punching number of arc serially punch (G142/G143)	0~9999	Decimal part omitted
D	Tool radius offset number	0~32	Decimal alarm
E	Unused		
F	G94 feed per minute	0~9999	Decimal
	G95 feed per rotation	0.0001~500	Round-off
	Tooth pitch in G74,G84	0.001~500	Round-off
G	G code	G command in system	Decimal alarm
H	Length offset number	0~32	Decimal alarm
	Operation command in G65	0~99	Decimal alarm
I	Distance from arc start point to center point in X direction	-9999.999~9999.999	Round-off
	G110~G115: radius value of circle	-9999.999~9999.999 Absolute value for negative	Round-off
	G134~G139: width of rectangle in X direction	-9999.999~9999.999 Absolute value for negative	Round-off
J	Distance from arc start point to center point in Y direction	-9999.999~9999.999	Round-off
	G112,G113: distance from start point to center point	-9999.999~9999.999 Absolute value for negative	Round-off
	G114,G115: distance from start point to circle	-9999.999~9999.999 Absolute value for negative	Round-off
	G134~G139: width of rectangle in Y direction	-9999.999~9999.999 Absolute value for negative	Round-off
	G140,G141: length of 2nd side of rectangle	-9999.999~9999.999 Absolute value for negative	Round-off

K	Distance from arc start point to the center point in Z direction	-9999.999~9999.999	Round-off
	G110,G111,G134,G135: cutting increment in XY plane each time	-9999.999~9999.999 Absolute value for negative	Round-off
	G136~G139: distance from start point to rectangle side in X axis direction	-9999.999~9999.999 Absolute value for negative	Round-off
L	The length of linear chamfering	-9999.999~9999.999 Absolute value for negative	Round-off
	Punching number for linear serial punch (use together with the canned cycle punch)	-9999.999~9999 Absolute value for negative	Round-off
	Tool life management, tool life value	0~ 999999	Decimal part omitted
M	M miscellaneous function	0~99	Decimal alarm
	M code subprogram call	9000~9999	Decimal alarm
N	Program number	0~2 ³¹	Decimal alarm
	Tool life: tool life unit (0-time, non-0 -time)	0 or other number	Decimal alarm
O	Program number	0~9999	
P	Delay time in G04 (ms)	-9999999~ 9999999	Decimal alarm
	What kind of number reference return in G30	2~4	Decimal part omitted
	Skip sequence or alarm number in G65	0~9999	Decimal alarm
	M98 subprogram call (times+program name)	0~99999999	Decimal alarm
	Sequence number of M99 subprogram return	0~9999	Decimal alarm
Q	Specifying G73 and G83 cut-in value per time	-9999.999~9999.999 Absolute value for negative	Round-off
	The value of operation in G65	-999999999 ~999999999	Decimal alarm
R	Radius value of arc	-9999.999~9999.999	Round-off
	R level value of canned cycle command	-9999.999~9999.999	Round-off
	The value of operation in G65	-999999999 ~999999999	Decimal alarm
S	Analog spindle	0~9999	Decimal alarm
	Shift spindle	0~99	Decimal alarm
T	Number of tool	0~55# parameter set value	Decimal alarm
	Tool compensation number	0~32	Decimal alarm
U	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	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 Positioning G00

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.

Explanations: 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.

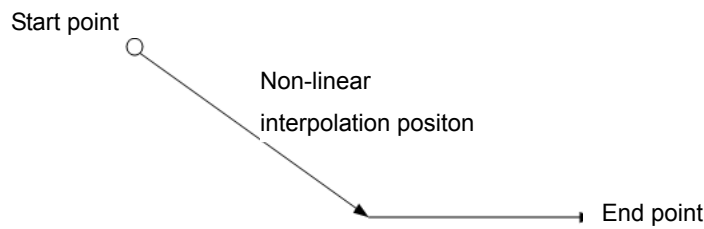


Fig. 3-1

X, Y and Z axes are separately set by the system data parameter No.022, No.023 and No.024 at their rapid traverse rate, the actual traverse rate can be modified by the rapid override keys on the machine panel.

The rapid traverse acceleration or deceleration time constant of X, Y and Z axes are separately set by the system data parameter No.025, No.026 and No.027.

Example: tool traverses from point A to point B. See Fig.3-2.

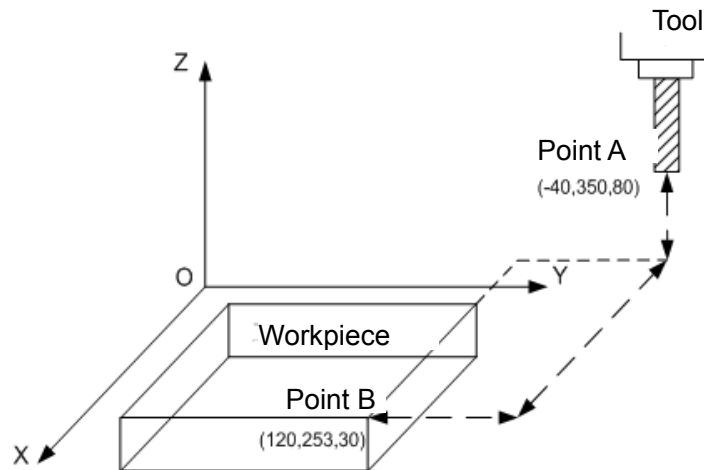


Fig. 3-2

G90 G0 X120 Y253 Z30; (absolute coordinate programming)

G91 G0 X160 Y-97 Z-50; (relative coordinate programming)

3.3 Linear Interpolation G01

Format: G01 X_Y_Z_F_;

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

Explanations: 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;

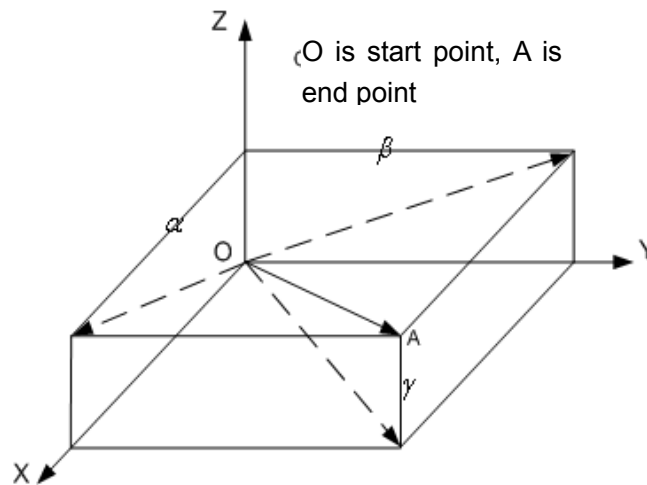
F command value is invariable after it is performed till the new one is executed. The following G command with F command word uses the same function.

The value range is indicated as follows:

Command function	G94 (mm/min)	G95 (mm/rev)
Value range	1~8000	0.001~500

Command path figure:

The linear interpolation is performed from point O to point A: G01 X α Y β Z γ F f;



$$L = \sqrt{\alpha^2 + \beta^2 + \gamma^2}$$

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

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

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

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

Note: The F initial default value is set by data parameter No.30 when the power is turned on.

3.4 Arc and Helical Interpolation G02, G03

Format:

Circular interpolation

Arc in the XY plane

$$G17 \left\{ \begin{array}{c} G02 \\ G03 \end{array} \right\} X_ Y_ \left\{ \begin{array}{c} R_ \\ I_ J_ \end{array} \right\} F_$$

Arc in the ZX plane

$$G18 \left\{ \begin{array}{c} G02 \\ G03 \end{array} \right\} X_ Z_ \left\{ \begin{array}{c} R_ \\ I_ K_ \end{array} \right\} F_$$

Arc in the YZ plane

$$G19 \left\{ \begin{array}{c} G02 \\ G03 \end{array} \right\} Y_ Z_ \left\{ \begin{array}{c} R_ \\ J_ K_ \end{array} \right\} F_$$

Helical interpolation

Arc interpolation in XY plane, Z axis linear interpolation linkage;

$$G17 \left\{ \begin{array}{c} G02 \\ G03 \end{array} \right\} X_ Y_ Z_ \left\{ \begin{array}{c} R_ \\ I_ J_ \end{array} \right\} F_$$

Arc interpolation in ZX plane, Y axis linear interpolation linkage:

$$G18 \left\{ \begin{array}{c} G02 \\ G03 \end{array} \right\} X_ Z_ Y_ \left\{ \begin{array}{c} R_ \\ I_ K_ \end{array} \right\} F_$$

Arc interpolation in YZ plane, X axis linear interpolation linkage:

$$G19 \left\{ \begin{array}{c} G02 \\ G03 \end{array} \right\} Y_ Z_ X_ \left\{ \begin{array}{c} R_ \\ J_ K_ \end{array} \right\} F_$$

Function: Only two axes of circular interpolation can be linked for controlling tool movement along with the arc on the selected plane in any time. If the 3rd axis is specified simultaneously in linear interpolation mode, it will be linked by linear interpolation type to constitute helical interpolation. G02 movement path is CW from start to end points. G03 movement path is CCW from start to end points.

Explanations:

G02 and G03 are modal G command;

R is arc radius, the value range are indicated as -9999.999~9999.999mm;

When the circle center is specified by address I, J and K, they are corresponding with the X, Y and Z axes separately.

I is the difference between the center point and the arc start point in the X axis direction, I= center point coordinate X- X coordinate of arc start point; the value range are indicated as -9999.999~9999.999mm;

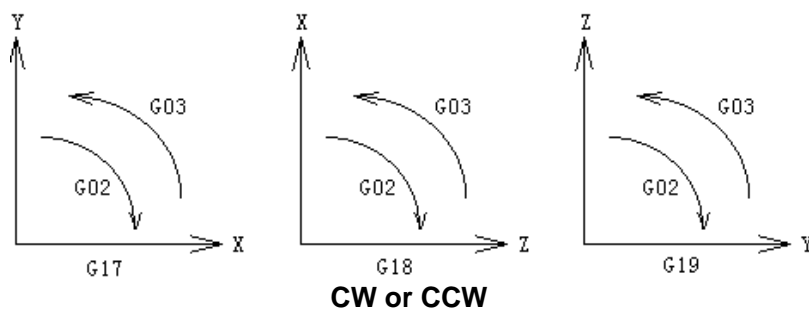
J is the difference between the center point and the arc start point in the Y axis direction, J=center point coordinate Y- Y coordinate of circle arc start point; the value range are indicated as -9999.999~9999.999mm;

K is the difference between the center point and circle start point in the Z axis direction, K=center point coordinate Z- Z coordinate of circle start point; the value range are indicated as -9999.999~9999.999mm.

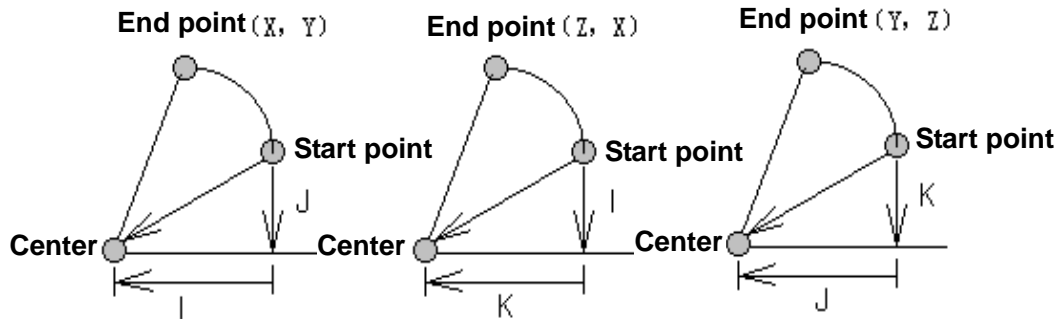
Note: When I, J and K are for whole-circle that they have signs according to the direction. They are positive values when I, J and K shares the same directions with X, Y and Z axes; otherwise they are negative ones.

Item	Specified content		Command	Meaning
1	Plane specification		G17	Specifying XY plane arc
			G18	Specifying ZX plane arc
			G19	Specifying YZ plane arc
2	Rotating direction		G02	CW
			G03	CCW
3	End point	G90 mode	Two axes of X, Y and Z	End point in the part coordinate system
		G91 mode	Two axes of X, Y and Z	Distance from start to end points
4	Distance from start point to circle center point		I	X axis distance from start point to the center point (with sign)
			J	Y axis distance from start point to the center point (with sign)
			K	Z axis distance from start point to the center point (with sign)
	Arc radius		R	Arc radius
5	Feedrate		F	Feedrate along the arc

“Clockwise” and “Counterclockwise” are defined when XY plane(ZX plane, YZ plane) is viewed in the positive-to-negative direction of the Z axis (Y axis, X axis) in the Cartesian coordinate system, see the following figure figure:



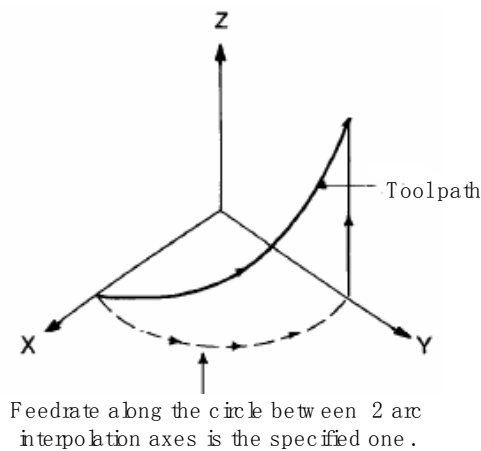
The end point of an arc is specified by using the address X, Y or Z, and is expressed as an absolute or incremental value according to G90 or G91. The incremental value is the distance value from start to end points of an arc. The arc center is specified by address I, J and K against the X, Y and Z respectively. The numerical value following I, J and K, however, is a vector component from start point of an arc to the center point, which is an incremental value with sign. See the following figure figure:



The F command is circular interpolation rate in helical interpolation, in order to achieve the linkage interpolation between linear axis and arc, the speed of linear interpolation by the 3rd axis has the following relationship to the F command:

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

Helical interpolation path is as follows:



I, J and K have signs according to the direction. The circular center also can be specified by radius R other than I, J and K, as follows:

G02 X_ Y_ R_ ;

G03 X_ Y_ R_ ;

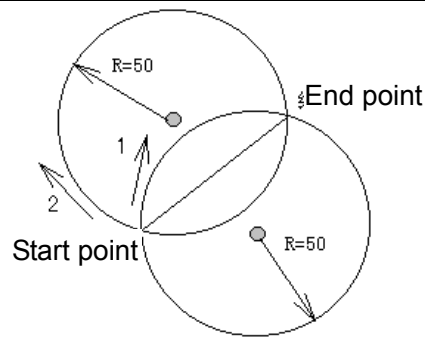
Now, the following two arcs can be described, one arc is more than 180°, the other is less than 180°. The arc radius which is less than 180° is specified by the positive value; the arc radius which is more than 180° is specified by the negative value. The radius is either positive or negative when the arc command is equal to 180°.

(Example) Arc ① less than 180°

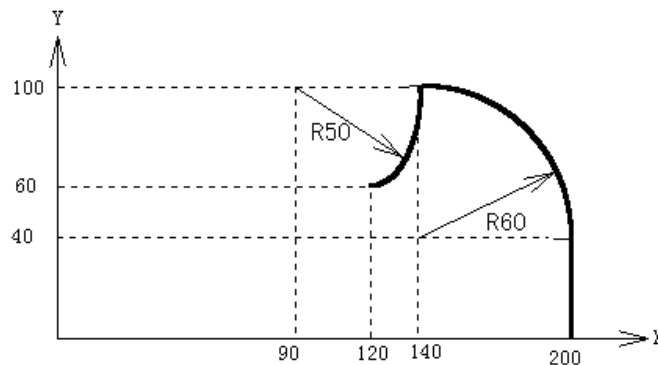
G91 G02 X60.0 Y20.0 R50.0 F300.0;

Arc ② more than 180°

G91 G02 X60.0 Y20.0 R-50.0 F300.0;



(Example for the programming)



To program the above paths using the absolute mode and incremental mode respectively:

(1) Absolute mode

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

(2) Incremental mode

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

The feedrate of circular interpolation is specified by F command; it is the speed of the tool along the arc tangent direction.

Note 1: I0, J0 and K0 can be omitted; but, it is very necessary to input one of the addresses I, J, K or R, or the system alarm is generated.

Note 2: The X, Y and Z can be omitted simultaneously when the end and start points share same position. When the center point is specified by address I, J and K, it is a 360° arc.

G02 I_; (Full circle)

The circle is 0° when using R.

G02 R_; (not move)

It is recommended that programming uses R. In order to guarantee the start and end points of the arc are consistent with the specified value, the system will move by counting R again according to the selected plane, when programming using the I, J and K.

Plane selection	Count the radius R value again
G17	$R = \sqrt{I^2 + J^2}$
G18	$R = \sqrt{I^2 + K^2}$
G19	$R = \sqrt{J^2 + K^2}$

Note 3: The error between the actual tool feedrate and the specified feedrate is $\pm 2\%$ or less. The command speed is movement speed after tool radius offset along the arc.

Note 4: The R is effective when address I, J and K are commanded with the R, but the I, J and K are disabled at one time.

Note 5: The axis not exist is specified on the set plane, the alarm occurs.

Note 6: If the radius difference between start and end points exceeds the permitted value by parameter (No.100), a P/S alarm occurs.

3.5 Dwell G04

Format: G04 P_ ; or
G04 X_ ;

Function: Axes stop, the current G command mode and the data, status are invariable, after delaying time specified, the next block will be executed.

Explanations: G04, which is a non-modal G-command;

G04 delay time is specified by command words P_, X_;

P and X are indicated by 0.001~9999.999 sec.

. See the following figure table for time unit of P_ and X_ command value:

Address	P	X
Unit	0.001sec.	second

Notice:

- 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 bewteen the blocks.
- 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.

3.6 Plane Selection Command G17, G18 and G19

Format:

G17 XY plane
G18 ZX plane
G19 YZ plane

Function: The plane of arc interpolation and tool radius compensation are chosen by using the G code

Explanation: G17, G18 and G19 are modal G commands, the plane will not be changed when a block without any command inside.

Command example:

G18 X_Z_ ; ZX plane
X- Y- ; invariable plane (ZX plane)

Note 1: The plane selection command can share the same block with other group G commands.

Note 2: The move command is regardless of the plane selection. For example, the Z axis is not

On XY plane, the Z axis movement is regardless of the XY plane in command G17 Z_ .

G17 Z_ ;

3.7 Conversion of Inch and Metric G20 and G21

Format:

G20/G21;

Function: The input unit either inch or metric is chosen by G code.

Explanations:

Unit system	G codes	Min. set unit
Metric	G20	0.0001 inch
Inch	G21	0.001 mm

The G code should be placed in front of the program when inch and metric is switched each other. Before the coordinate system is set, it is specified by a single block command.

The following unit systems vary according to the G code for inch or metric conversion.

- (1) Feedrate command value by F.
- (2) Command value related to the position.
- (3) Offset.
- (4) 1 scale value for MPG.
- (5) Step amount value.
- (6) A part of numerical value of the parameter.

Note 1: The G code for inch or metric conversion when the power is turned on is same as that at the power off.

Note 2: Changing G20 and G21 are unallowed during programming.

Note 3: When the unit systems between the machine and input are different, the max. error is 0.5 of the min. move unit; and the error is not be cumulated.

Note 4: As the inch input (G20) and the metric input (G21) switches each other, the offset should be suited to the reset of the input unit.

3.8 Reference Point Return G28

Format: G28 X_ Y_ Z_;

Function: The middle point position specified by X, Y and Z is reached from the start point at the rapid traverse rate, then it returns to the reference point.

Explanations: G28 is a non-modal G-command;

X: The absolute coordinate of middle point in X axis is indicated by G90, the middle point increment against current point in X axis is indicated by G91;

Y: The absolute coordinate of middle point in Y axis is indicated by G90, the middle point increment against current point in Y axis is indicated by G91;

Z: The absolute coordinate of middle point in Z axis is indicated by G90, the middle point increment against current point in Z axis is indicated by G91.

One of the command address X, Y and Z or all of them can be omitted, as follows:

Commands	Functions
G28	3 axes holds on at the initial position, the next block continued.
G28 X__	X axis reference point return, Y and Z axes still in the original position
G28 Y__	Y axis reference point return, X and Z axes still in the original position
G28 Z__	Z axis reference point return, X and Y axes still in the original position
G28 X__ Z__	X and Z axes reference point return simultaneously, Y axis in the original position
G28 X__ Y__	X and Y axes reference point return simultaneously, Z axis in the original position
G28 Y__ Z__	Y and Z axes reference point return simultaneously, X axis in the original position
G28 X__ Y__ Z__	X, Y and Z reference point return simultaneously

Process for command action (See the figure 3-10):

- (1) Positioning from current position to intermediate point of command axis at the rapid traverse rate (From point A to B)
- (2) Positioning to the reference point from intermediate point at the rapid traverse rate (From point B to R)
- (3) If the machine tool is unlocked, the zero return indicator lights up when the reference point return is finished.

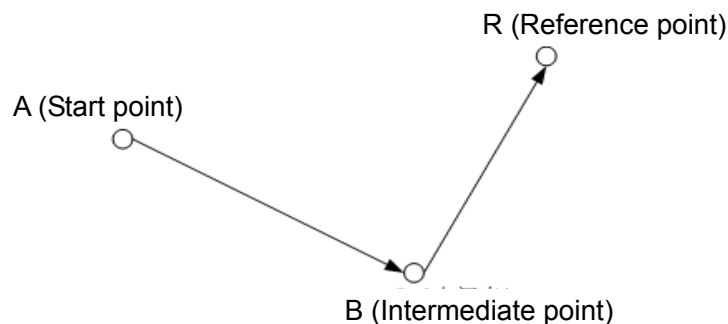


Fig. 3-10

Note 1: The process is consistent for machine zero point return between by manual and the G28 command, it is very necessary to check the deceleration signal and one-turn signal each time;

Note 2: Two axes moves at their rapid traverse rate separately from point A to B or from B to R. So, the movement path is not always a straight line;

Note 3: After the G28 command machine zero operation return is performed, the tool length compensation cancellation is determined by the bit 7 of system parameter No.183

Note 4: The G28 command can not be performed or machine zero operation can not be returned if the machine tool is not installed the zero point switch.

3.9 Return From Reference point G29

Format: G29 X_ Y_ Z_;

Function: When a rapid traverse is performed from the current point to mid point, it positions to the specified position by X, Y and Z at the rapid traverse rate.

Explanations:

X: The absolute coordinate of aim point in X axis is indicated by G90; the aim point increment against the mid point in X axis is indicated by G91;

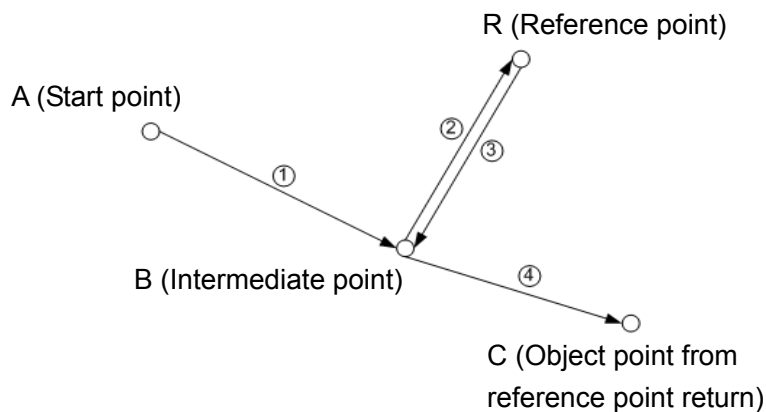
Y: The absolute coordinate of aim point in Y axis is indicated by G90; the aim point increment against the mid point in Y axis is indicated by G91;

Z: The absolute coordinate of aim point in Z axis is indicated by G90; the aim point increment against the mid point in Z axis is indicated by G91;

One of the command address X, Y and Z or all of them can be omitted, see the following figure:

Commands	Functions
G29	X,Y and Z axes are in the original position, the next block continued
G29 X__	Only X axis performs the command returning from the reference point
G29 Y__	Only Y axis performs the command returning from the reference point
G29 Z__	Only Z axis performs the command returning from the reference point
G29 X__ Z__	Only X and Z axes performs the command returning from the reference point
G29 X__ Y__	Only X and Y axes performs the command returning from the reference point
G29 Y__ Z__	Only Y and Z axes performs the command returning from the reference point
G29 X__ Y__ Z__	X, Y and Z performs the command returning from the reference point

Process for command action:



(1) The command axis direction performs positioning at the intermediate point specified by G28 (from point R to B), the action is **ERROR! REFERENCE SOURCE NOT FOUND. →ERROR! REFERENCE SOURCE NOT FOUND.**

(2) The positioning is performed from intermediate point to specified point (from point B to C), moving to the intermediate and command point at a rapid feedrate, the operation is **ERROR! REFERENCE SOURCE NOT FOUND. →ERROR! REFERENCE SOURCE NOT FOUND.**

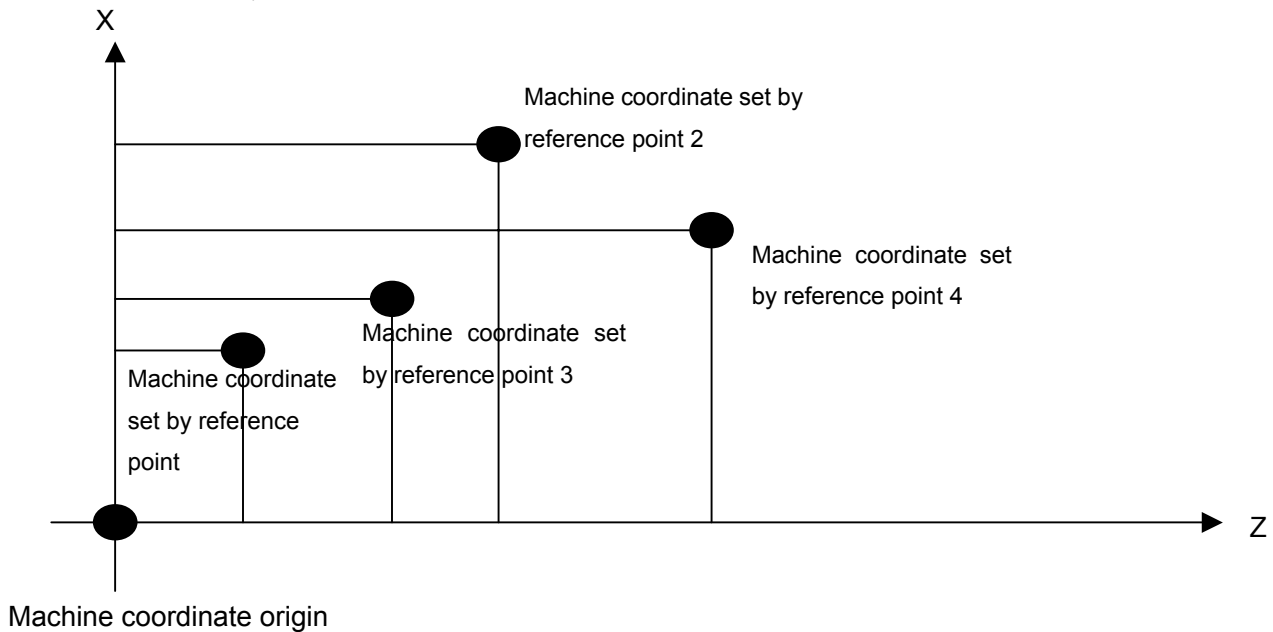
Note 1: G29 is specified after G28, if an intermediate point is not specified by any of axes, the system alarm will be generated.

Note 2: It is incremental distance against the intermediate point in G91 coordinate programming.

Note 3: Current position is reference point when the G29 command is followed to G28 or G30, it returns from reference point directly; or, it returns from current position if G29 command is not followed by G28 or G30.

3.10 The 2nd, 3rd and 4th Reference Point Return G30

Reference point is a fixed point on the machine. By parameters (80#~90#) it can set four reference points in the machine coordinate system.



Format:

G30 P₂ X₋ Y₋ Z₋ ; the machine 2nd reference point *return* (P2 can be omitted)

G30 P₃ X₋ Y₋ Z₋ ; the machine 3rd reference point *return*

G30 P₄ X₋ Y₋ Z₋ ; the machine 4th reference point *return*

Function: From the start point, after the intermediate point by X, Y and Z is reached at a rapid traverse rate, the machine 2nd, 3rd and 4th reference points are returned. The command word P2 can be omitted when the machine 2nd reference point is returned.

Explanations: G30, which is a non-modal G-command;

X: X axis coordinate for intermediate point;

Y: Y axis coordinate for intermediate point;

Z: Z axis coordinate for intermediate point;

One of the command address X, Y and Z or all of them can be omitted, see the following figure:

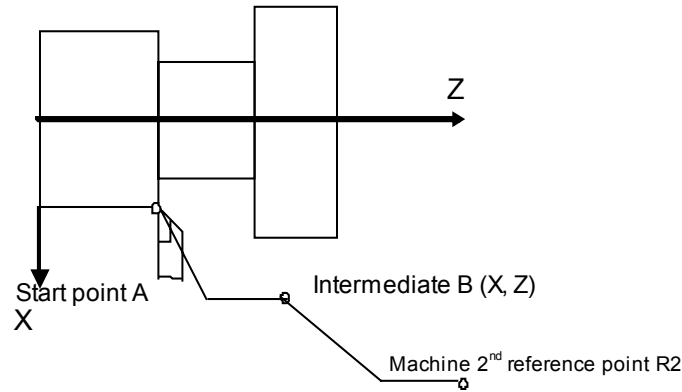
Commands	Functions
G30 P _n X ₋	Machine n th reference point return for X axis, Y and Z axes in the original position
G30 P _n Y ₋ Z ₋	Machine n th reference point return for Y and Z axes, X axis in the original position
G30	3 axes in the original position, the next block continued
G30 P _n X ₋ Y ₋ Z ₋	X, Y and Z axes return to the machine n th reference point simultaneously

Note 1: n is 2, 3 or 4 in above table;

Note 2: Deceleration and zero signals check are not needed when the machine 2nd, 3rd and 4th reference points are returned to.

Command action process (see the following figure figure, an instance of machine 2nd reference point return):

- (1) Positioning to intermediate point of the specified axis from current position at a rapid traverse rate (from point A to point B);
- (2) Positioning to the 2nd reference position set by data parameter No.84 and No.86 at the setting speed by data parameter No.75 (from point B to point R2)
- (3) When the reference point returns if the machine is unlocked, the Bit 0 and Bit 1 of the reference point returning end signal ZP21 are HIGH.



Note 1: After returning the machine reference point by manual or the G28 command is performed, the machine 2nd, 3rd and 4th reference point return function can be employed only;

Note 2: From point A to B or from point B to R2, the 2 axes are moved at their separately rate, so the path is not straight line possibly.

Note 3: After machine 2nd, 3rd and 4th reference point return by the G30 command, the system tool length compensation cancellation is defined by bit 7 of the parameter No.183.

Note 4: The 2nd, 3rd and 4th reference point operation of G30 command can not be executed if the the zero swith is not installed on the machine tool.

Note 5: The workpiece coordinate system is set after the machine 2nd, 3rd and 4th reference point are returned.

3.11 Skip Function G31

As G01 linear interpolation is performed, if an external SKIP signal is valid during execution of this command, execution of this command is interrupted and the next block is executed. The skip function is used when the end of machining is not programmed but specified with a signal from the machine, for example, in grinding. It is used also for measuring the dimensions of a workpiece.

Format:

G31 X__ Y__ Z__

Explanations:

1. G31, which is a non-modal G-code, it is effective only in the block in which it is specified.
2. G31 can not be specified in the C tool compensation and chamfering, or the alarm will be generated. It is very necessary to cancel the C tool compensation and chamfering firstly before the G31 command is specified.
3. Error is allowed in the position of the tool when a skip signal is input.

Signal:

The SKIP signal input is on the fixed address X1.0 (XS40-20).

Parameter:

0	1	3								G31P	SKPI
---	---	---	--	--	--	--	--	--	--	------	------

SKIP 1: HIGH level SKIP is valid;

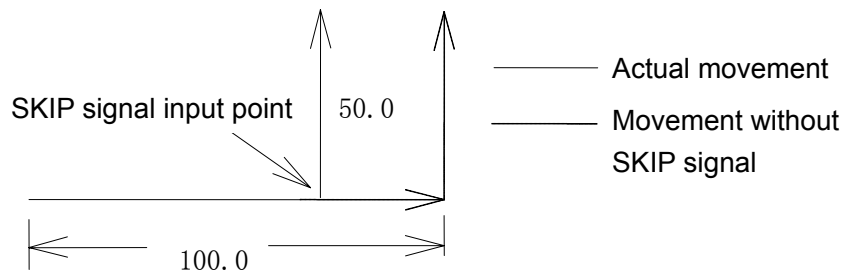
0: LOW level SKIP is valid.

G31P 1: G31 is for immediate stop as the SKIP signal is valid;

2: G31 is for decelerating stop as the SKIP signal is valid.

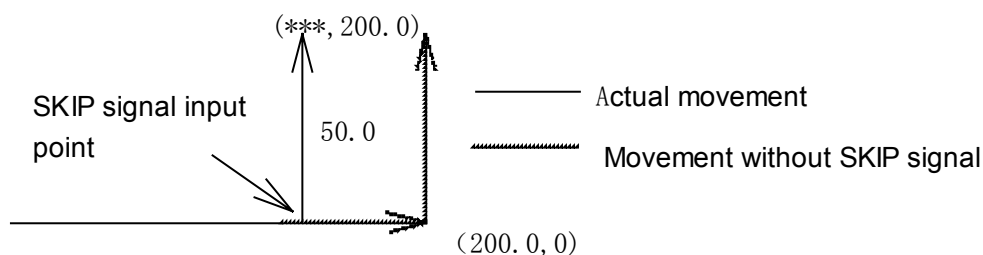
1. The next block to G31 is incremental command 1: it moves with incremental value from the position interrupted by the skip signal.

Example: G31 G91 X100.0 F100 ;
Y50.0 ;



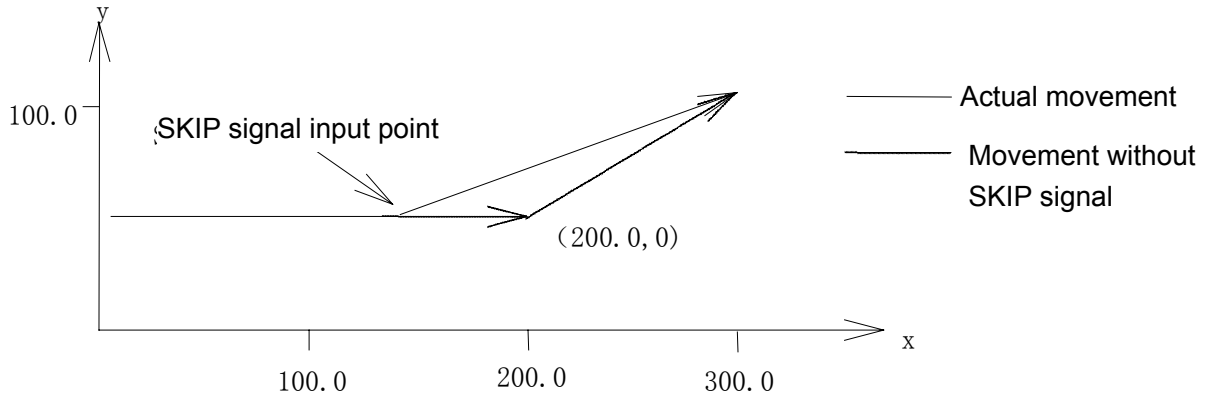
2. The next block to G31 is absolute command for one axis: The command axis moves to the specified position, and the axis not specified keeps at the skip signal input position.

Example: G31 G90 X200.0 F100 ;
Y100.0 ;



3. The next block to G31 is absolute command for 2 axes: Wherever the skip signal input is, the tool moves to specified position of next block.

Example: G31 G90 X200.0 F100 ;
X300.0 Y100.0 ;



3.12 Tool Radius Compensation C (G40, G41 and G42)

Format

$$\left. \begin{array}{l} G17 \\ G18 \\ G19 \end{array} \right\} \left. \begin{array}{l} G41 \\ G42 \end{array} \right\} D_$$

Functions

Tool nose radius compensation function

To cancel or perform the tool radius compensation vector by using the commands G40, G41 and G42. They are combined with the commands G00, G01, G02 and G03 for specifying a mode which can be confirmed the compensation vector value, direction and the direction of tool movement.

G codes	Functions
G40	Tool radius compensation cancellation
G41	Tool radius left compensation
G42	Tool radius right compensation

G41 or G42 drives the system into compensation mode; G40 cancels the system compensation mode.

Explanations:

- Compensation plane

The compensation plane can be confirmed based upon plane selection command; the C tool compensation is calculated in this plane.

Plane selection	Plane compensation
G17	X—Y plane
G18	Z—X plane
G19	Y—Z plane

- Compensation value (D code)

This system can be set for 32 compensation values at most. Two digits specified by D code in the program, is called serial number of compensation value, the compensation value should be set by MDI/LCD unit.

D code determines the compensation value in tool offset page according to the bit 1 of parameter No.003, it is very important to notice that the value applied is diameter or radius.

Setting range of compensation value is as follows:

	Millimeter Input (mm)	Inch input (inch)
Compensation value	0~+9999.999mm	0~+999.999 inch

- Compensation vector

The compensation vector is two-dimensional vector; it is equal to the compensation value specified with D code. The compensation vector is calculated in control unit, its direction is real-time modified along with the tool path in each block. You can calculate how much compensation is needed for tool movement when the compensation value is applied in control unit. Compensation path (tool center path) = programmed path \pm tool radius (or diameter) (determined by compensation direction)

Notes 1: Compensation operation is performed within the plane selected by G17, G18 and G19. For example, compensation and vector calculations are performed with (X, Y) or (I, J) in block when XY plane is selected. The axis coordinate values outside the compensation plane are not affected by compensation.

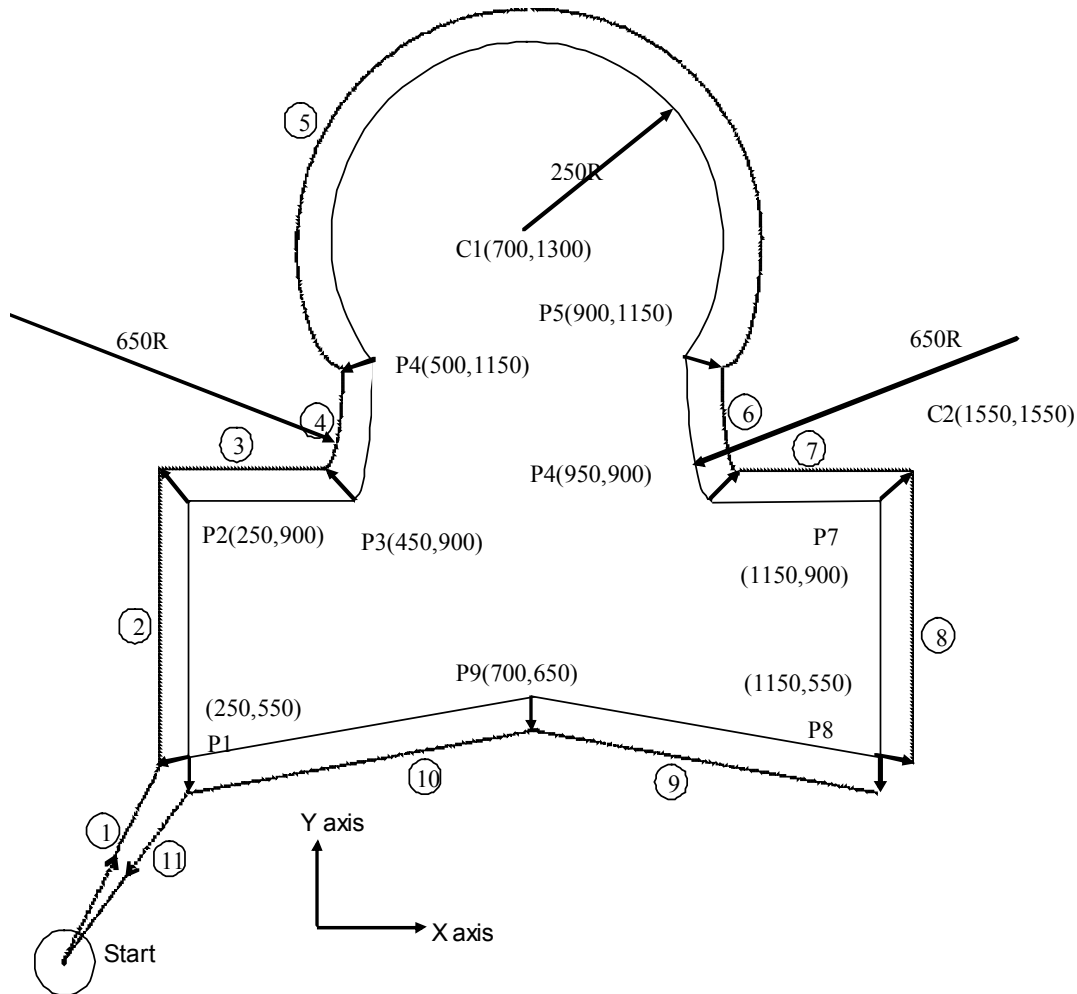
Notes 2: The tool path projected in compensation plane is compensated only when 3 axes are controlled simultaneously.

Notes 3: Change of compensation plane should be performed after cancelling the compensation mode. If it is performed in compensation mode, the system alarm will be generated, and then the machine will be stopped.

Command example:

Block (1) is named start; the compensation cancellation mode becomes compensation mode by G41 in this block. At the end of this block, tool center is compensated in the direction that tool radius is vertical to next program path (From P1 to P2). Tool compensation value is specified with D07, so set the compensation number to 7, then the G41 is indicated with tool path compensation left.

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



```

N00 G92 X0 Y0 Z0;
N01 G90 G17 G00 G41 D7 X250.0 Y550.0; (The compensation value should be pre-set with
                                         compensation number)
N02 G01 Y900.0 F150;
N03 X450.0;
N04 G03 X500.0 Y1150.0 R650.0;
N05 G02 X900.0 R-250.0;
N06 G03 X950.0 Y900.0 R650.0;
N07 G01 X1150.0;
N08 Y550.0;
N09 X700.0 Y650.0;
N10 X250.0 Y550.0;
N11 G00 G40 X0 Y0;
    
```

3.13 Tool Length Compensation (G43, G44, G49)

Format:

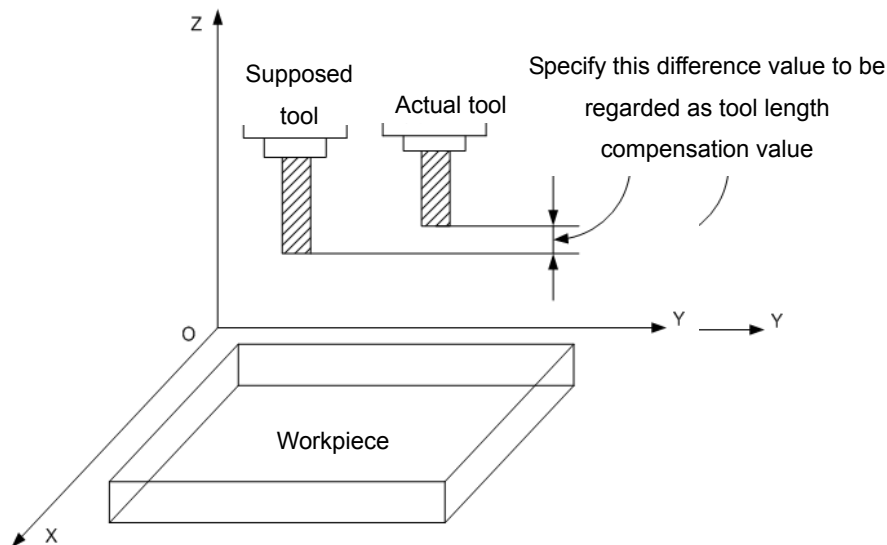
G17	}	G43	}	H__
G18		G44		
G19				

Function:

Tool length compensation function.

Explanations:

G43 and G44 are modal G codes; they are effective before meeting other G codes in the same group.



The end point specified by Z axis moves an offset value, as above figure G17 plane is selected. Difference between supposed and actual machined tool length value is pre-set at the offset storage when the program is applied. Different length tool can be employed by changing tool length compensation value, so, program change is not needed.

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

- Offset axis

The offset axes are vertical to the specified planes (G17, G18 and G19)

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

Tool position offset for two or more axes can be used to specify the offset axis and the offset axis changed by 2~3 blocks

(Example) X and Y axes compensation

G19 G43 H_ ; ...X axis offset

G18 G43 H_ ; ...Y axis offset, composed with the previous block, X and Y axes are compensated.

- Offset direction

G43: Positive offset

G44: Negative offset

Compensation axes can be regarded as Z, Y and X. Either absolute or incremental command, the end point coordinate value specified by Z axis movement command in program adds the offset specified by H codes in G43 (set in the offset storage), or subtracts the offset specified by H code in G44, finally, the value calculated is regarded as the end point coordinate.

The following command is indicated for Z axis move omitting: When the offset is positive, G43 is for an offset in the positive direction; G44 is for an offset in the negative direction.

It reversely moves when the offset is negative value.

● Specifying the offset

An offset number is specified by H code and its corresponding offset adds or subtracts Z axis movement command value in program to get a new Z axis movement command value. The offset number is H00~H32.

Offset value corresponded with offset number is pre-set in the offset storage by using the panel of LCD/MDI.

Setting range for offset is as follows:

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

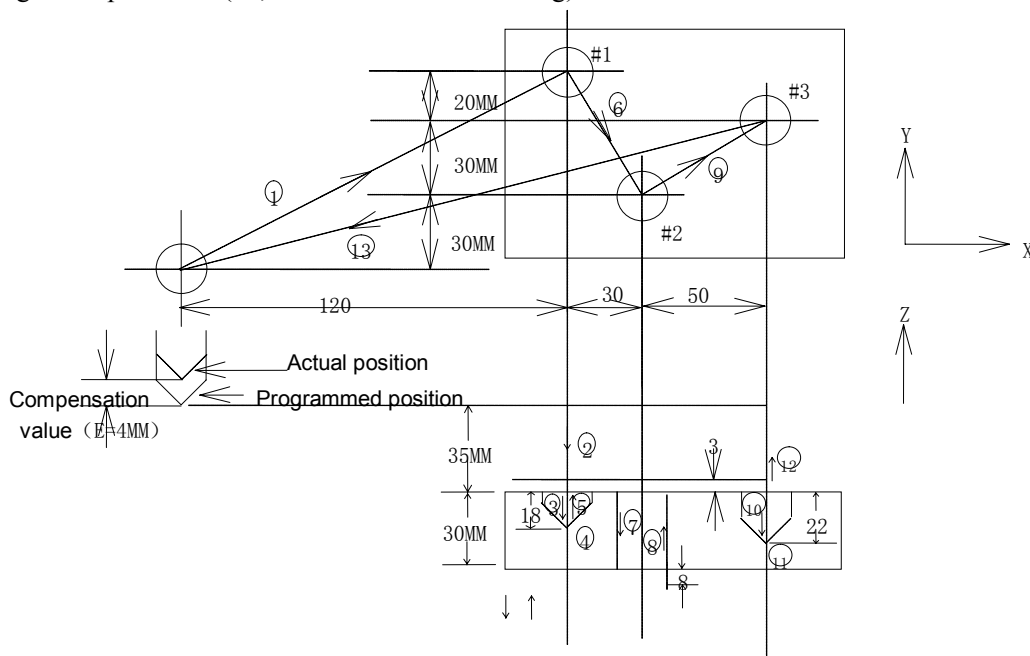
Offset number 00, i.e. H00 corresponds to the 0 offset. It is disabled to set offset value to H00.

● Tool length compensation cancellation

G49 or H00 can be specified when the tool length compensation is cancelled. When two or more axes compensations are cancelled, all of the axes compensation will be cancelled if the G49 is applied. Compensation value of the vertical axis for currently specified plane is cancelled with H00. After G49 or H00 is specified, the system immediately cancels the compensation value.

Command Example:

Tool length compensation (#1, #2 and #3 hole machining)



Offset H01=4.0

N1 G91 G00 X120.0 Y80.0;..... (1)

N2 G43 Z-32.0 H01;..... (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.

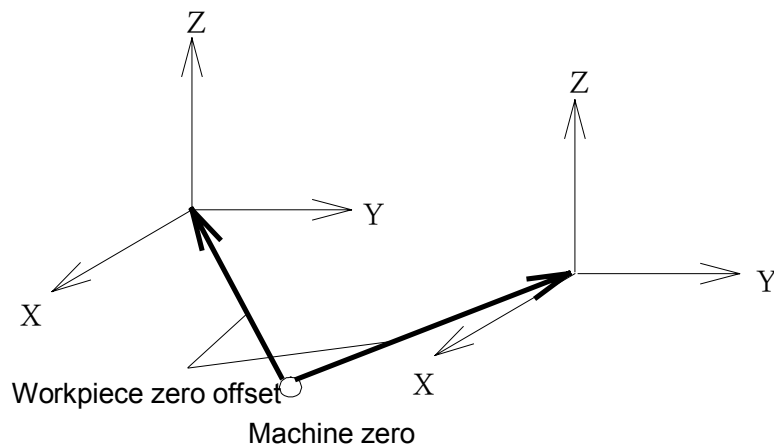
Explanations:

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.

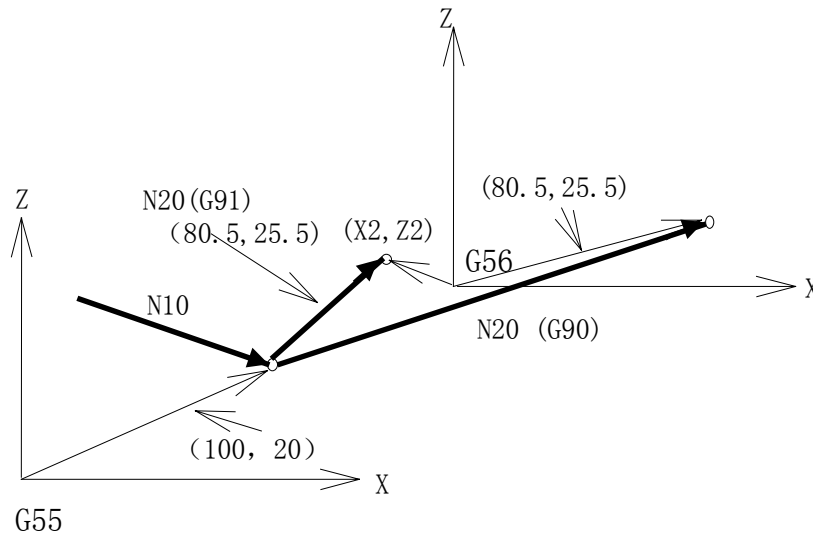


Examples:

N10 G55 G90 G00 X100.0 Z20.0;

N20 G56 X80.5 Z25.5;

Rapidly positioning to workpiece coordinate system 3 (X=80.5, Z=25.5) from workpiece coordinate system 2 (X=100.0, Z=20.0). For example, if N20 block is G91, it is incremental movement. The absolute coordinates automatically become the coordinates in coordinate system G56.



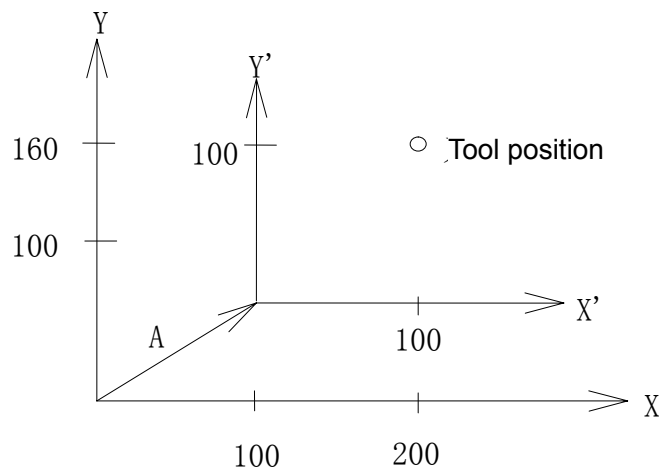
The absolute position for the figure is coordinate value under the current coordinate system.

Notes 1: The workpiece system, from 1 to 6, can be set after returning to the machine zero with the power-on. The coordinate system is determined by bit 7 (the coordinate system is restored to G54 when the power is turned off) of parameter No.013 when resetting.

Notes 2: If relative position is set with the coordinate system or not, depends on the PPD of parameter No.005, PPD is not changed when its value is 0, but, it is changed if PPD is 1.

Notes 3: Current coordinates are not changed and they are set as program zero when the command X, Y and Z are not input. It keeps the original setting value when X, Y or Z is not input.

Notes 4: Normally, the G92 coordinate system set is not needed after the workpiece coordinate system function is applied. The workpiece coordinate system 1~6 will be moved if it is set by G92. Do not intermix G92 with G54~G59, unless the workpiece coordinate system G54~G59 are moved.



If it performs G92 X100 Y100 commands when the tool is positioned at (200, 160) in the G54 coordinate system; the offset vector A for workpiece coordinate system 1 is (X', Y'). The other workpiece coordinate systems offsets for vector A.

3.15 Compound Cycle Command

3.15.1 Brief for canned cycle

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 frequently-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.

3.15.1.1 Canned cycle list

G codes	Drilling	Operation at the bottom of a hole	Retraction	Application
G73	Intermittent feed	—	Rapid feed	High-speed peck drilling cycle
G74	Feed	Dwell, spindle CCW	Feed	Left-hand tapping cycle
G80	—	—	—	Canned cycle cancellation
G81	Feed	—	Rapid feed	Drilling, point drilling
G82	Feed	Dwell	Rapid feed	Drilling, boring, counterbore
G83	Intermittent feed	—	Rapid feed	Peck drilling cycle
G84	Feed	Dwell, spindle CW	Feed	Tapping
G85	Feed	—	Feed	Boring
G86	Feed	Spindle stop	Rapid feed	Boring
G88	Feed	Dwell, spindle stop	manual	Boring
G89	Feed	Dwell	Feed	Boring
G110	Intermittent feed	Full-circle helical rough milling	Rapid feed	Round groove internal rough milling CCW
G111	Intermittent feed	Full-circle helical rough milling	Rapid feed	Round groove internal rough milling CW
G112	Feed	Full-circle fine milling	Rapid feed	Full-circle internal fine milling CCW
G113	Feed	Full-circle fine milling	Rapid feed	Full-circle internal fine milling CW
G114	Feed	Full-circle fine milling	Rapid feed	External round fine milling CCW
G115	Feed	Full-circle fine milling	Rapid feed	External round fine milling CW
G134	Intermittent feed	Rectangle rough milling	Rapid feed	Rectangle groove internal rough milling CCW
G135	Intermittent feed	Rectangle rough milling	Rapid feed	Rectangle groove internal rough milling CW
G136	Feed	Rectangle fine milling	Rapid feed	Rectangle groove internal fine milling CCW
G137	Feed	Rectangle fine milling	Rapid feed	Rectangle groove internal fine milling CW
G138	Feed	Rectangle fine milling	Rapid feed	Rectangle groove external fine milling CCW
G139	Feed	Rectangle fine milling	Rapid feed	Rectangle groove external fine milling CW

3.15.1.2 Canned circle explanations

Generally, a canned cycle consists of a sequence of the following operations, see the right figure.

Operation 1... Positioning of axes X and Y

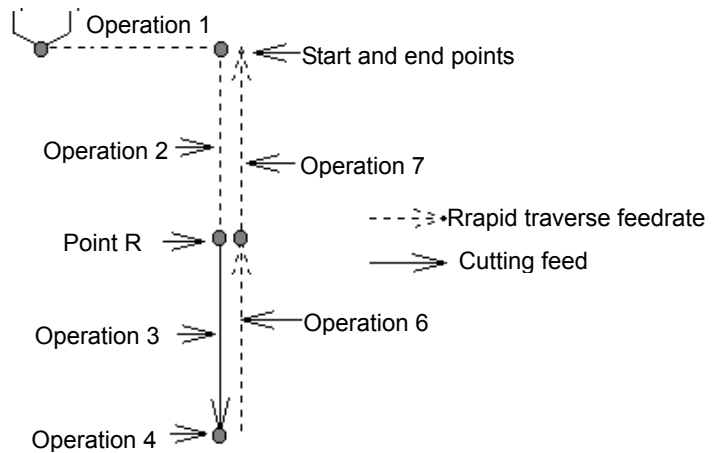
Operation 2...Rapid traverse to point R level

Operation 3...Hole machining;

Operation 4...Operation at the bottom of hole;

Operation 5...Retraction to point R level

Operation 6...Rapid traverse to the initial point



3.15.1.3 G90/G91

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

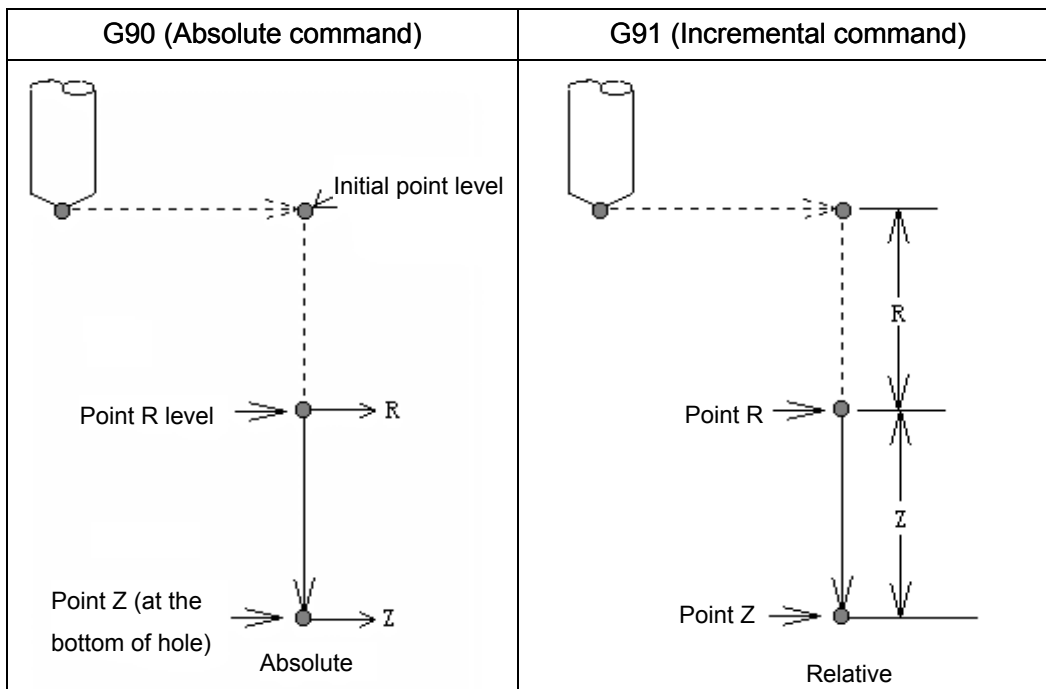


Fig. 13.1 (B) Absolute and incremental commands for canned cycle

3.15.1.4 Returning point level G98/G99

Tool can be returned to the initial level or point R level according to G98 and G99 during returning. See the following figure Fig. 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.

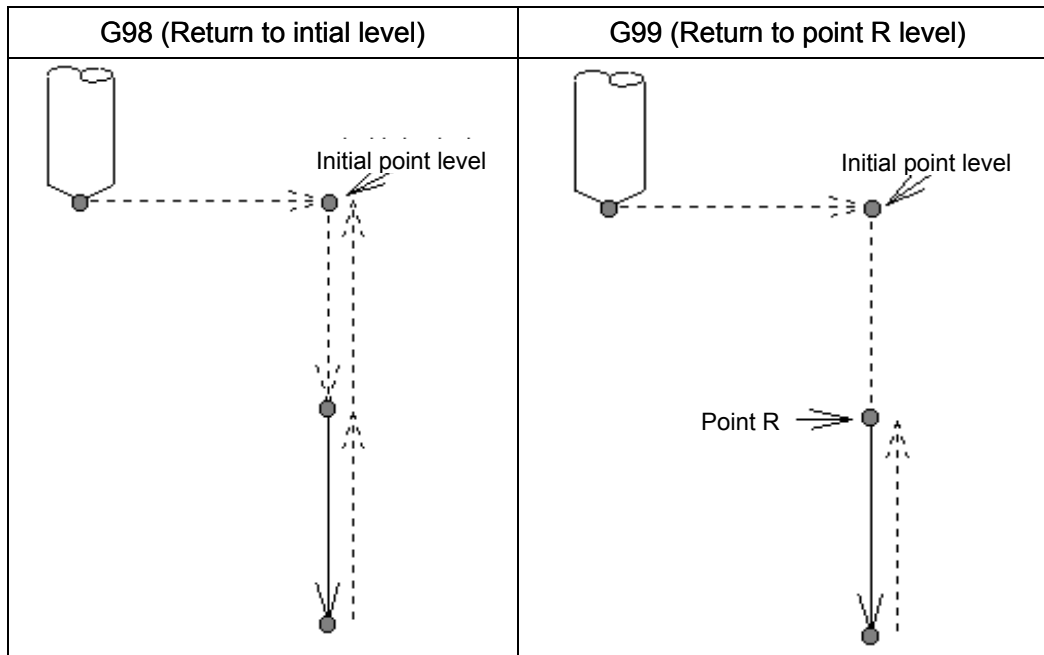
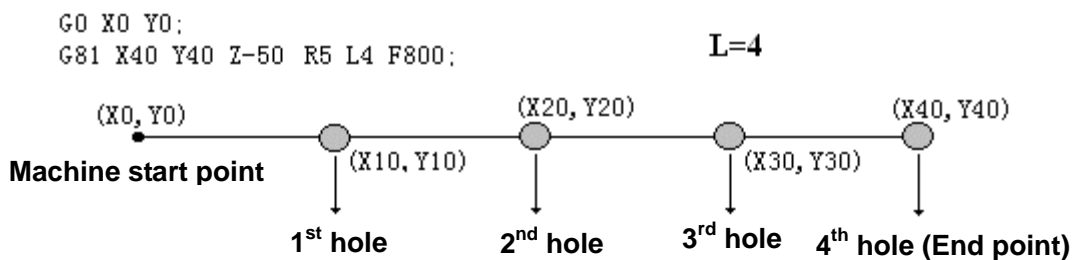


Fig.13.1 (C) Levels for initial and point R

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 Series punch (L function)

L holes machining cycle should be performed from current plane position to end point specified by X and Y are indicated if the L 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 equal-spaced, as follows:



The max. command value of L is 9999

The L command is disabled when it is negative value. Even if the negative is specified, the sign is also invalid.

It is normal operation when the L or L=1 is not specified, namely, to machine once only; if the L is equal to 0, then the drilling is not performed, so the tool is not moved, but the related canned cycle modal data is saved

Note 1: The command word L is effective only in current program.

Note 2: The returned levels are all point R in series punch, as long as the last hole is finished, can the corresponding plane be returned to by G98/G99 specified in block.

Note 3: The canned cycle G110, G111, G112, G113, G114, G115, G134, G135, G136, G137, G138 and G139 have no series punch functions.

3.15.1.6 Canned cycle cancellation

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

1. Cancelling 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.

3.15.1.7 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 13.1.7 Command explanations for canned cycle

Specifying content	Address	Explanation for command address
Hole machining	G	Refer to the canned cycle list.
Hole position data	X, Y	Specifying the hole position with the absolute and incremental value, control is same with G00 position. Unit: mm;
Hole machining data	R	See the fig.13.1 (B), the distance from initial point level to point R level is specified by using the incremental value, or specifying the coordinate value of the point R by absolute value. Unit: mm;
	Z	Hole 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 specifying the coordinate value of the hole bottom by absolute value. Unit: mm;
	Q	Specifying each cut-in in G73 and G83 or translational value in G76 and G87. Unit: mm;

	P	Specifying the dwell at the bottom of a hole. Relation of time and the numerical specified are same with G04. Unit: ms;
	L	Machining cycle for L holes are performed from start (start position of block) to XY coordinate position.
	F	The cutting feedrate is specified, tooth pitch is indicated in G74 and G84.

A part of command of canned cycle such as G110, G111, G112, G113, G114, G115, G134, G135, G136, G137, G138 and G139 are explained in the following chapters or sections.

3.15.2 Description for canned cycle

3.15.2.1 High-speed peck drilling cylce G73

Format: G98/G99 G73 X_Y_R_Z_Q_F_L_;

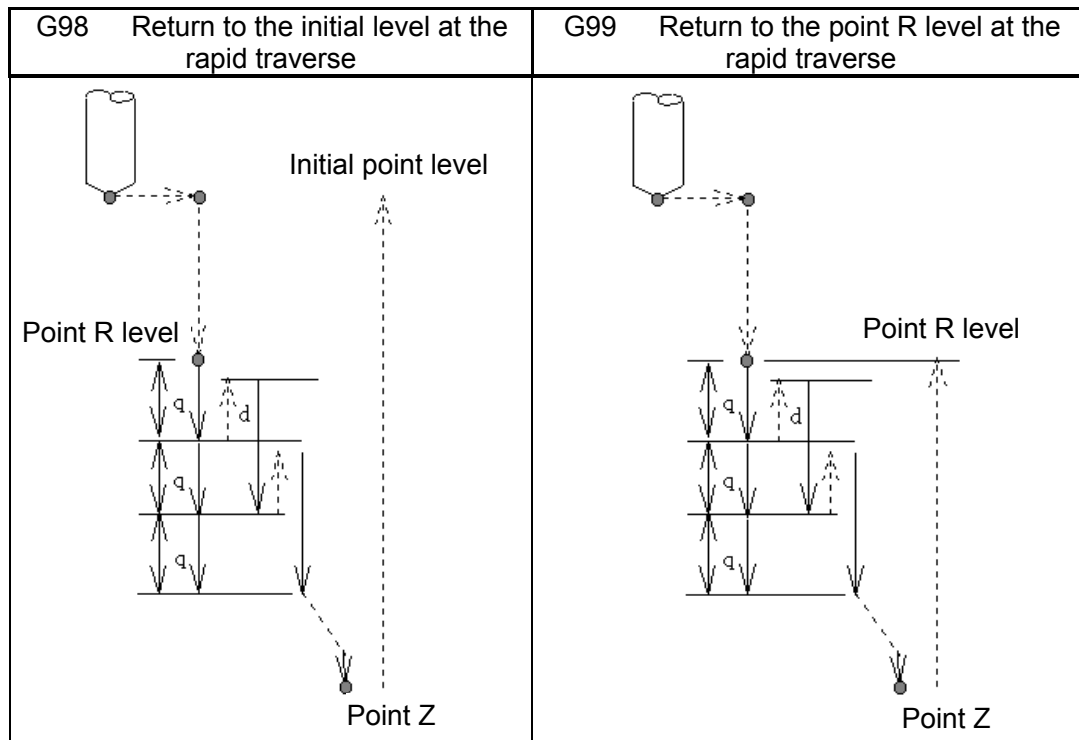
Function: This kind of cycle performs high-speed peck drilling, it performs intermittent cutting feed to the bottom of a hole, and eliminating the chips from the hole simultaneously.

Explanation: Refer to the command explanation of canned cycle in Table 13.1.7.

Cycle process:

- (1) Rapid positioning to XY plane level;
- (2) Down to the point R level at the rapid traverse rate;
- (3) Cutting feed for Q distance;
- (4) Retract d distance in rapid traverse;
- (5) Cutting feed for (Q+d) distance
- (6) Machine to the Z axis hole bottom by cycling the (4) and (5);
- (7) Return to the start point level or point R level according to G98 or G99 at the rapid traverse.

Command Path:



Related Explanations:

(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.

3.15.2.2 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.

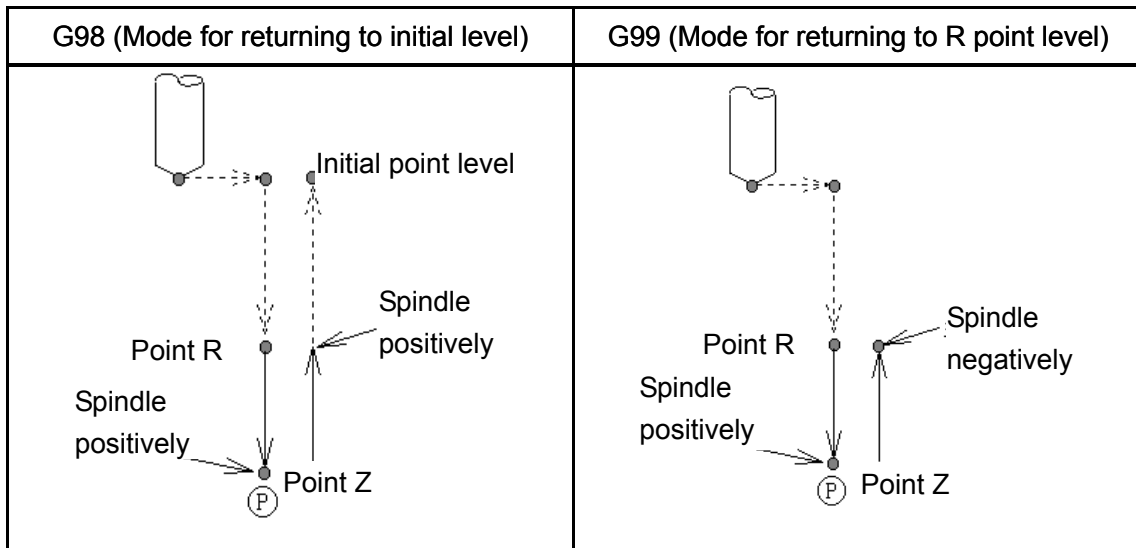
Explanation: For canned cycle explanation, see the Table 13.1.7

Thereinto, the F is indicated for tooth pitch. The value range are indicated as 0.001~500.00mm (metric), 0.06~25400 teeth/inch (inch)

Cycle process:

- (1) Positioning to XY plane level at the rapid traverse;
- (2) Down to the point R level 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 level;
- (7) The spindle is stopped; pause for time P if dwell is specified;
- (8) Spindle rotates CW;
- (9) Return to the initial level if it is G98.

Command Path:



Related Explanations:

- (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.

3.15.2.3 Drilling cycle, spot drilling cycle G81

Format: G98/G99 G81 X_ Y_ R_ Z_ F_ L_ ;

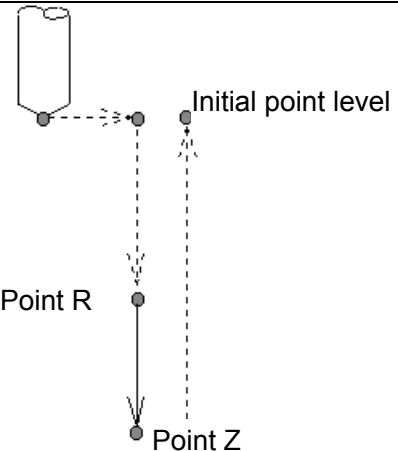
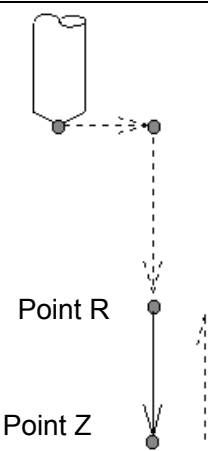
Function: This cycle is used for normal drilling. Cutting feed is performed to the bottom of the hole, the tool is then retracted from the bottom of the hole in rapid traverse.

Explanation: For the command explanation of canned cycle, see the Table 13.1.7.

Cycle Process:

- (1) Positioning to the XY plane level position at the rapid traverse;
- (2) Down to the point R level at the rapid traverse;
- (3) Cutting feed to the bottom of the hole;
- (4) Returning to the initial point or point R level at rapid traverse according to the G98 or G99;

Command Path:

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

3.15.2.4 Drilling cycle, counterboring cycle G82

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

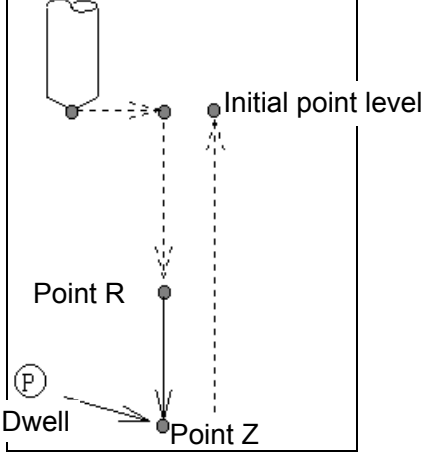
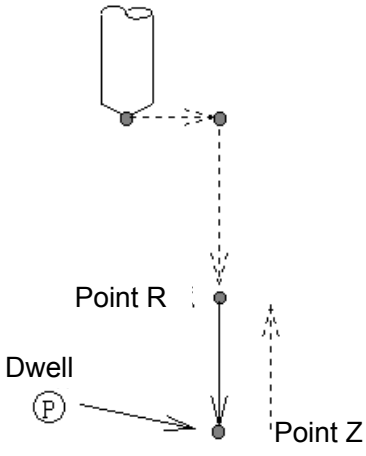
Function: Cutting feed is performed to the bottom of the hole. Hole depth precision is added when the dwell is performed, and then the tool is retracted from the bottom of the hole at rapid traverse.

Explanation: For the command explanation of these canned cycles, see the Table 13.1.7

Cycle process:

- (1) Positioning to the XY plane level at the rapid traverse;
- (2) Down to the point R level 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 level 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 level at the rapid traverse
	

Related Explanations:

(1) They are basically 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.

3.15.2.5 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 13.1.7.

Cycle Process:

- (1) Positioning to the XY plane level at the rapid traverse;
- (2) Down to the point R level at the rapid traverse;
- (3) Cutting feed for Q distance;
- (4) Retract to the point R level at the rapid traverse;
- (5) Rapid feed to d distance to the end surface
- (6) Cutting feed for (Q+d) distance;
- (7) Cycling (4) (5) and (6) to the bottom of a hole along Z-axis;
- (8) Return to the initial point or point R level according to the G98 or G99 at the rapid traverse;

Command Path:

G98 returned to the initial level at the rapid traverse	G99 returned to the point R level at the rapid traverse
<p>Initial point level</p> <p>Point R level</p> <p>Point Z</p>	<p>Point R level</p> <p>Point Z</p>

Related Explanations:

- (1) Same as the G73, after feeds for Q, returning to the point R level at the rapid traverse firstly, and then rapid feeding to dmm 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.

3.15.2.6 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.

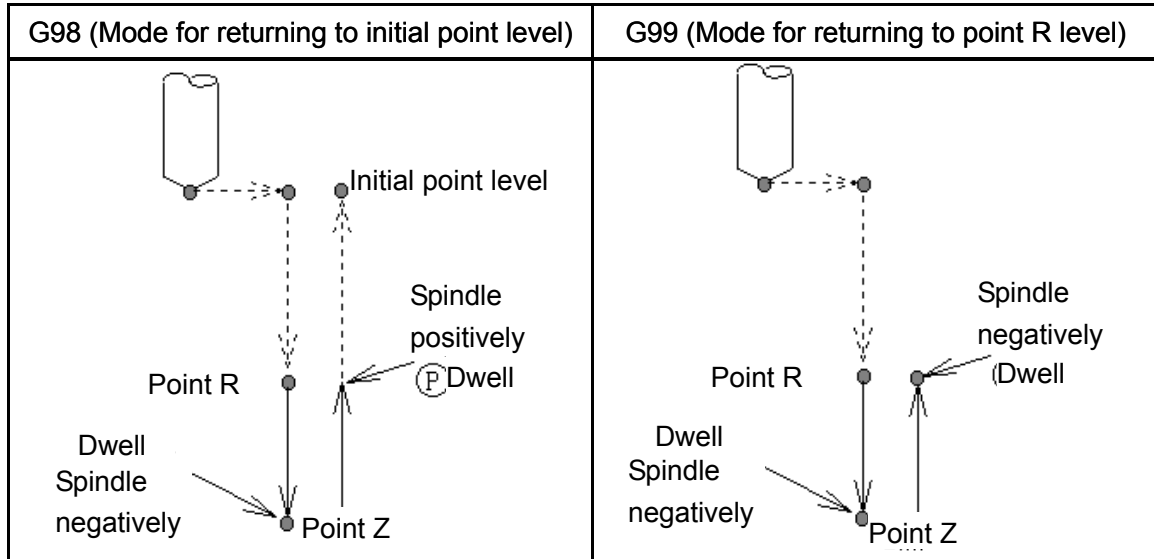
Explanations: For command explanation of canned cycle, see the Table 13.1.7

Thereinto, the F is tooth-pitch. The value range is 0.001~500.00mm (metric), 0.06~25400 tooth/inch (inch).

Cycle Process:

- (1) Positioning to the XY plane level at the rapid traverse;
- (2) Down to the point R level 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 level 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:



Related Explanation:

Please refer to the related explanation for G74 (Counter tapping cycle)

3.15.2.7 Boring cycle G85

Format: G98/G99 G85 X_ Y_ R_ Z_ F_ L_ ;

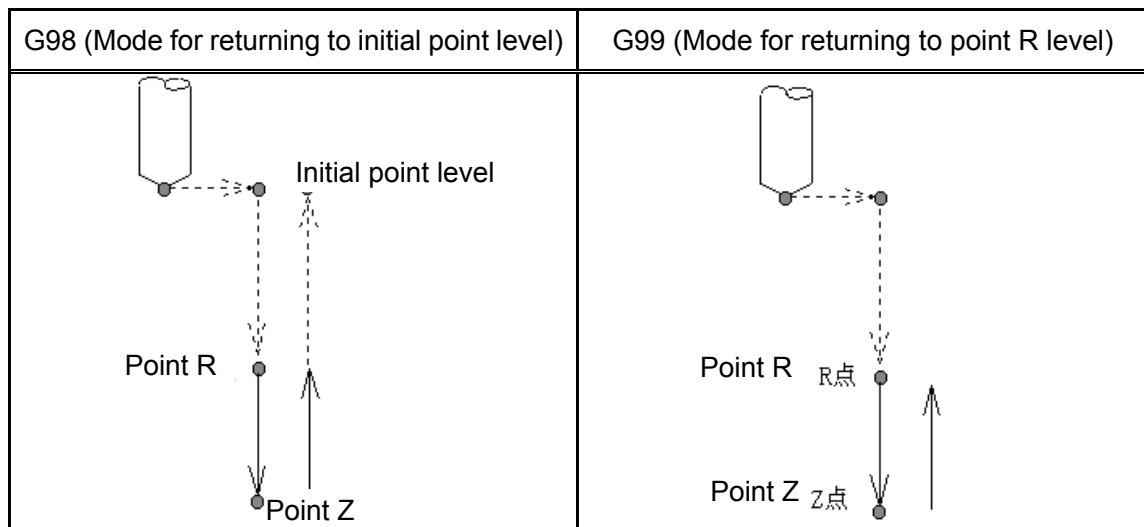
Function: After positioning along X and Y axes, rapid traverse is performed to point R; the boring is performed from point R to point Z thereafter. Cutting feed is performed to return point R level when the Z point has been reached the bottom of a hole.

Explanation: Command explanation for the canned cycle, see the table 13.1.7.

Cycle process:

- (1) Positioning to the XY plane level at the rapid traverse;
- (2) Down to the point R level at the rapid traverse;
- (3) Cutting feed to the bottom of a hole;
- (4) Cutting feed to the point R level;
- (5) Returning to the initial point level if it is G98;

Command Path:



Related Explanations:

- (1) This cycle is used to bore a hole. The command motion is basically same as the G81 (Drilling, Spot-drilling cycle), the difference is that by the G81 it returns to the point R level in rapid traverse rate, while by the G85 it returns to the point R level 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.

3.15.2.8 Boring cycle G86

Format: G98/G99 G86 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.

Explanation: For command explanation for canned cycle, see the table 13.1.7.

Cycle process:

- (1) Positioning to the XY plane level at the rapid traverse;
- (2) Down to the point R level at the rapid traverse;
- (3) Cutting feed to the bottom of a hole;
- (4) The spindle stops;
- (5) Returning to the initial point or point R level at rapid traverse according to the G98 or G99;
- (6) The spindle is rotated in the positive direction;

Command Path:

G98 (Mode for returning to start point level)	G99 (Mode for returning to point R level)
<p>Spindle positively Initial point level Point R Spindle stop Point Z</p>	<p>Initial point level Spindle positively Point R Spindle stop Point Z</p>

Related Explanations:

- (1) This cycle is used to 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 M05 is executed (spindle stops), then the point R level is retracted at the rapid traverse, the M03 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.

3.15.2.9 Boring cycle G88

Format: G98/G99 G88 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

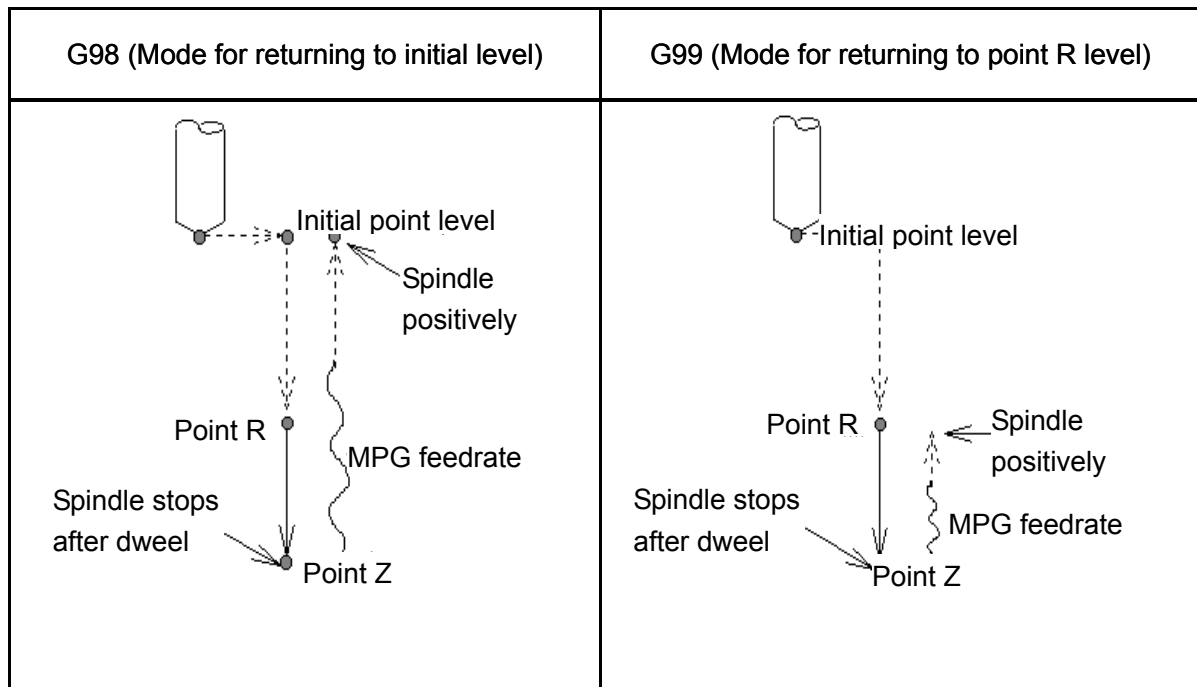
applied now, tool can be removed manually. It is better to retract the tool safely from the hole regardless of any kind of manual operation. It is rapidly retracted to point R or initial level when the automatic operation is performed again, the spindle is stopped and G88 is finished.

Explanation: For the command explanation of the canned cycle, see the table 13.1.7.

Cycle process:

- (1) Positioning to the XY plane at the rapid traverse rate;
- (2) Down to the point R level at the rapid traverse rate;
- (3) Cutting feed to the bottom of hole;
- (4) The spindle is stopped;
- (5) P time is delayed if it is specified.
- (6) Manual operation will be performed if the dwell is executed.
- (7) Restoring the automatic mode, retracting to initial point or poin R level according to the G98 or G99 at the rapid traverse rate.
- (8) The spindle rotates positively;

Command Path:



Related Explanation:

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

3.15.2.10 Boring cycle G89

Format: G98/G99 G89 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.

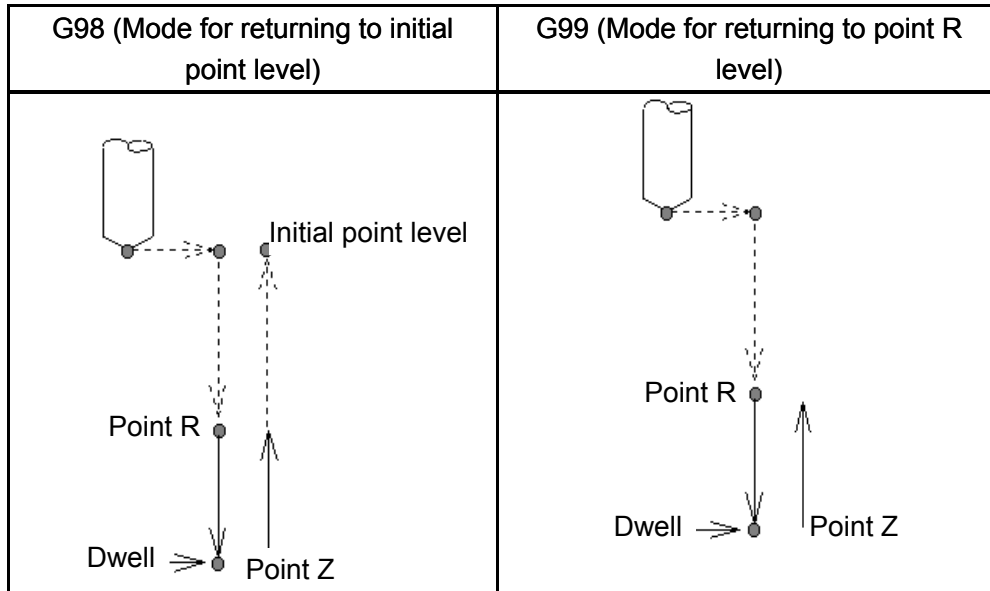
Explanation: For The command explanation of the canned cycle, see the table 13.1.7.

Cycle process:

- (1) Positioning to XY plane at the rapid traverse rate;
- (2) Down to the point R level at the rapid traverse rate;
- (3) Cutting feed to the bottom of a hole;

- (4) For dwell time P if the P is specified;
- (5) Cutting feed to the point R level;
- (6) Returning to the initial point level if it is G98;
- (7) Returning to the initial point or point R level at the rapid traverse according to the G98 or G99;

Command Path:



Related Explanations:

- (1) G89 (Boring cycle) is basically same as the G85, a dwell is applied at the bottom of a hole (Dwell 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.

3.15.2.11 Rectangle series punch G140/G141

Format:

G140
G98/G99 **Gxx X_ Y_ R_ Z_ A_ B_ J_ F_**
G141

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

Explanations:

- G140 – Punching in CW
- G141 – Punching in CCW
- Gxx – Punching type (G73, G74, G81, G83, G84, G85, G86, G88, G89)
- X, Y – End coordinate of the first rectangle side
- R – R level position
- Z – Hole depth
- A – The punching number on the 1st and 3rd side
- B – The punching number on the 2nd and 4th side
- J- The length of the 2nd side
- F – Cutting feedrate

Related Parameter:

Bit 7 of the parameter 014

- 1: Hole positioning of serial punching is performed by cutting path (G01~G03).
- 0: Hole positioning of serial punching is performed by the rapid traverse path (G00).

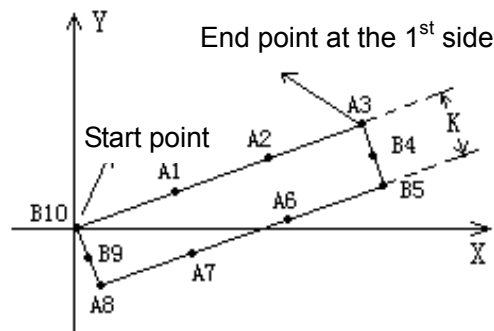
For example:

The end point coordinate of the rectangle first side is X90, Y40; 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 4th side each other, the hole depth is 25mm;

Rectangle punching A=3,
B=2, K=20

Its programming is as follows:

```
G90 G17 G0 X0 Y0 Z25;
M03;
G140 G81 X90 Y40 R5 Z-25 A3 B2 J20 F800;
G80 G0 X100 Y100 M05;
M30
```



There are 10 holes such as A1~A3, B4, B5, A6~A8, B9 and B10 to be machined as in above figure.

Note 1: If the G140 or G141 is specified in the canned cycle, it is indicated that the rectangle serial punching will be performed. The rectangle data are defined according to specified X, Y coordinates and J value in a program, and the serial punching cycle is performed according to the punch mode (canned cycle command).

Note 2: The command value of maximum punching number A and B at each side is 9999; the command is disabled when it is negative. The decimal part will be rounded off if the command is decimal; if the A or B is not specified, then 0 is a default.

Note 3: The rectangle is defined by the current start point, the end of the 1st side and the length of the 2nd side; the default is current start point if the end of 1st side is not specified; the alarm will be generated if the length (namely, the J is not specified) of 2nd side is not specified.

Note 4: The returned levels are all R point plane in serial punching, the corresponding plane will be retracted according to G98/G99 specified in a block when the last hole is performed.

Note 5: Canned cycles, such as G110, G111, G112, G113, G114, G115, G134, G136, G137, G138 and G139 has no serial punching functions.

Note 6: The command words G140, G141, A, B and J are only effective in current block. The alarm will be generated if the G140 and G141 are specified without the canned cycle (punching). The A, B and K will be ignored if A, B and K are specified instead of the G140 or G141.

3.15.2.12 Arc serial punching G142/G143

Format:

G142

G98/G99 **Gxx** X_ Y_ R_ Z_ B_(I_ J_) C_ F_

G143

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

Explanations:

- G142 – Punching in CW
- G143 – Punching in CCW
- Gxx – Punching type (G73, G74, G81, G82, G83, G84, G85, G86, G88, G89)
- X, Y – End point coordinate for the arc, it is fixed for G17 plane.
- R – R level position
- Z – Hole depth
- B – Radius of arc, when a negative value is specified, it is major arc.
- (I_ J_) – The circle center and radius are calculated by I or J when the R value is not specified.
- C – Number of punching
- F – Cutting feedrate

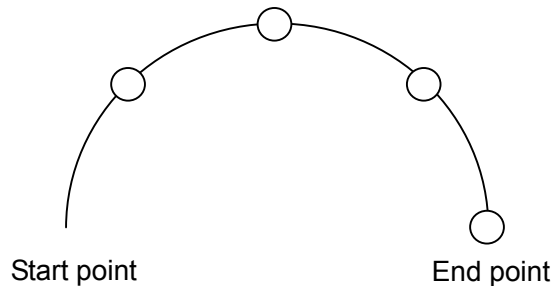
Related Parameter:

Bit 7 of the parameter 014

- 1: Hole positioning for serial punching is performed by cutting path (G01~G03).
- 0: Hole positioning for serial punching is performed by the rapid traverse path (G00).

For example:

G91 G142 G81 X100 R50 Z-50 C4



Note 1: The returned planes are all R point level in serial punching, the corresponding plane will be returned according to the G98/G99 specified in the block when the last hole is performed.

Note 2: Canned cycles, such as G110, G111, G112, G113, G114, G115, G134, G136, G137, G138 and G139 have no serial punching functions.

3.15.2.13 Groove rough milling inside the round G110/G111

Format:

G110

G98/G99 X_ Y_ R_ Z_ I_ W_ Q_ K_ V_ D_ 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.

Explanations: For command explanation of the canned cycle, see the table 13.1.7.

G110: Groove rough-milling inside the round in CCW;

G111: Groove rough-milling inside the round in CW;

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;

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

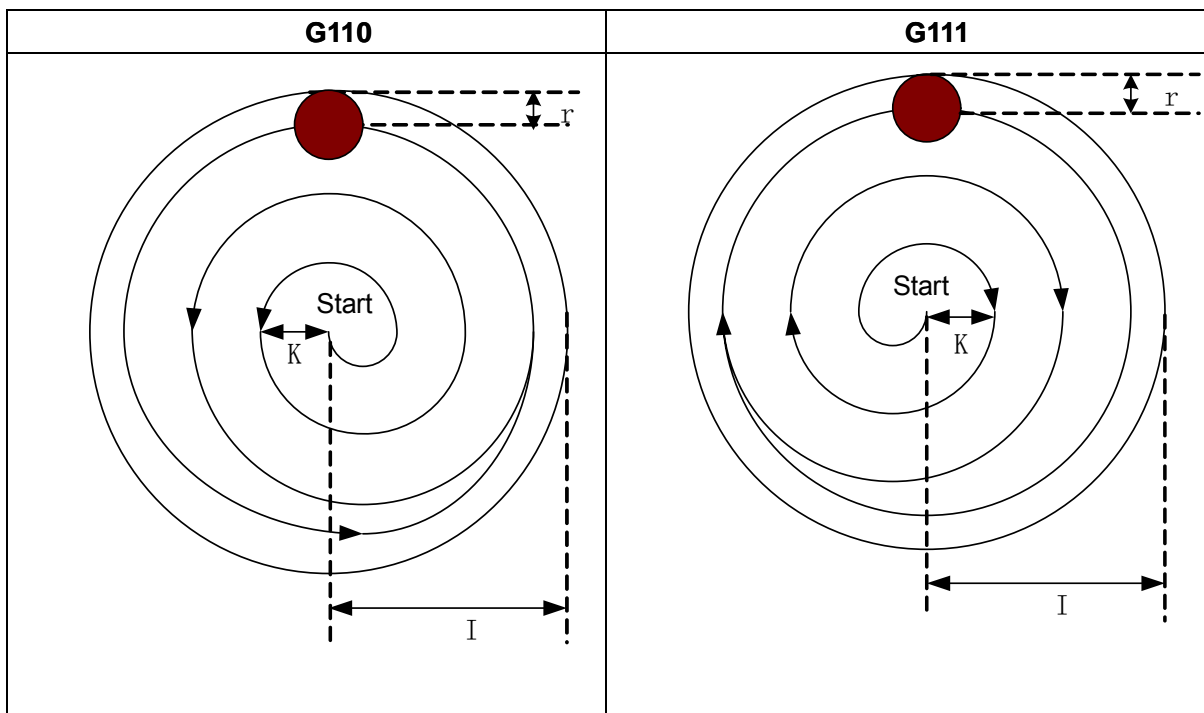
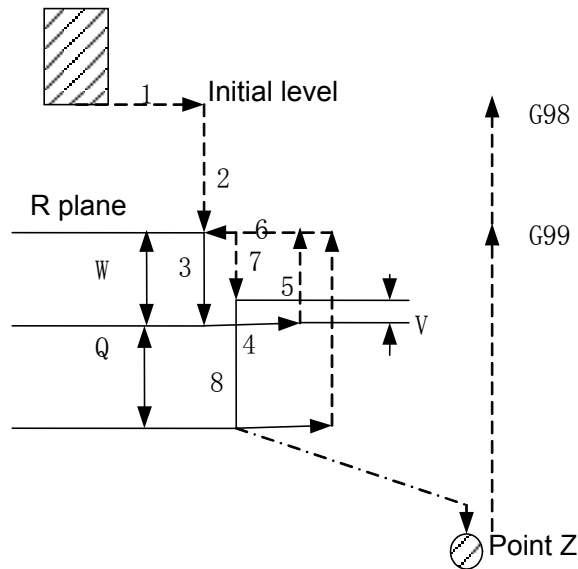
V: The distance to the end machining plane at the rapid traverse, it should be more than 0 when cutting;

D: Tool radius serial number, the value range is 0~32, 0 is the default of D0. The current tool radius is determined by the specified serial number.

Cycle process:

- (1) Positioning to the XY plane level at the rapid traverse rate;
- (2) Down to the point R level at the rapid traverse rate;
- (3) Cut W depth downwards in cutting feedrate
- (4) Mill a round face with radius I helically by K increment each time from center point to outside.
- (5) The Z axis is retracted to the R reference surface at the rapid traverse rate;
- (6) X and Y axes are positioned to the center 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 operations from (4) ~ (8) till the round surface of total depth is finished.
- (10) Return to the initial level or point R level according to G98 or G99.

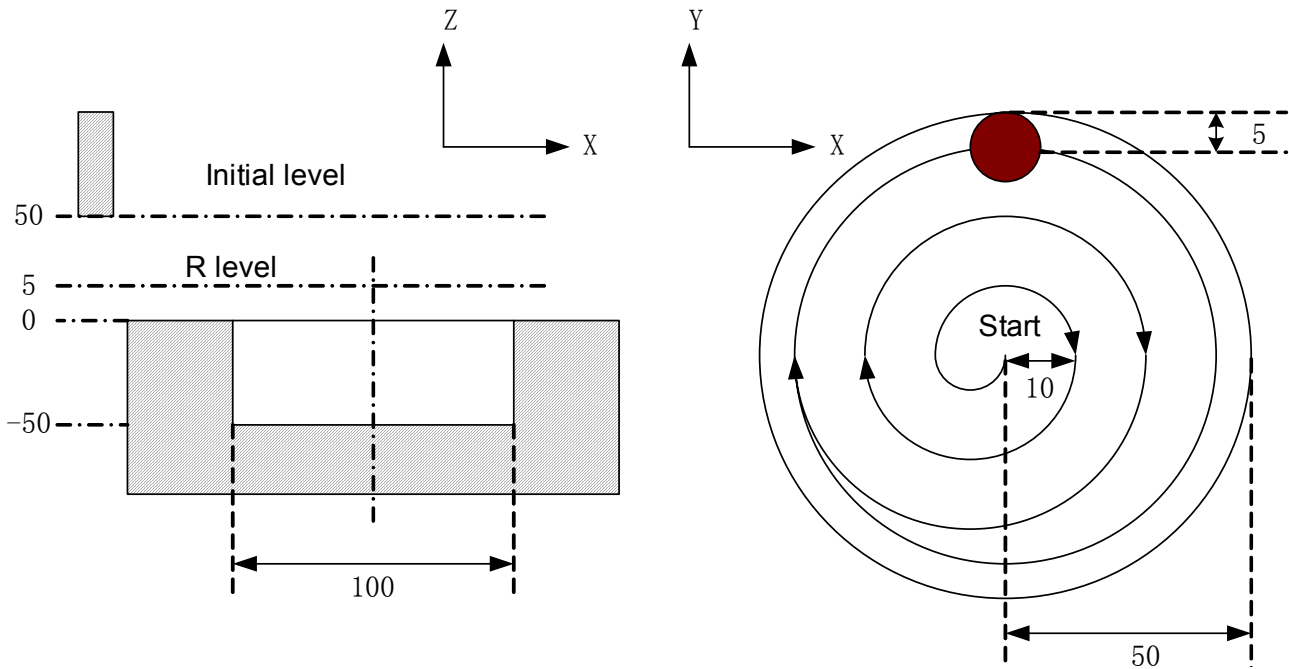
Command Path:



Related Explanation:

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

For example: A round inside groove rough-milling is specified in canned cycle G111, see the following figure



G90 G00 X50 Y50 Z50; (G00 positioning at the rapid traverse rate)

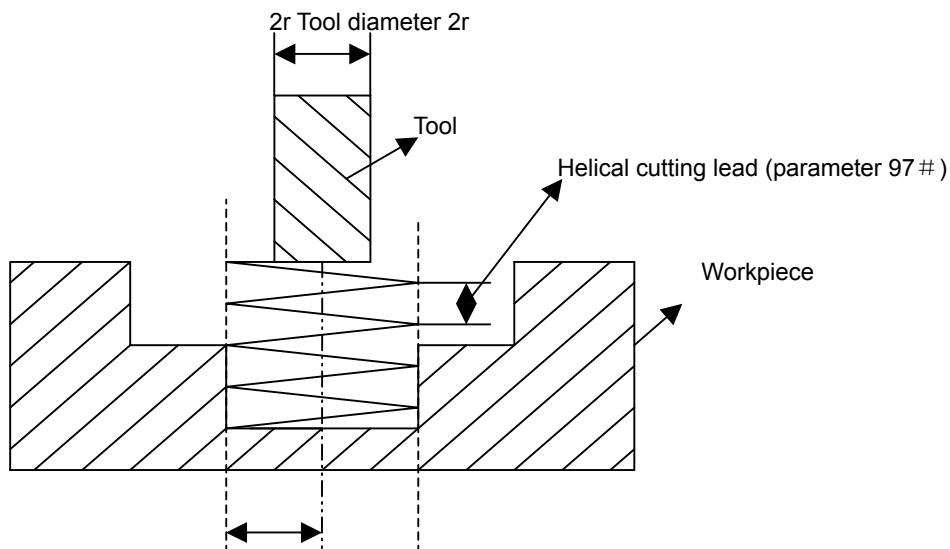
G99 G111 X25 Y25 R5 Z-50 150 W20 Q10 K10V10 F800 D1; (Rough-milling cycle inside the round groove D1=5)

G80 X50 Y50 Z50; (Cancelling canned cycle, returning from the point R level)

M30;

Note: Set the 97# parameter value to one which is more than 10, by G110 and G111 it feeds helically along Z axis. Rough-milling machining can be directly performed for non-groove workpiece.

See the following figure figure for helical cutting path:



3.15.2.14 Fine-milling cycle inside full circle G112/G113

Format:

G112
G98/G99 **X_ Y_ R_ Z_ I_ J_ D_ 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.

Explanations: For command explanation of canned cycle, see the table 13.1.7.

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

G113: Fine-milling cycle inside the full circle in CW.

I: Fine-milling circle radius, the value range is indicated as 0~9999.999mm, the absolute value is taken when it is negative.

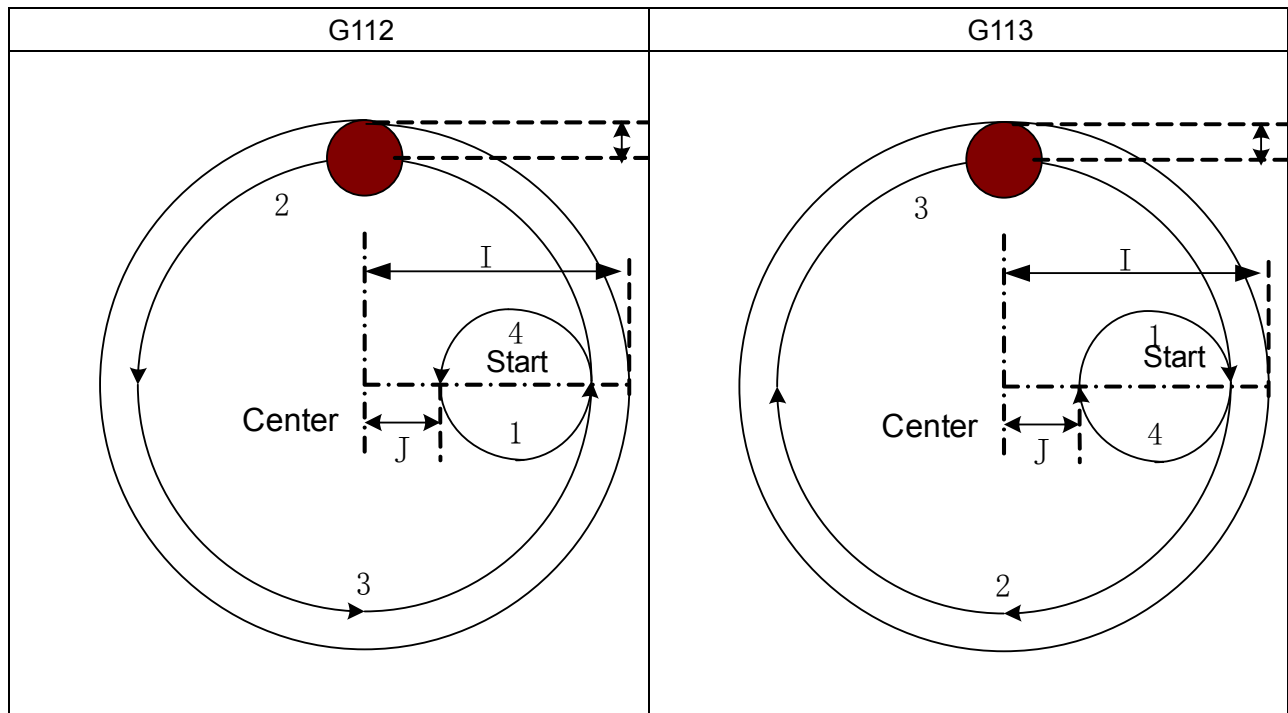
J: Fine-milling distance from start point to the center point, the value range is indicated as 0~9999.999mm, the absolute value is taken when it is negative.

D: Sequence number of tool radius, the value range is indicated as 0~32, the 0 is 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 P level at the rapid traverse rate;
- (3) 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 level according to G98 or G99.

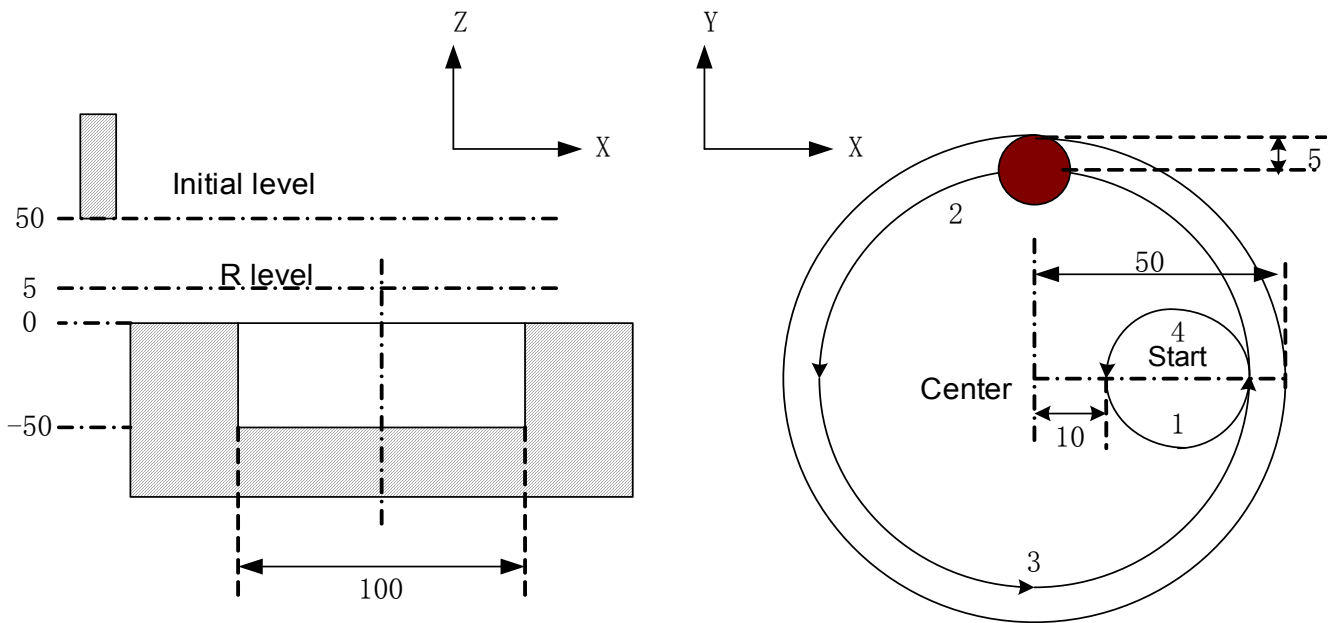
Command Path:



Related Explanation:

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

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



G90 G00 X50 Y50 Z50; (G00 rapid positioning)

G99 G112 X25 Y25 R5 Z-50 150 J10 F800 D1; (Start canned cycle, fine-milling cycle inside the circle at the bottom of a hole D1=5)

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

M30;

3.15.2.15 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.

Explanations: For command explanation of canned cycle, see the table 13.1.7.

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 0~9999.999mm, the absolute value is taken when it is negative.

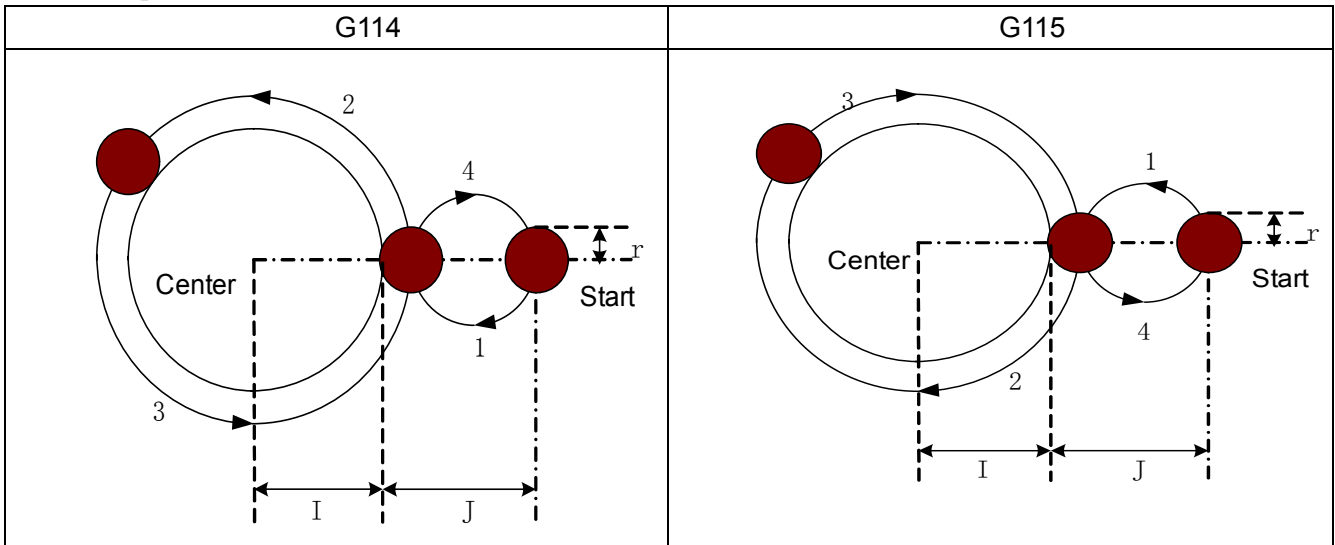
J: Distance of fine-milling between the start point and the circle, the value range is indicated as 0~9999.999mm; the absolute value is taken when it is negative.

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 level 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 level according to G98 or G99.

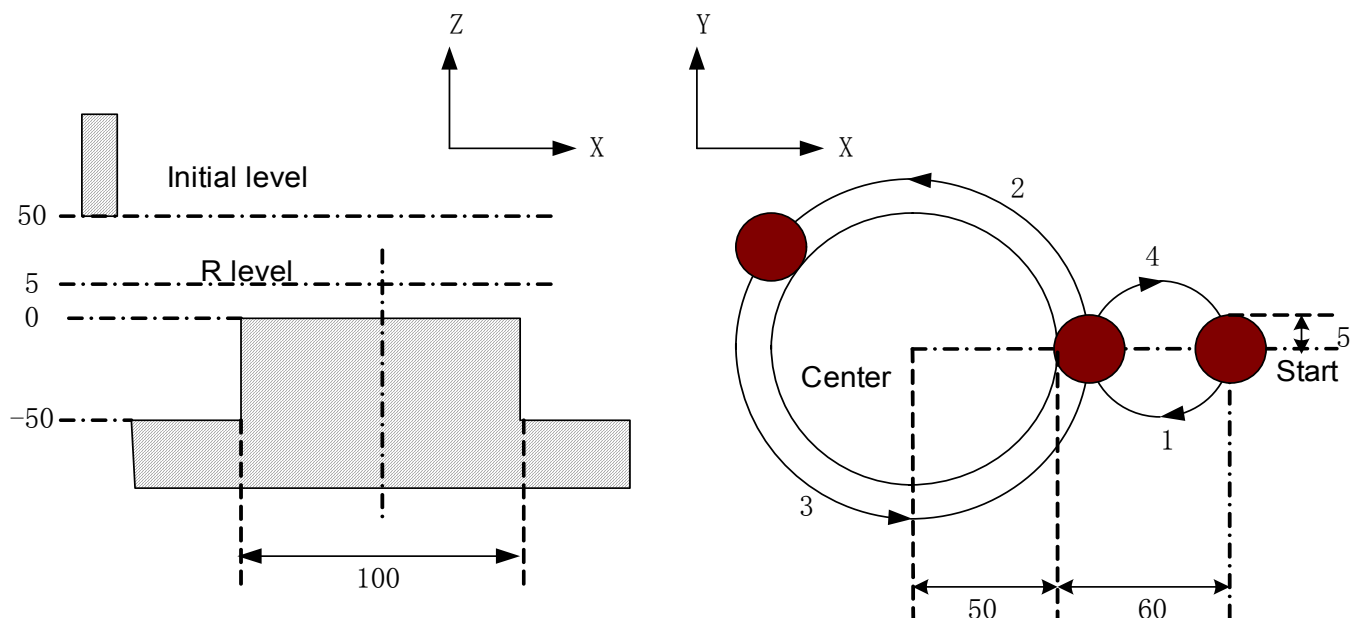
Command path:



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 L are disabled in this cycle, but the Q and P value are reserved as canned cycle modal value.

For example: A finished rough-milling round groove is performed by fine-milling with the canned cycle G114 command, see the following figure figure:



```
G90 G00 X50 Y50 Z50;    (G00 rapid positioning)
G99 G114 X25 Y25 R5 Z-50 150 J60 F800 D1;    (Start canned cycle, the fine-milling cycle
is performed outside the circle at the bottom of a hole D1=5)
G80 X50 Y50 Z50;    (The canned cycle is cancelled, returning from the point R level)
M30;
```

3.15.2.16 Rectangle groove rough-milling G134/G135

Format:

G134
G98/G99 X_ Y_ Z_ R_ I_ J_ K_ W_ Q_ V_ U_ D_ 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.

Explanations: For command explanation of canned cycle, see the table 13.1.7.

G134: Rectangle groove rough-milling in CCW

G135: Rectangle groove rough-milling in CW

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

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

K: 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.

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

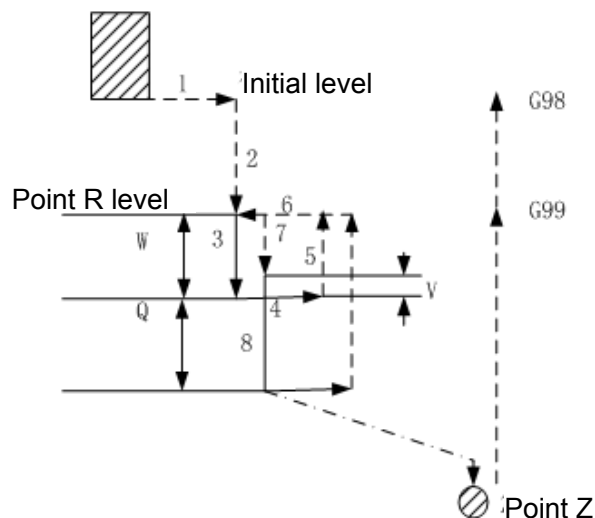
U: Corner arc radius, if it is omitted, that is no corner arc transition is not shown.

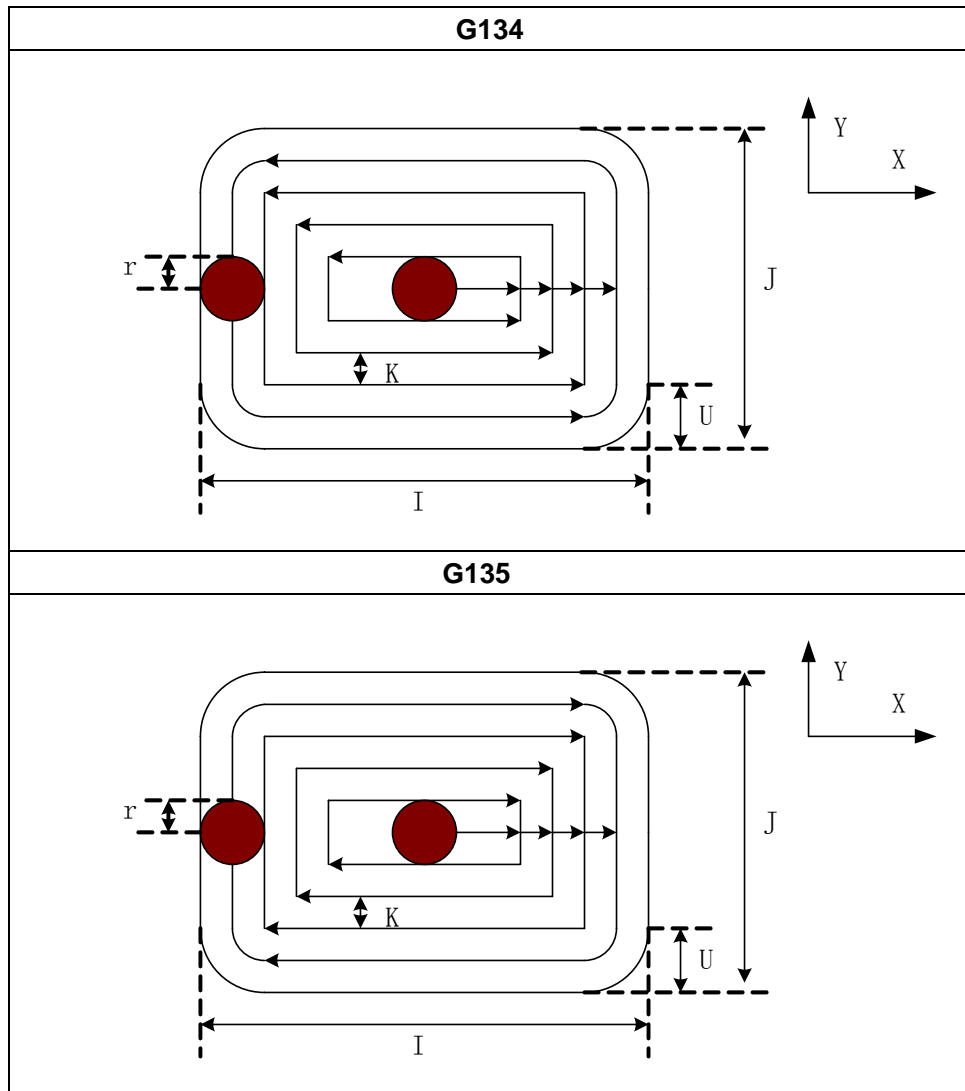
D: Sequence number of tool radius, its value range is indicated as 0 ~ 32, thereinto, the 0 is default of D0. The current tool radius value is taken out according to the specified sequence number.

Cycle process:

- (1) Positioning to the XY plane at the rapid traverse rate;
- (2) Down to the point R level at the rapid traverse rate;
- (3) W distance depth is cut downwards by cutting feedrate
- (4) Mill a rectangle face helically by K 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 level or point R level according to G98 or G99.

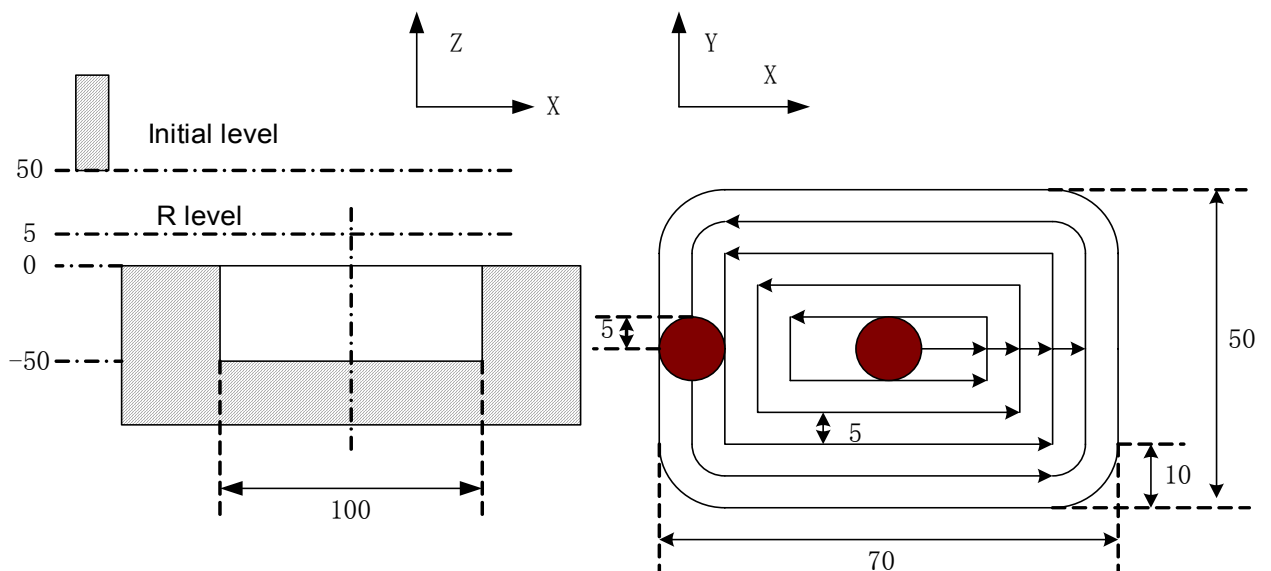
Command Path:



**Related Explanation:**

The commands P and L 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 G134 in canned cycle, see the following figure:



G90 G00 X50 Y50 Z50; (G00 rapid positioning)

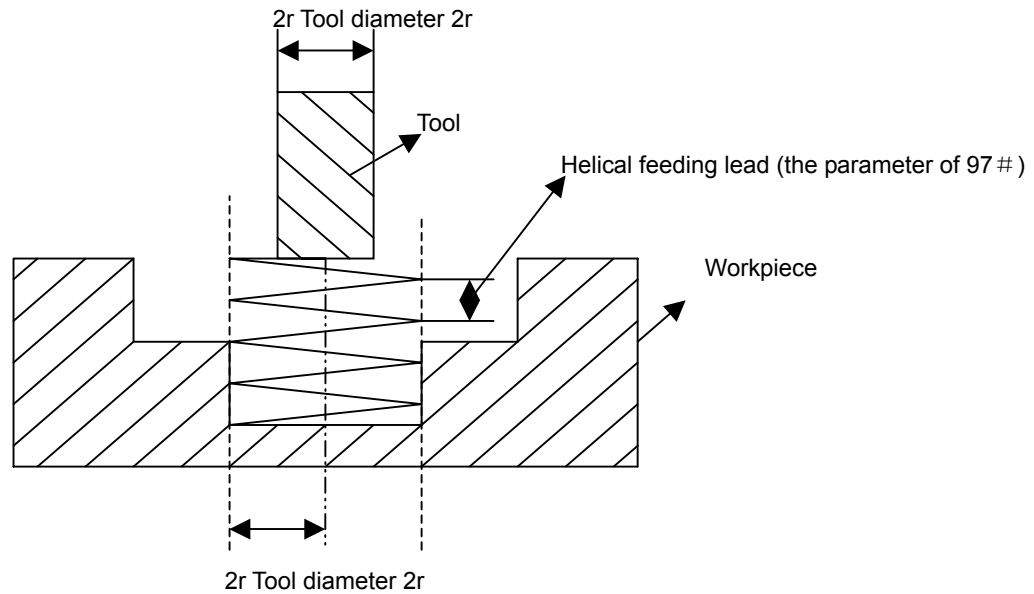
G99 G134 X25 Y25 R5 Z-50 I70 J50 W20 Q10 K5 V10 U10 F800 D1; (Groove rough-milling cycle inside rectangle is performed D1=5)

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

M30;

Note 1: If the parameter value of 97# is set for more than 10, the helical cutting feed along the Z axis will be performed by G110 and G111. So, the workpiece without groove can be machined by rough-milling directly.

The helical feeding path is as follows:



3.15.2.17 Rectangle groove inner fine-milling cycle G136/G137

Format:

G136

G98/G99

X_ Y_ R_ Z_ I_ J_ D_ K_ U_ F_;

G137

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

Explanation: For command explanation of canned cycle, see the table 13.1.7.

G136: Finish-milling cycle inside groove of rectangle in CCW.

G137: Fininsh-milling cycle inside groove of rectangle in CW.

I: The rectangle width along the X axis, the value range is indicated as 0~9999.999mm.

J: The rectangle width along the Y axis, the value range is indicated as 0~9999.999mm.

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

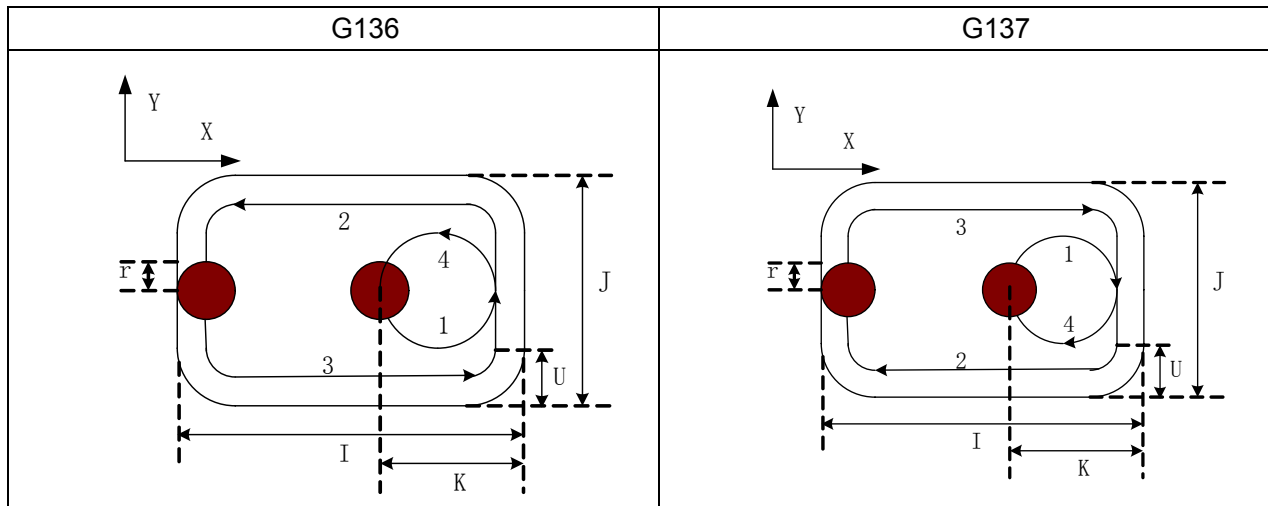
K: The distance between the finish-milling start point and the rectangle side in X axis direction, the value range is indicated as 0~9999.999mm.

U: Corner arc radius; no corner arc transition if it is omitted. When the U is omitted or it is equal to 0 and the tool radius is more than 0, the alarm is generated.

Cycle process:

- (1) Positioning to XY plane at the rapid traverse rate;
- (2) Down to point R level 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 circular and linear interpolation by the path of 2-3-4-5-6;
- (6) Perform circular interpolation by the path of transit arc 7 and return to the start point;
- (7) Returning to the initial level or point R level according to G98 or G99.

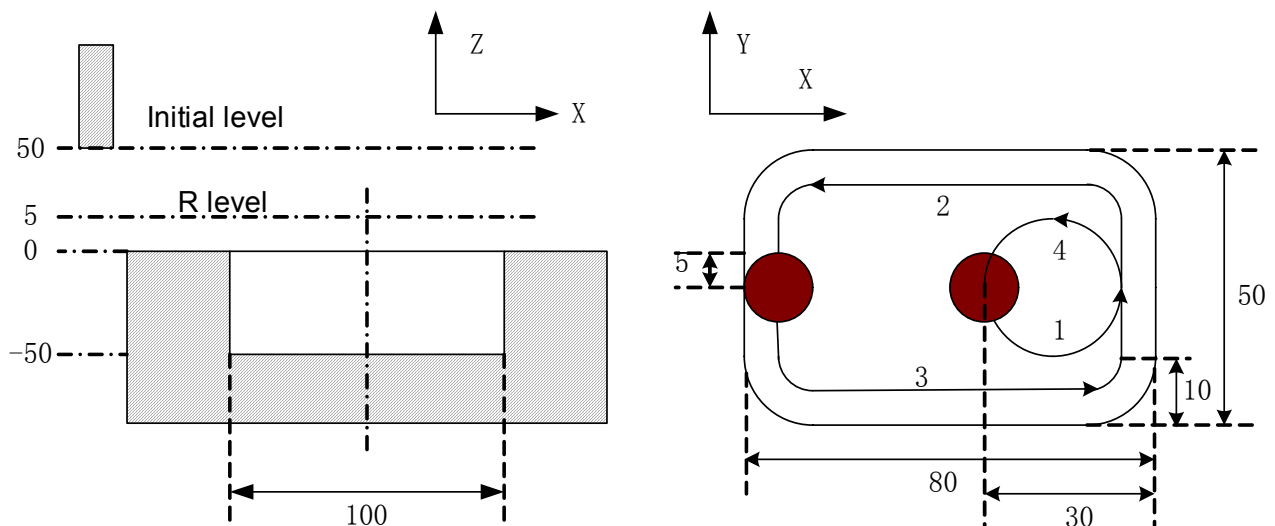
Command Path:



Related Explanation:

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

For example: To perform a fine-milling for the finished rough-milling rectangle groove with the canned cycle G136 command, see the following figure:



G90 G00 X50 Y50 Z50; (G00 rapid positioning)

G136 X25 Y25 R5 Z-50 I80 J50 K30 U10 F800 D1; (Perform finish-milling inside the rectangle groove at the bottom of a hole in the canned cycle D1=5)

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

M30;

3.15.2.18 Finish-milling cycle outside the rectangle G138/G139

Format:

G138
G98/G99 X_ Y_ R_ Z_ I_ J_ D_ K_ U_ F_
G139

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

Explanations:

G138: Finish-milling cycle outside the rectangle in CCW.

G139: Finish-milling cycle outside the rectangle in CW.

I: The width of rectangle along the X axis, the value range is indicated as 0~9999.999mm.

J: The width of the rectangle along the Y axis, the value range is indicated as 0~9999.999mm.

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

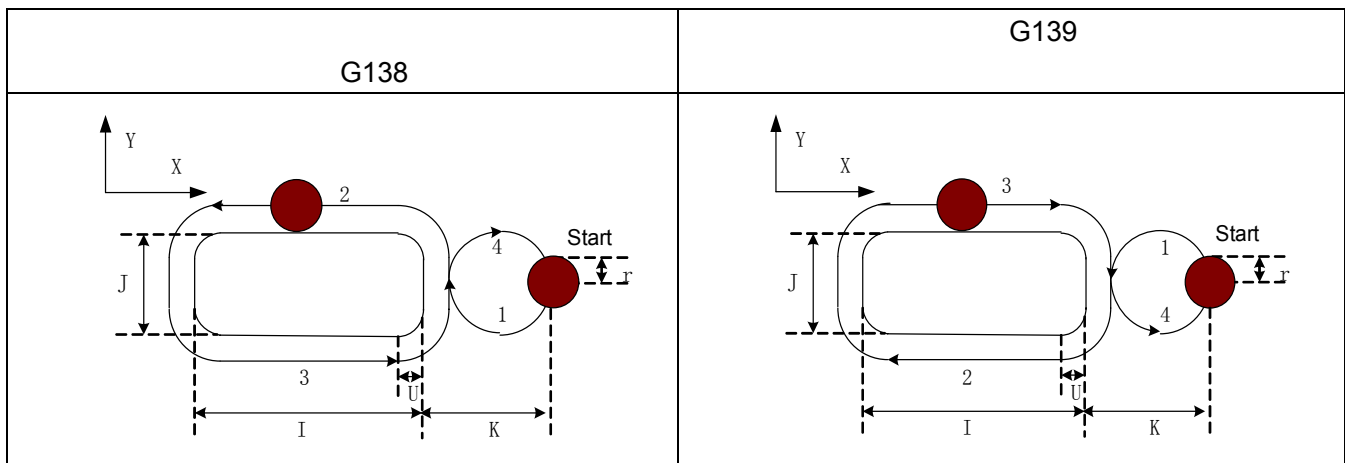
K: The distance between the finish-milling start point and the side of rectangle along the X axis, the value range is indicated as 0~9999.999mm.

U: Corner arc radius, if it is omitted, no corner arc transition.

Cycle process:

- (1) Positioning to the XY plane at the rapid traverse rate;
- (2) Down to the point R level 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 circular and linear interpolation by the path of 2-3-4-5-6;
- (6) Perform circular interpolation by the path of transit arc 7 and return to the start point;
- (7) Returning to the initial level or point R level according to G98 or G99.

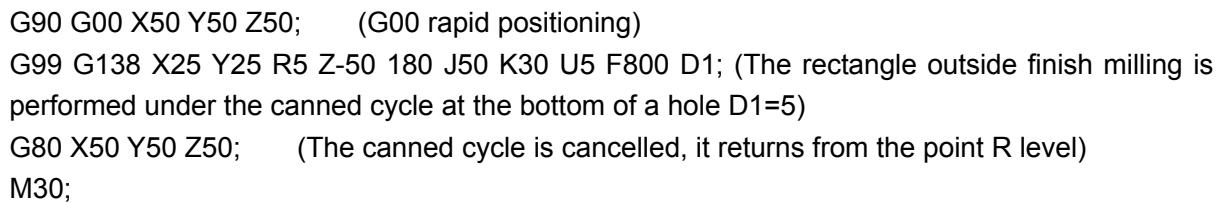
Command Path:



Related Explanations:

- (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 L are disabled in this cycle, but, the value of Q and P are reserved as canned cycle modal value.

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



- (1) The spindle should be rotated (The M code should be correctly specified, or, the alarm will be generated, the G74 by M04, G84 by M03) by using the miscellaneous function (M 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 G73~G89. If neither data is contained in the block, the hole machining is not performed (G110, G111, G112, G113, G114, G115, G134, G135, G136, G137, G138 and G139 are still needed to specify the corresponding address I, J and K, or the alarm occurs). But the hole machining is not performed when the G04 X_ is specified in the circumstance of X, because the X indicates for time when the G04 is specified.

(3) When the canned cycle (G74 or G84) is employed in spindle rotation controllation, if the hole position (X, Y) or distance from initial point level to the point R level 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 G04 is inserted into each hole machining, which is shown as follows:



G86 X_ Y_ Z_ R_ F_ ;

G04 P_ ; (For dwell time P, without hole machining)

X_ Y_ ; (The next hole is machined)

G04 P_ ; (For dwell time P, without hole machining)

X_ Y_ ; (The next hole is machined)

G04 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.

(4) 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- K-; (For canned cycle)

G□□ G# X- Y- Z- R- Q- P- F- K-; The X, Y and Z axes are moved by G#, the R, P, Q and K 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 M and MF 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.

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

(7) If the tool length offset commands (G43, G44 and G49) are specified in a canned cycle block. Then, the offset is performed when the point R level 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.

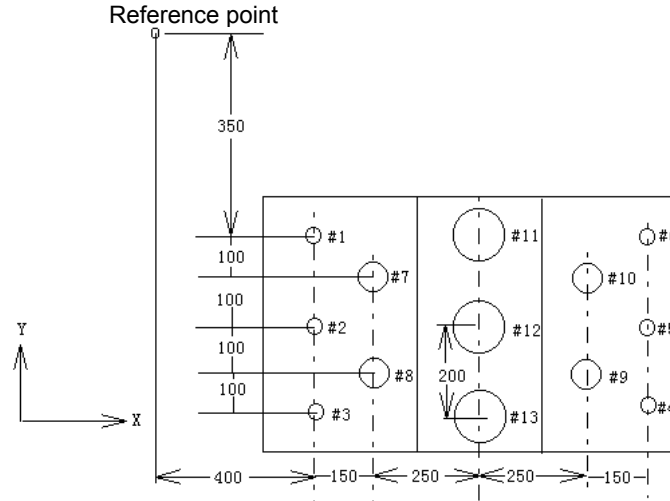
(9) When the bit 1 of parameter 3 (D_R) is set to 1, the D value in tool compensation page indicates diameter value.

3.15.4 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.

Note: Address I, J, K and U of canned cycle G110, G111, G112, G113, G114, G115, G134, G135, G136, G137, G138 and G139 are not saved as canned cycle modal data, so the I, J and K values need to be specified in each block, or the alarm will be generated.

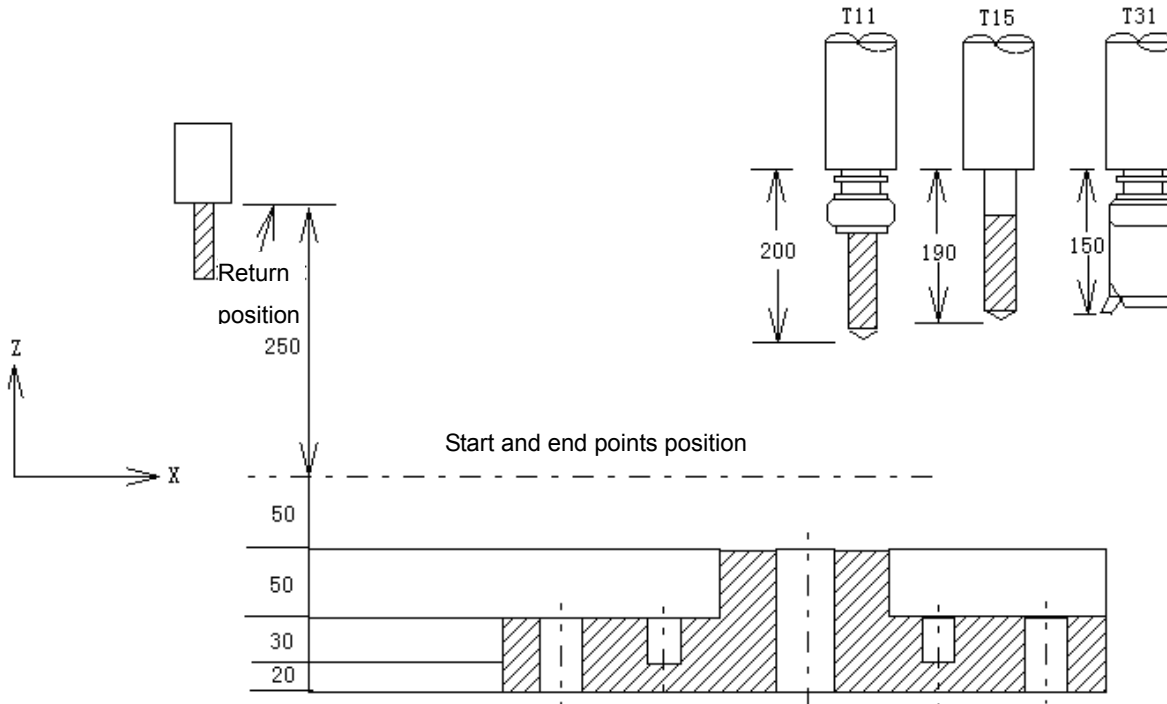
3.15.5 Examples for canned cycle and tool length compensation



The hole number from 1 to 6... drilling $\Phi 10$

The hole number from 7 to 10... drilling $\Phi 20$

The hole number from 11 to 13... boring $\Phi 95$ hole (depth is 50mm)



The values of offset numbers H11, H15 and H 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 H11 ;	Plane tool length compensation is performed at the initial level.
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 level returned.

N007 G98 Y-750.0 ;	#3 hole is machined after positioning, initial level returned.
N008 G99 X1200.0 ;	#4 hole is machined after positioning, point R level returned.
N009 Y-550.0 ;	#5 hole is machined after positioning, point R level returned.
N010 G98 Y-350.0 ;	#6 hole is machined after positioning, initial level returned
N011 G00 X0 Y0 M5 ;	Reference point return, the spindle stops.
N012 G49 Z250.0 ;	Tool length compensation cancellation
N013 G43 Z0 H15 ;	Initial level, 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 level returned.
N016 G98 Y-650.0 ;	#8 hole is machined after positioning, initial level returned.
N017 G99 X1050.0 ;	#9 hole is machined after positioning, point R level returned.
N018 G98 Y-450.0 ;	#10 hole is machined after positioning, initial level returned.
N019 G00 X0 Y0 M5 ;	Reference point return, the spindle stops.
N020 G49 Z250.0 ;	Tool length compensation cancellation.
N021 G43 Z0 H31 ;	Tool length compensation at initial level.
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 level returned.
N024 G91 Y-200.0 ; Y-200.0 ;	#12 and #13 are machined after positioning, point R level 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

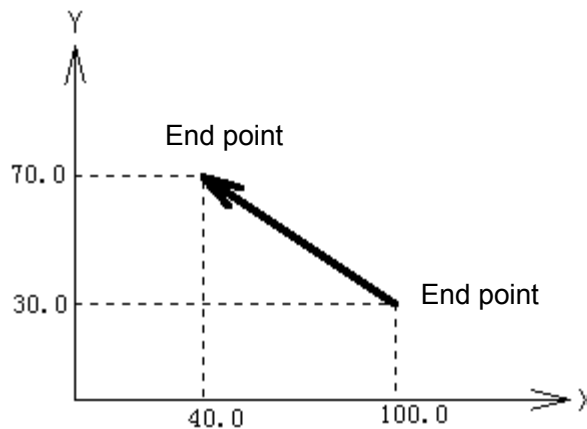
Format:

G90; Absolute command
G91; Incremental command

Function:

There are two kind 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

Format: G92 X_ Y_ Z_;

Function: 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.

Command 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 G92 command, current coordinate value will be not changed if the X, Y and Z are not input, the program zero is set by the current coordinate value. When the X, Y or Z is not input, the coordinate axis not input keeps on the original set value.

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

Format: G94 Fxxxx; (F0001~ F8000, the leading zero 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 Fxxxx; (F0.0001~F500, 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 Fxxxx is performed by system. The actual cutting feedrate varies with the spindle speed. The spindle cutting feed value per rev is specified by G95 Fxxxx, 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 paramter 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)

Notice: 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 transducer during machining.

3.19 G98、G99

Format:

G98;

G99;

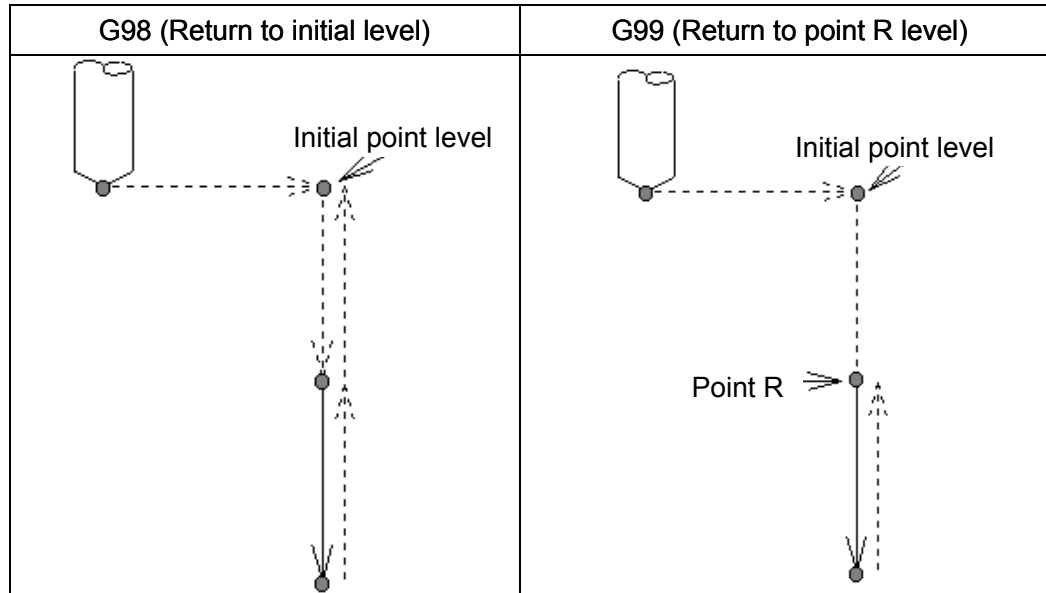
Function:

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

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

Explanation:

Modal G command



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 transfered from one figure to another. GSK980MD owns two chamfering functions, one is linear chamfering, and the other is arc chamfering.

3.20.1 Linear chamfering

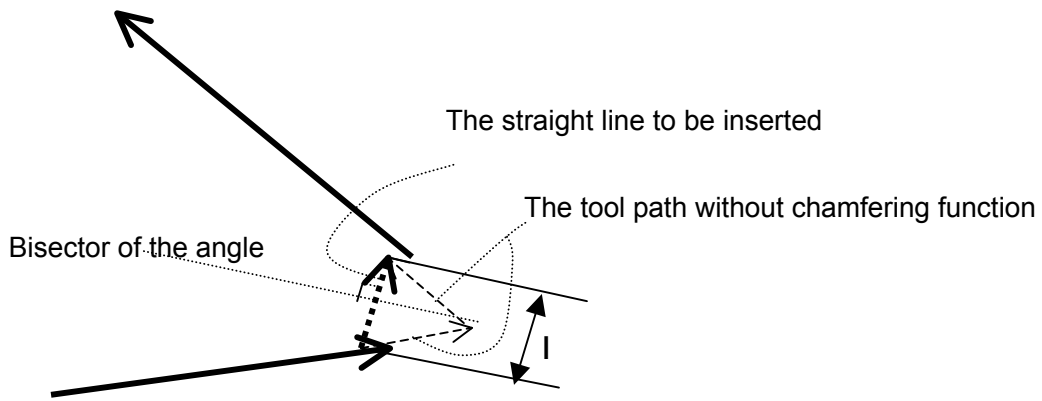
The linear chamfering is that a straight line is inserted between figures of the straight lines, the arcs, as well as the straight line and arc. The command address for linear chamfering is L. The data followed by command address L is the length of chamfering straight line. The linear chamfering should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

Linear to linear

Format: G01 IP_ L_; (IP is axis movement command)

G01 IP_;

Function: A straight line is inserted into interpolation between 2 straight lines.

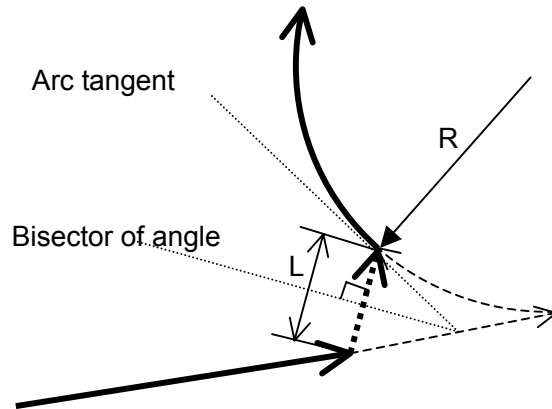


1.Linear to circular

Format:

```
G01 IP_ L_;  
G02/G03 IP_ R_(I_ J_ K_);
```

Function: A straight line is inserted between straight line and arc interpolation.

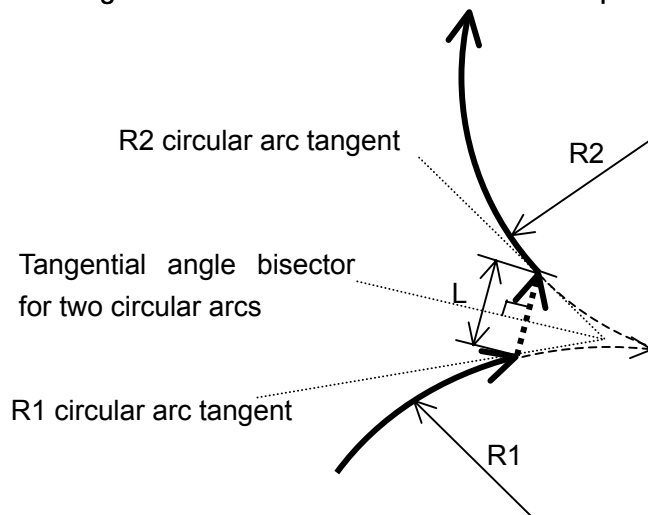


2.Circular to circular

Format:

```
G02/G03 IP_ R_(I_ J_ K_) L_;  
G02/G03 IP_ R_(I_ J_ K_);
```

Function: A straight line is inserted between two arc interpolations.

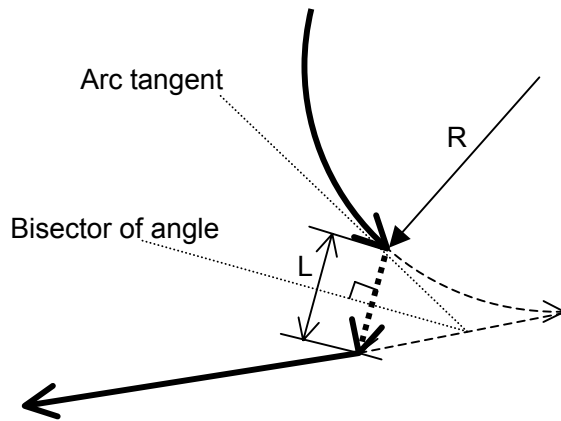


3. Circular to linear

Format:

```
G02/G03 IP_ R_(I_ J_ K_) L_;  
G01 IP_;
```

Function: A straight line is inserted between the arc and linear interpolation.



3.20.2 Circular chamfering

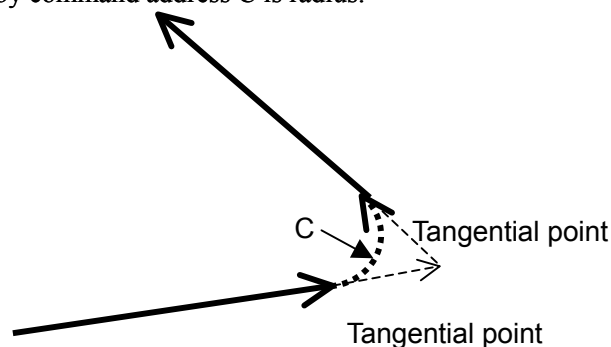
An arc is inserted between the two linear figures, arc figures or linear and arc figures, this is called circular chamfering. Tangent transition is performed between arc and figure line. The command address is C for the arc chamfering, the data followed by command address C is the radius of chamfering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

1. Linear to linear

Format:

```
G01 IP_ C_;  
G01 IP_;
```

Function: 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.

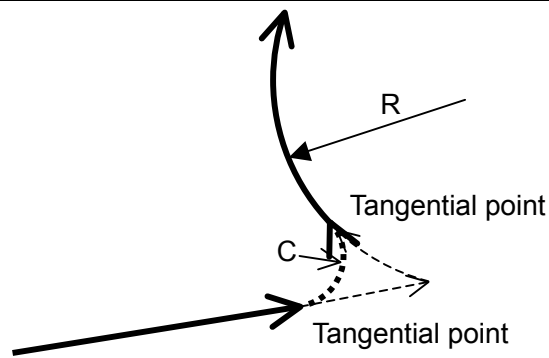


2. Linear to Circular

Format:

```
G01 IP_ C_;  
G02/G03 IP_ R_(I_ J_ K_) ;
```

Function: 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.

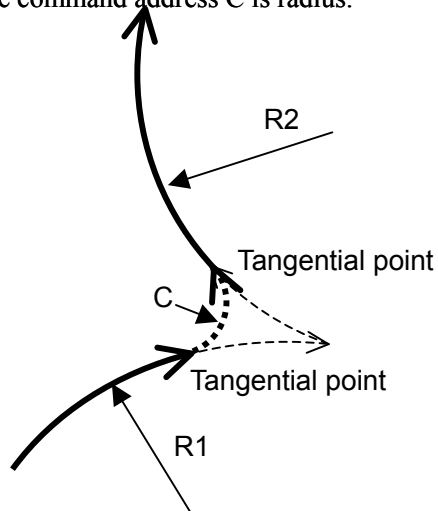


3. Circular to Circular

Format:

```
G02/G03 IP_ R_(I_ J_ K_) C_;
G02/G03 IP_ R_(I_ J_ K_);
```

Function: 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.

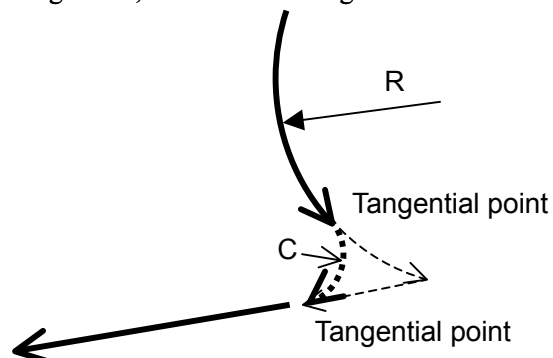


4. Circular to Linear

Format:

```
G02/G03 IP_ R_(I_ J_ K_) C_;
G01 IP_;
```

Function: 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.

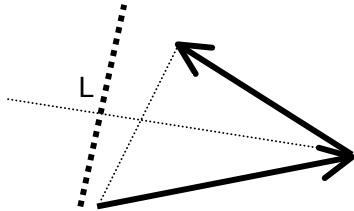


3.20.3 Special

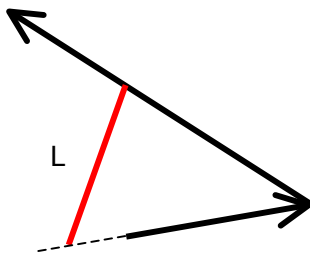
The chamfering function is ineffective or alarm is issued in the following circumstances:

1. Linear chamfering

- A. The chamfering function is ineffective when two interpolation lines is shown on the same line.
- B. If the chamfering linear length is too long, and the CNC alarm occurs.

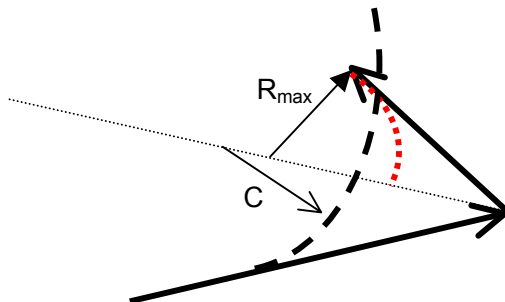


- C. If some line (arc) is too short, the alarm occurs.

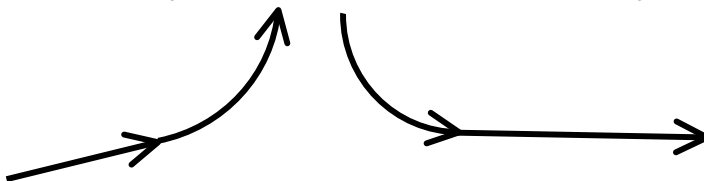


2. Arc Chamfering

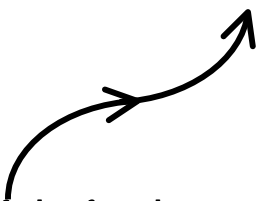
- A. The arc chamfering function is disabled when two interpolation lines is shown on the same line.
- B. If the chamfering radius is excessive, the CNC alarm occurs.



- C. 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: The chamfering function can be performed only in the plane specified by G17, G18 or G19, these functions can not be performed in parallel axes.

Note 2: Changing the coordinate system by G92 or G54 to G59, or, the block followed by performing the reference point return from G28 to G30 can not specify the chamfering.

Note 3: Chamfering function can not be employed in the DNC mode.

3.21 Macro Command

The macro command of similar advanced language is offered by this GSK980MD, the custom macro command can realize variable assignment, arithmetical operation, logical judgement and conditional transfer. The macro command is useful to program the processing for special parts, and reduce the fussy numerical value calculation when the manual programming is performed, and the custom program is simplified.

3.21.1 Macro Variable

(1) Usage for the variable

The address value in program can be commanded by variable. The variable can be assigned by program command or set by keyboard directly. Several variables can be employed in one program, which are distinguished by variable numbers.

- **Variable representation**

To indicate with “#” + variable number;

Format: # i (i=200, 202, 203.....) ;

Example: #205, #209, #225.

- **Citation of variables**

1. The value following the address is replaced by variable.

Format: <address>+”#i: or <address> + “- #I”, indicates that the variable value or the negative value of variable is regarded as address value.

Example: F#203... The command function is same with F15 when #203 is equal to 15;

Z- #210... The command function is same with Z-250 when #210 is equal to 250

G#230... The command function is same with G3 when #230 is equal to 3.

2. Variable replaces variable number.

Format: “#” + “9” + the variable number replaced.

Examples: X#9200 and X500 share same command function and X- #9200 and X-500 share same command function when #200 equals to 205 or #205 equals to 500.

Note 1: Address O and N can not be applied to the variable, so they can not be programmed by O#200 and N#220;

Note2: If the maximum command value specified by address is excessive, it is can not be employed; for example: M#230 is exceeded the max. command value when #230 equals to 130.

(2) Types of Variables

Variable can be divided into common and system variables according to the variable number, and their usage and characteristics are also different.

- **Common Variables**

The common variables are divided into #200~#300 and #500~#515, they are commonly used in a program. The variable and operation result defined in program 1 are also suited for program 2 and 3.

The common variables are #200~#231 and #500~#515, all variable values are reserved when the power is turned off.

● System Variables

The usage of system variable is fixed in system, system variable interface input signal is from #1000 to #1015, and the interface output signal is from #1100 to #1105;

The interface input/output signal of system variable and the other function interface signal share same interface, which is set for effective by the parameter. The interface input signal of system variable is effective only when the corresponding interface signal function is disabled.

The interface input signal status is got for performing operations such as the skip judging after the system reads interface input signal value #1000~1015,.

Interface signal of system variables #1000~#1015 are defined by PLC.

3.21.2 Operation and transfer command G65

General format:

G65 Hm P#i Q#j R#k;

Where: m: Indicates an operation command function or a transfer command function.

i: Indicates the variable name with operation result.

j: Indicates the variable name 1 for operation , or a constant.

k: Indicates the variable name 2 for operation , or a constant.

Command meaning: # i = # j O # k

For example:

P#200 Q#201 R#202.....#200 = #201 O #202, Operation sign, decided by Hm

P#200 Q#201 R15.....#200 = #201 O 15;

P#200 Q-100 R#202.....#200 = -100 O #202;

Explanations:

- The variable value does not contain the decimal point, the unit is 0.001mm; for example, X#100 equals to X0.03mm when #100 is 30;
- The variable has no “#” when it is indicated by constant directly;

Macro list

Format	Function	Definition
G65 H01 P#i Q#j	Assignment	# i = # j; Assign the variable value j to i
G65 H02 P#i Q#j R#k;	Decimal addition operation	# i = # j + # k
G65 H03 P#i Q#j R#k;	Decimal subtraction operation	# i = # j - # k
G65 H04 P#i Q#j R#k;	Decimal multiplication operation	# i = # j × # k
G65 H05 P#i Q#j R#k;	Decimal division operation	# i = # j ÷ # k
G65 H11 P#i Q#j R#k;	Binary addition (or operation)	# i = # j OR # k
G65 H12 P#i Q#j R#k;	Binary multiplication (and operation)	# i = # j AND # k
G65 H13 P#i Q#j R#k;	Binary XOR	# i = # j XOR # k
G65 H21 P#i Q#j;	Decimal square rooting	# i = $\sqrt{\# j}$
G65 H22 P#i Q#j;	Absolute value for decimal	# i = # j
G65 H23 P#i Q#j R#k;	Complement for decimal	Complement for # i = (#j ÷ # k)
G65 H24 P#i Q#j;	Decimal to binary	# i = BIN(# j)
G65 H25 P#i Q#j;	Binary to decimal	# i = DEC(# j)

G65 H26 P#i Q#j R#k;	Decimal multiplication and division operation	# i = # i×# j÷# k
G65 H27 P#i Q#j R#k;	Complex square root	$\# i = \sqrt{\# j^2 + \# k^2}$
G65 H31 P#i Q#j R#k;	Sine	# i = # j×sin(# k)
G65 H32 P#i Q#j R#k;	Cosine	# i = # j×cos(# k)
G65 H33 P#i Q#j R#k;	Tangent	# i = # j×tan(# k)
G65 H34 P#i Q#j R#k;	Arc tangent	# i = ATAN(# j / # k)
G65 H80 Pn;	Unconditional transfer	Skip to the block n
G65 H81 Pn Q#j R#k;	Conditional transfer transfer 1	If # j = # k, skip to block n, or, sequentially executes
G65 H82 Pn Q#j R#k;	Conditional transfer 2	If # j = # k, skip to the block n, or, sequentially executes
G65 H83 Pn Q#j R#k;	Conditional transfer 3	If # j = # k, skip to the block n, or, sequence performs
G65 H84 Pn Q#j R#k;	Conditional transfer 4	If # j = # k, skip to the block n, or, sequentially executes
G65 H85 Pn Q#j R#k;	Conditional transfer 5	If # j = # k, skip to the block n, or, sequentially executes
G65 H86 Pn Q#j R#k;	Conditional transfer 6	If # j = # k, skip to the block n, or, sequentially executes
G65 H99 Pn;	User alarm occurs	(500+n) user alarm occurs.

1 Operation Command

- 1) Variable assignment: # I = # J

G65 H01 P#I Q#J

(e.g.) G65 H01 P# 201 Q1005; (#201 = 1005)

G65 H01 P#201 Q#210; (#201 = #210)

G65 H01 P#201 Q-#202; (#201 = -#202)

- 2) Decimal addition operation: # I = # J+# K

G65 H02 P#I Q#J R#K

(e.g.) G65 H02 P#201 Q#202 R15; (#201 = #202+15)

- 3) Decimal subtraction operation: # I = # J-# K

G65 H03 P#I Q#J R# K

(e.g.) G65 H03 P#201 Q#202 R#203; (#201 = #202-#203)

- 4) Decimal multiplication operation: # I = # J×# K

G65 H04 P#I Q#J R#K

(e.g.) G65 H04 P#201 Q#202 R#203; (#201 = #202×#203)

- 5) Decimal division operation: # I = # J÷# K

G65 H05 P#I Q#J R#K

(e.g.) G65 H05 P#201 Q#202 R#203; (#201 = #202÷#203)

- 6) Binary logic addition (OR): # I = # J.OR. # K

G65 H11 P#I Q#J R#K

(e.g.) G65 H11 P#201 Q#202 R#203; (#201 = #202.OR. #203)

7) Binary logic multiplication (AND): # I = # J.AND. # K

G65 H12 P#I Q#J R#K

(e.g.) G65 H12 P# 201 Q#202 R#203; (#201 = #202.AND.#203)

8) Binary XOR: # I = # J.XOR. # K

G65 H13 P#I Q#J R#K

(e.g.) G65 H13 P#201 Q#202 R#203; (#201 = #202.XOR. #203)

9) Decimal square rooting: # I = $\sqrt{\#J}$

G65 H21 P#I Q#J

(e.g.) G65 H21 P#201 Q#202 ; (#201 = $\sqrt{\#202}$)

10) Absolute value for the decimal: # I = | # J |

G65 H22 P#I Q#J

(e.g.) G65 H22 P#201 Q#202 ; (#201 = | #202 |)

11) Complement for decimal: # I = # J—TRUNC(#J/#K)×# K, TRUNC: round off decimal part

G65 H23 P#I Q#J R#K

(e.g.) G65 H23 P#201 Q#202 R#203; (#201 = #202- TRUNC (#202/#203)×#203

12) Decimal to binary: # I = BIN (# J)

G65 H24 P#I Q#J

(e.g.) G65 H24 P#201 Q#202 ; (#201 = BIN (#202))

13) Binary to decimal: # I = BCD (# J)

G65 H25 P#I Q#J

(e.g.) G65 H25 P#201 Q#202 ; (#201 = BCD (#202))

14) Multiplication and division operation for decimal: # I = (# I×# J) ÷# K

G65 H26 P#I Q#J R# k

(e.g.) G65 H26 P#201 Q#202 R#203; (#201 = (# 201×# 202) ÷# 203)

15) Complex square root: # I = $\sqrt{\#J^2 + \#K^2}$

G65 H27 P#I Q#J R#K

(e.g.) G65 H27 P#201 Q#202 R#203; (#201 = $\sqrt{\#202^2 + \#203^2}$)

16) Sine: # I = # J•SIN (# K) (unit: %degree)

G65 H31 P#I Q#J R#K

(e.g.) G65 H31 P#201 Q#202 R#203; (#201 = #202•SIN (#203))

17) Cosine: # I = # J•COS (# K) (unit: %degree)

G65 H32 P#I Q#J R# k

(e.g.) G65 H32 P#201 Q#202 R#203; (#201 =#202•COS (#203))

18) Tangent: # I = # J•TAM (# K) (unit: % degree)

G65 H33 P#I Q#J R# K

(例 e.g.) G65 H33 P#201 Q#202 R#203; (#201 = #202•TAM (#203))

19) Cotangent: # I = ATAN (# J /# K) (unit: % degree)

G65 H34 P#I Q#J R# k

(e.g.) G65 H34 P#201 Q#202 R#203; (#201 =ATAN (#202/#203))

Note 1: The (P) ~ (S) units are specified by degree, the unit is 1‰ degree;

Note 2: The variable value is taken as integer during the operations; the decimal point will be rounded off when it is shown in the operation, and the variable unit is μm ;

Note 3: The variable value is in the range of $-2^{32} \rightarrow +2^{32}-1$, it is indicated from -9999999 to 9999999, it shows ***** when it exceeds the range.

2 Transfer Command

- 1) Unconditional transfer

G65 H80 Pn; n: Sequence number

(For example) G65 H80 P120; (Transfer to N120 block)

- 2) Conditional transfer 1 #J.EQ.# K (=)

G65 H81 Pn Q#J R# K; n: Sequence number

(e.g) G65 H81 P1000 Q#201 R#202;

Transfer to N1000 block when # 201 = #202, the program is performed sequentially when #201 \neq #202.

- 3) Conditional transfer 2 #J.NE.# K (\neq)

G65 H82 Pn Q#J R# K; n: Sequence number

(e.g) G65 H82 P1000 Q#201 R#202;

Transfer to N1000 block when # 201 \neq #202, the program is performed sequentially when #201 = #202

- 4) Conditional branch 3 #J.GT.# K (>)

G65 H83 Pn Q#J R# K; n: Sequence number

(e.g) G65 H83 P1000 Q#201 R#202;

Turning to N1000 block when 201 > #202, the program is performed sequentially when #201 \leq #202

- 5) Conditional transfer 4 #J.LT.# K (<)

G65 H84 Pn Q#J R# K; n: Sequence number

(e.g) G65 H84 P1000 Q#201 R#202;

Transfer to N1000 block when # 201 < #202, the program is performed sequentially when #201 \geq #202

- 6) Conditional transfer 5 #J.GE.# K (\geq)

G65 H85 Pn Q#J R# K; n: Sequence number

(e.g) G65 H85 P1000 Q#201 R#202;

Transfer to N1000 block when # 201 \leq #202, the program is performed sequentially when #201 < #202

- 7) Conditional transfer 6 #J.LE.# K (\leq)

G65 H86 Pn Q#J R# K; n: Sequence number

(e.g) G65 H86 P1000 Q#201 R#202;

Transfer to N1000 block when # 201 \leq #202, the program is performed sequentially when #201 > #202

8) P/S alarm is generated.

G65 H99 Pi; i: Alarm number+500

(e.g) G65 H99 P15;

P/S alarm 515 is generated.

Note: The sequence number can be specified by using the variable. For example: G65 H81 P#200 Q#201 R#202; it transfers to the block with sequence number specified by #200 when the condition is met.

CHAPTER 4 CUTTER RADIUS COMPENSATION

4.1 Application for Cutter Radius Compensation

4.1.1 Brief

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 overcut 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 overcutting will be generated at the intersection point of two blocks.

In order to settle the above issues and eliminate the error, the C type tool compensation should be setup. When a block is read in, the C type tool compensation 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 C type tool compensation performs more accurate compensation in figure because two blocks are read for processing in advance. See the Fig. 4-1.

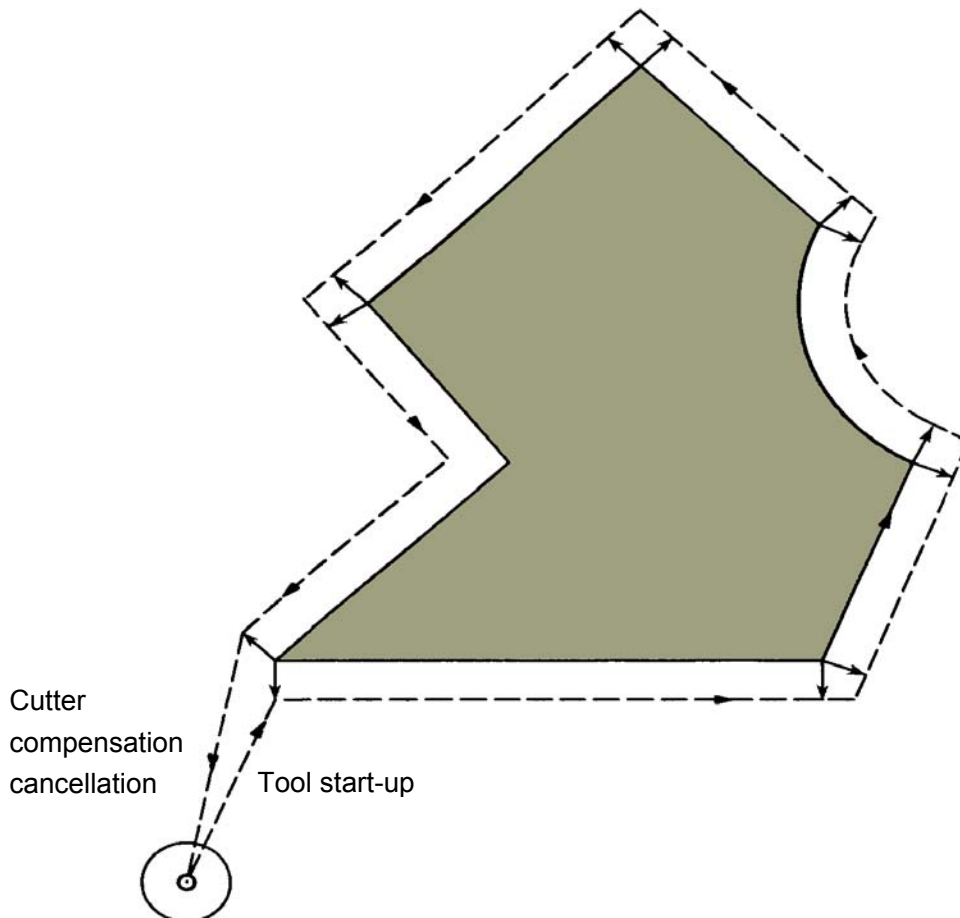


Fig.4-1 C type cutter radius compensation

4.1.2 Compensation value setting

The radius value of each tool should be set before C type tool compensation is applied. Tool radius compensation value is set in the OFFSET page (table 4-1), this page contains tool geometric radius and tool radius wear. Thereinto, 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 4-1 Display page for CNC cutter radius compensation value

No.	Geometric (H)	Wearing (H)	Geometric (D)	Wearing (D)
001	20.020	0.030	5.000	0.020
002	10.020	0.123	0.500	0.030
...
032	10.050	0.038	2.300	0.160

4.1.3 Command format

$$\left\{ \begin{matrix} G17 \\ G18 \\ G19 \end{matrix} \right\} \left\{ \begin{matrix} G40 \\ G41 \\ G42 \end{matrix} \right\} \left\{ \begin{matrix} G00 \\ G01 \end{matrix} \right\} X_ Y_ Z_ D_ ;$$

Commands	Explanation	Remarks
G17	Offset plane selection command (XY plane)	See the Fig.4-2
G18	Offset plane selection command (XZ plane)	
G19	Offset plane selection command (YZ plane)	
G40	Cutter radius compensation cancellation	
G41	Cutter radius compensation left along advancing direction	
G42	Cutter radius compensation right along advancing direction	

4.1.4 Compensation direction

Tool compensation direction is determined according to the relative position of tool with workpiece, when the cutter radius compensation is applied. See the Fig.4-2.

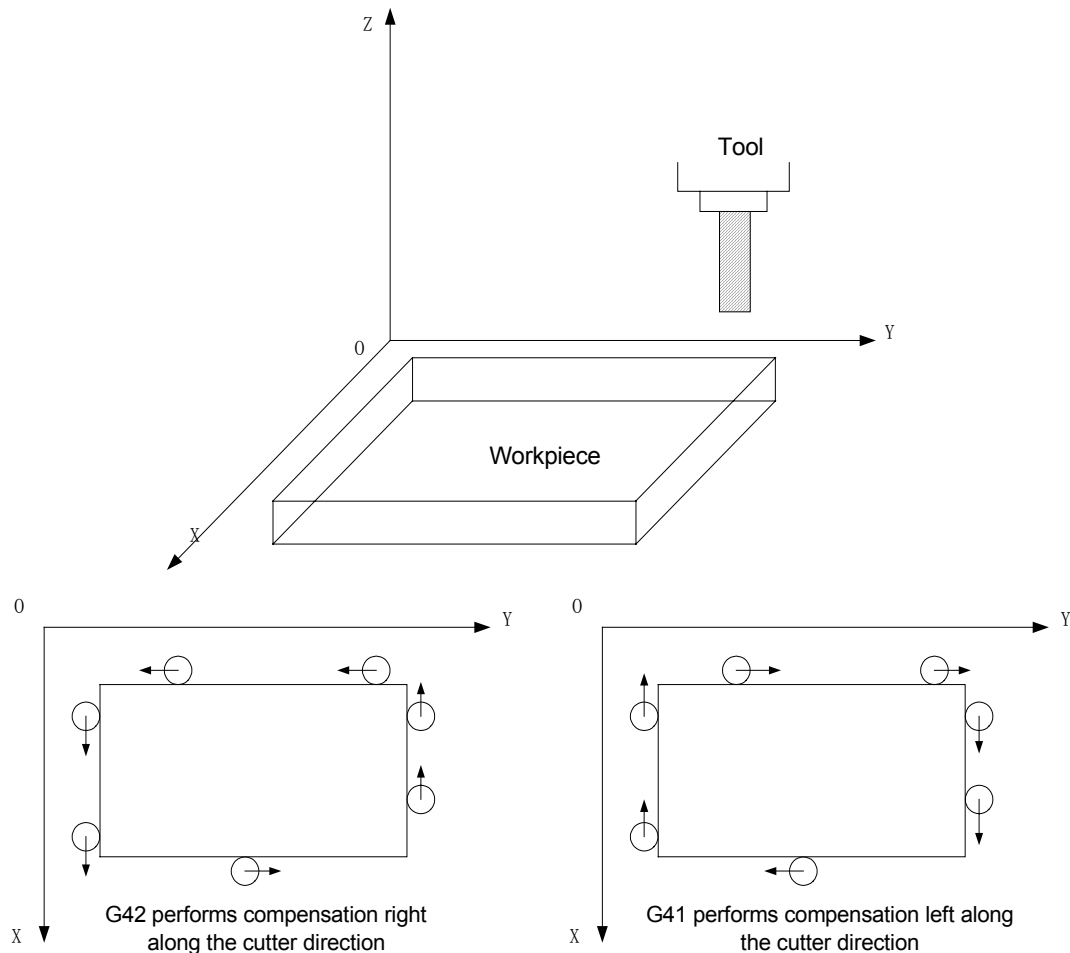


Table 4-2 Compensation direction (G17 plane)

4.1.5 Caution

- In initial status CNC is in cutter radius compensation cancellation mode. CNC sets cutter radius compensation offset mode when the G41 or G42 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 C type tool compensation can be performed in the MDI mode.
- The cutter radius compensation value can not be a negative, normally, the wearing value is negative (negative value indicates for wearing)
- Instead of G02 or G03, the setting or cancellation of cutter radius compensation can be commanded only by using G00 or G01, or the alarm occurs.
- CNC will cancel C type tool compensation mode when you press RESET key.
- Corresponding offset should be specified while the G40, G41 or G42 is specified in the block, or the alarm occurs.
- When cutter radius compensation is employed in main-program and sub-program, the CNC should cancel compensation mode before calling or exiting sub-program (namely, before M98 or M99 is performed), or the alarm occurs.
- Cancel the compensation mode temporarily when G54-59, G28-31 and canned cycle command are executed. Restore the cutter radius compensation mode when the above commands are finished.

4.1.6 Example for application

The parts are machined in the coordinate system in Fig. 4-3. The tool compensation number D07 is employed, tool geometric radius is 2mm and the tool radius wearing is 0.

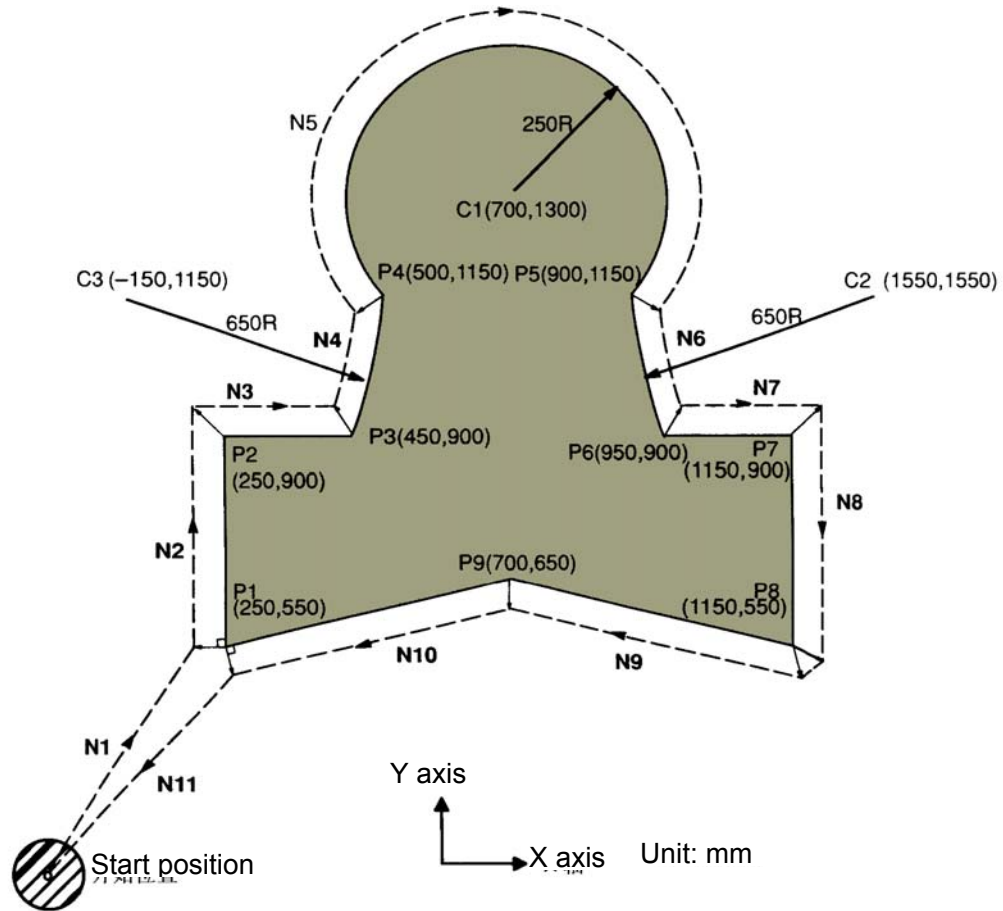


Fig. 4-3

Perform tool setting in the mode of offset cancellation, after finishing the tool setting, and set the tool radius D in OFFSET page.

Table 4-2

No.	Geometric (H)	Wearing (H)	Geometric (D)	Wearing (D)
01
...
07	2.000	0.000
08
...
32

Programs:

N0 G92 X0 Y0 Z0; Tool are positioned at start position X0, Y0 and Z0 when the absolute coordinate system is specified

N1 G90 G17 G00 G41 D07 X250.0 Y550.0; Start-up cutter, the tool is shifted to the tool path by the distance specified in D07, geometric radius of D07 is set to 2.0mm, tool wearing 0, then the tool radius is 2mm.

N2 G01 Y900.0 F150; Specifies machining from P1 to P2

N3 X450.0; Specifies machining from P2 to P3

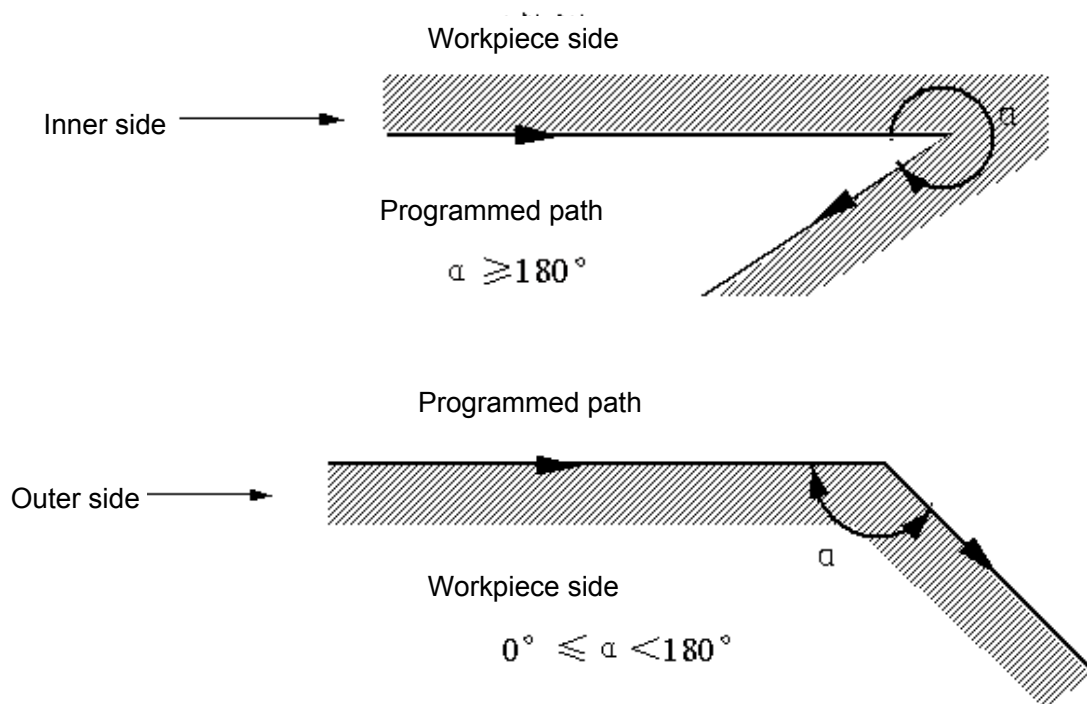
N4 G03 X500.0 Y1150.0 R650.0; Specifies machining from P3 to P4

N5 G02 X900.0 R-250.0;	Specifies machining from P4 to P5
N6 G03 X950.0 Y900.0 R650.0;	Specifies machining from P5 to P6
N7 G01 X1150.0;	Specifies machining from P6 to P7
N8 Y550.0;	Specifies machining from P7 to P8
N9 X700.0 Y650.0;	Specifies machining from P8 to P9
N10 X250.0 Y550.0;	Specifies machining from P9 to P1
N11 G00 G40 X0 Y0;	Cancels the offset mode, the tool is returned to the start position (X0, Y0)

4.2 Offset Path Explanation for Cutter Radius Compensation

4.2.1 Conception for inner side or outer side

“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 equal to 180° , it is referred to as “inner side”. When the angle is between 0° and 180° , it is referred to as “outer side”.



4.2.2 Tool movement in start-up

There are 3 steps should be performed for cutter radius compensation: establishment, performing and cancellation.

The tool movement performed from offset cancellation mode to G41 or G42 command establishment is called tool compensation establishment (also called start-up)

Note: For S, L and C labelled in the following figures, if not especially described, they should be regarded as the following meaning:

- S----Single block stop point;
- L----Linear;
- C----Circular arc.

(a) Tool movement along an inner side of a corner ($\alpha \geq 180^\circ$)

1) Linear to linear

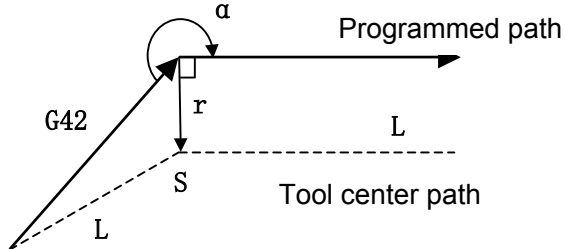


Fig.4-4a Linear to linear (start-up from inner side)

2) Linear to circular

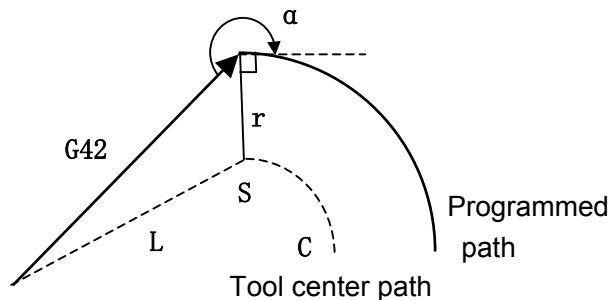


Fig.4-4b Linear to circular (start-up from inner side)

(b) Tool movement along the outside of a corner at an obtuse angle ($180^\circ > \alpha \geq 90^\circ$)

1) Linear to linear

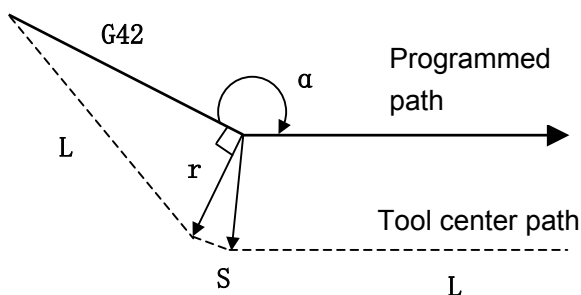


Fig.4-5a Linear to linear (start-up outside)

2) Linear to circular

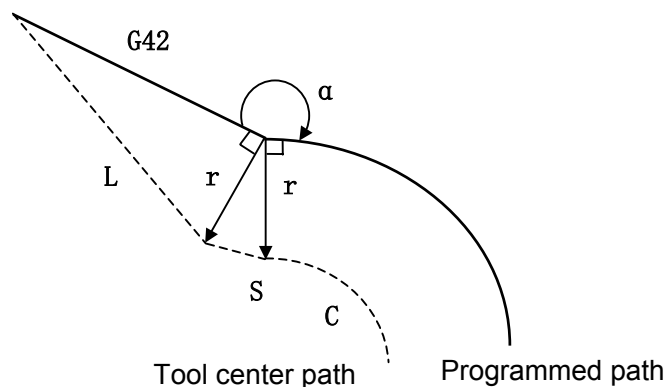


Fig.4-5b Linear to circular (Start-up outside)

(c) Tool movement along the outside of a corner at an acute angle ($\alpha < 90^\circ$)

1) Linear to Linear

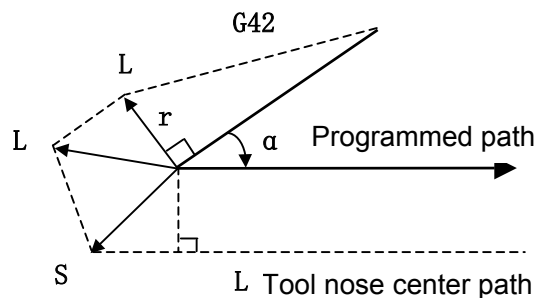


Fig.4-6a Linear to linear (start-up from outer side)

2) Linear to circular

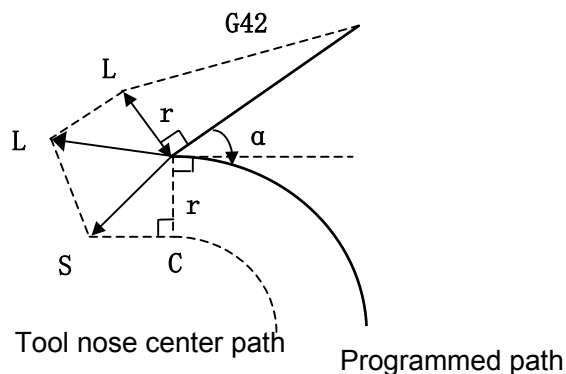


Fig.4-6b Linear to circular (start-up from outer side)

(d) Tool movement along the outside linear to linear at an acute angle less than 1 degree ($\alpha \leq 1^\circ$)

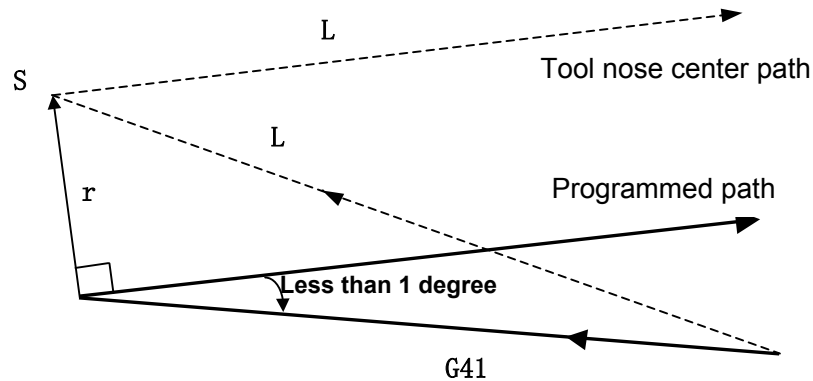


Fig.4-7 Linear to linear (the corner is less than 1 degree, start-up from outer side)

4.2.3 Tool movement in offset mode

The mode after setting the cutter radius compensation and before cancelling the cutter radius compensation is called offset mode.

● Offset path of invariable compensation direction in compensation mode

(a) Tool movement along the inside of a corner ($\alpha \geq 180^\circ$)

1) Linear to linear

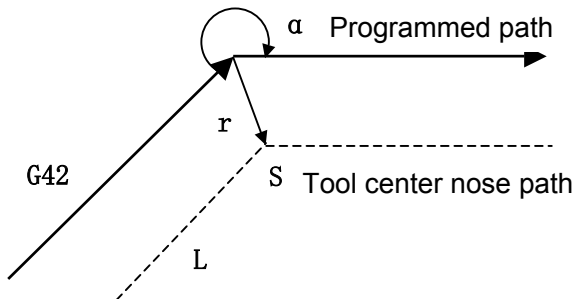


Fig.4-8a Linear to linear (inside movement)

2) Linear to circular

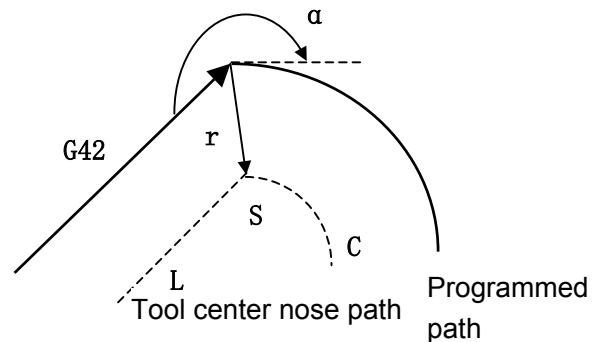


Fig.4-8b Linear to circular (inside movement)

3) Circular to linear

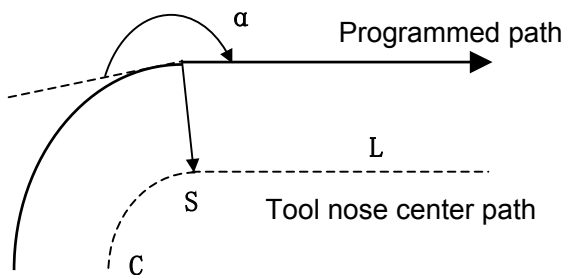


Fig.4-8c Circular to linear (inside movement)

4) Circular to circular

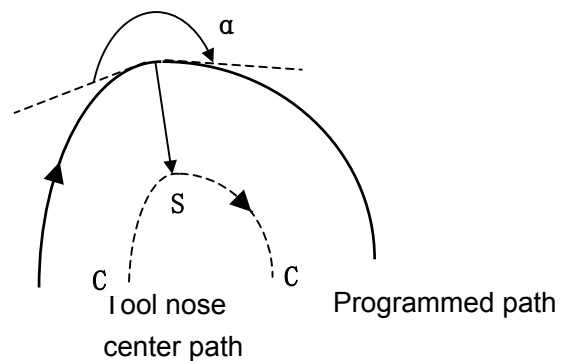


Fig.4-8d Circular to circular (inside movement)

5) Inner side machining less than 1 degree and compensation vector amplification

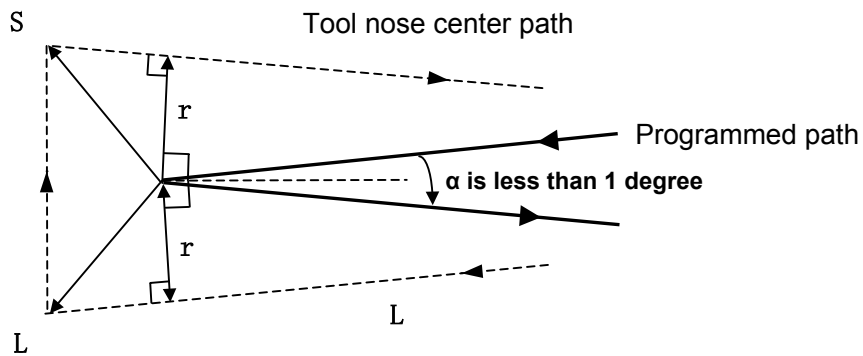


Fig.4-8e Linear to linear (corner is less than 1 degree, inside movement)

(b) Tool movement along the outside of a corner at an obtuse angle ($180^\circ > \alpha \geq 90^\circ$)

1) Linear to linear

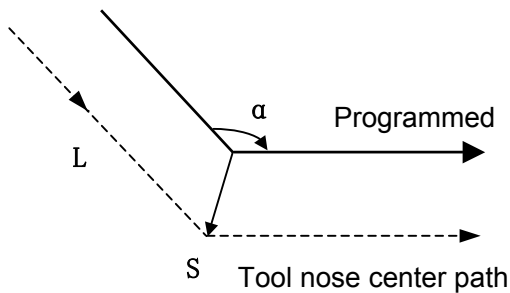


Fig.4-9a Linear to linear (obtuse angle, outside movement)

2) Linear to circular

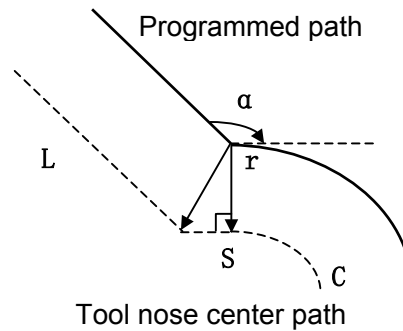


Fig.4-9b Linear to circular (obtuse, outside movement)

3) Linear to linear

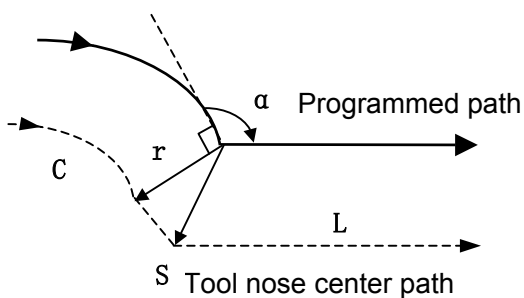


Fig.4-9c Circular to linear (obtuse angle, outside movement)

4) Circular to circular

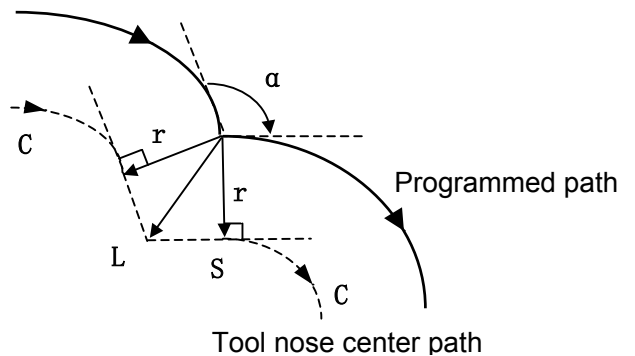


Fig.4-9d Circular to circular (obtuse angle, outside movement)

(c) Tool movement along the outside of a corner at an acute angle ($\alpha < 90^\circ$)

1) Linear to linear

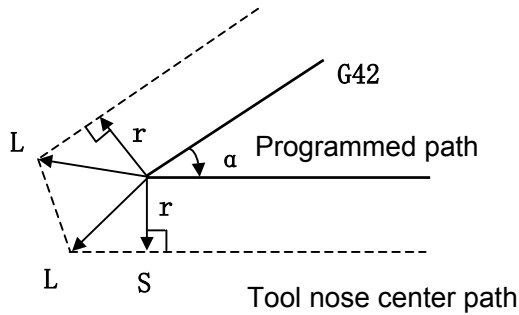


Fig.4-10a Linear to linear (acute, movement outside)

2) Linear to circular

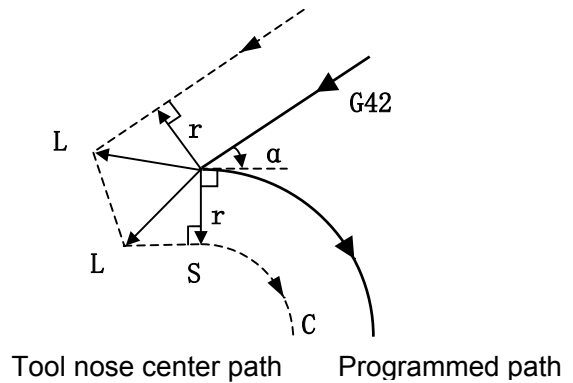


Fig.4-10b Linear to circular (acute, movement outside)

3) Circular to linear

Linear to linear

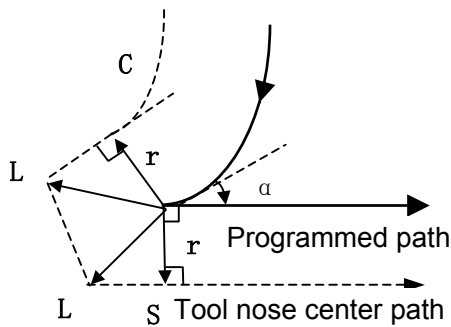


Fig.4-10c Circular to linear (acute, movement outside)

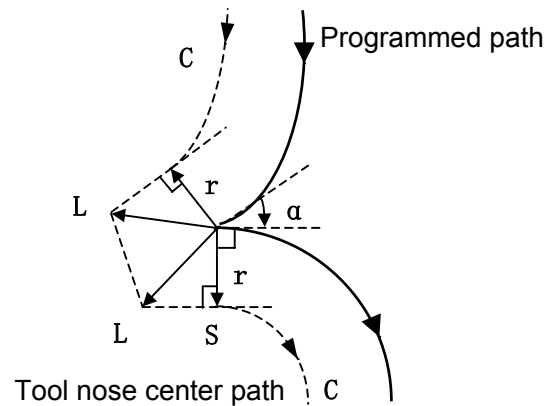
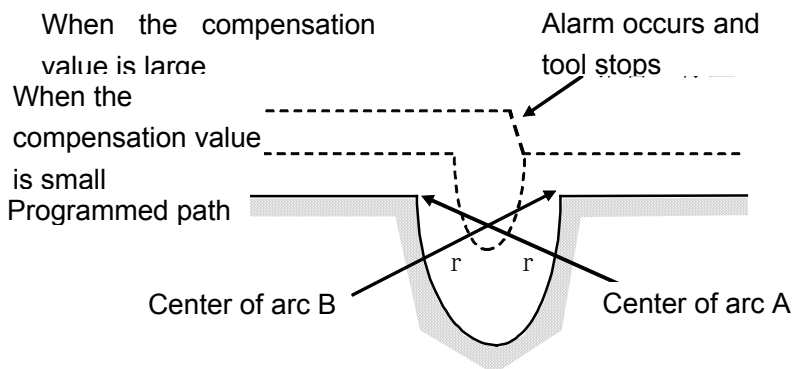


Fig.4-10d Circular to circular (acute, movement outside)

(d) When it is exceptional

1) There is no intersection



When the tool radius value is small, there is an intersection for the arc compensation, when the radius is bigger, the intersection may not exist, the tool stops at the end of previous block, and then the alarm occurs.

Fig.4-11 Exceptional -----There is no intersection after the path offset

- 2) The arc center is consistent to the start point or end point

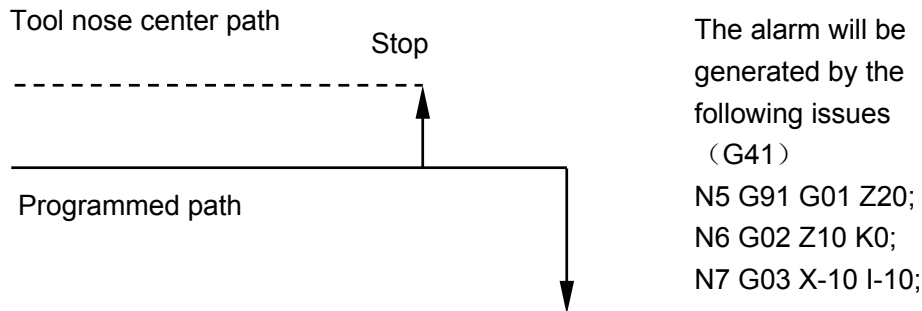


Fig.4-12 Center of arc is consistent to the start point or end point

● Offset path with the compensation direction changed in compensation mode

The compensation direction can be changed in special occasion, but it cannot be changed at the beginning and the following block. There are no inner side and outer side for the full compensation.

- 1) Linear to linear

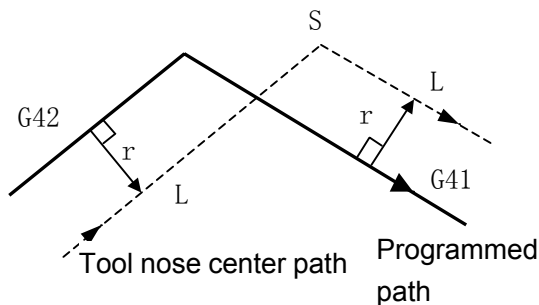


Fig.4-13a Linear to linear (compensation direction changed)

- 2) Linear to circular

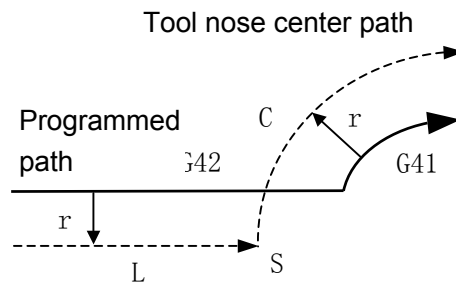


Fig.4-13b Linear to circular (compensation direction changed)

- 3) Circular to linear

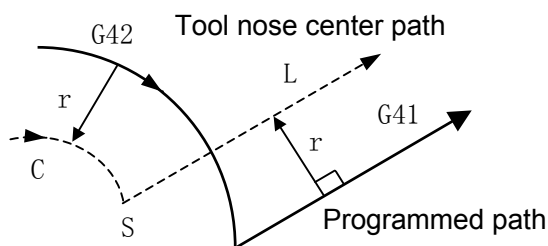


Fig.4-13c Circular to linear (compensation direction changed)

- 4) Circular to circular

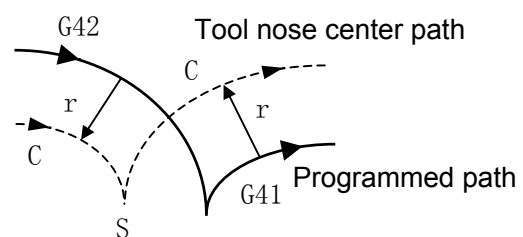


Fig.4-13d Circular to circular (compensation direction changed)

- 5) When there is no intersection if the compensation is normally performed.

When changing the offset direction from block A to block B using G41 and G42, if the intersection of the offset path is not required, create the vector vertical to block B at the start point of block B.

i) Linear to linear

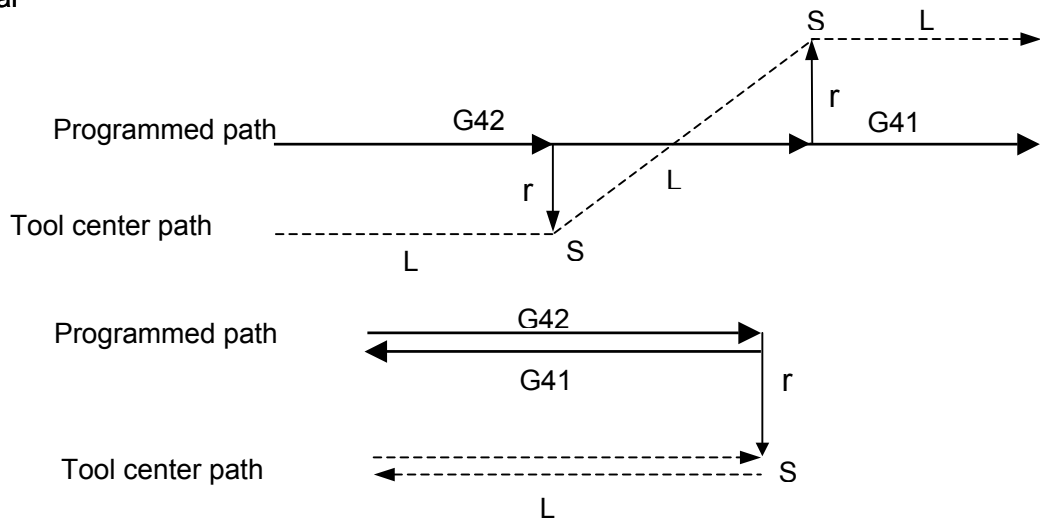


Fig. 4-14a Linear to linear, there is no intersection
(Compensation direction changed)

ii) Linear to circular

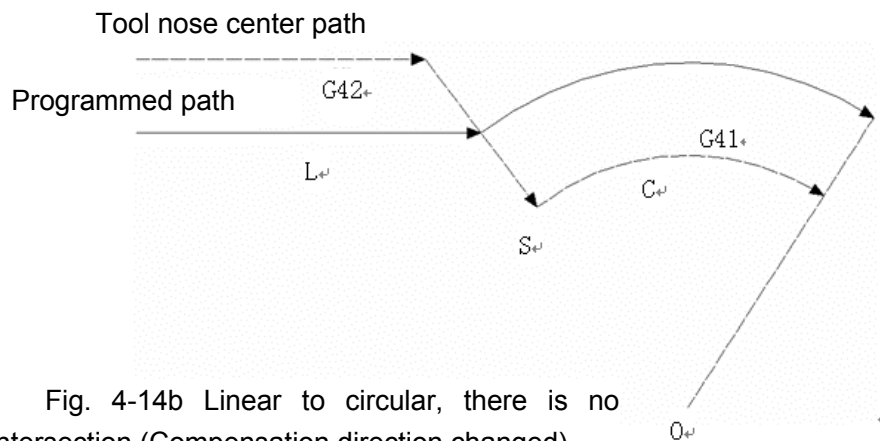


Fig. 4-14b Linear to circular, there is no intersection
(Compensation direction changed)

iii) Circular to circular

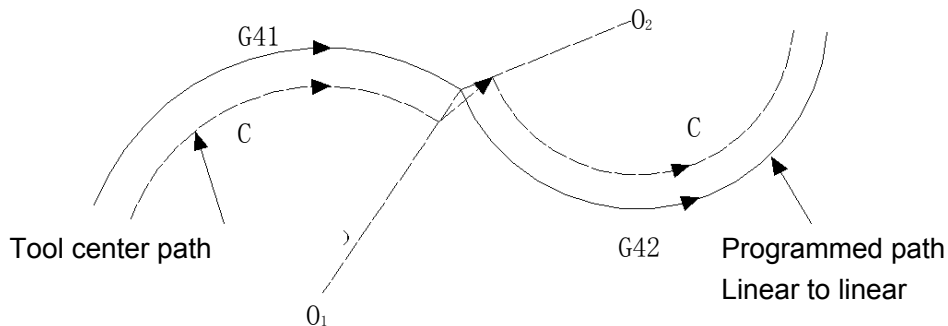


Fig. 4-14c Circular to circular, there is no intersection
(Compensation direction changed)

4.2.4 Tool operation in offset cancellation mode

When the G40 command is employed in block in compensation mode, the CNC enters the compensation cancellation mode. This is called compensation cancellation.

The circular arc command (G02 and G03) can not be employed when the C type cutter radius compensation is cancelled. If they are commanded, alarm is generated and the operation is stopped

It controls and performs this block and the blocks in the cutter radius compensation buffer memory in the compensation cancellation mode. If the single block switch is turned on, it stops after executing a block. The next block is executed instead of reading it when the start key is pressed again.

(a) Tool movement along an inner side of a corner ($\alpha \geq 180^\circ$)

1) Linear to linear

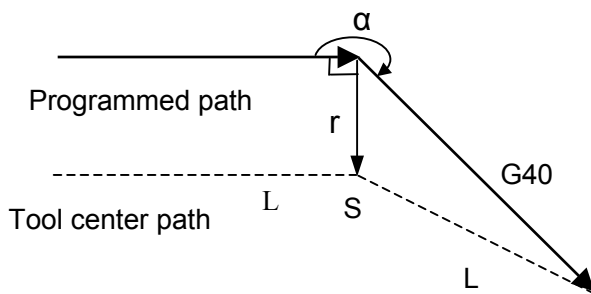


Fig.4-15a Linear to linear (inner side, offset cancellation)

2) Circular to linear

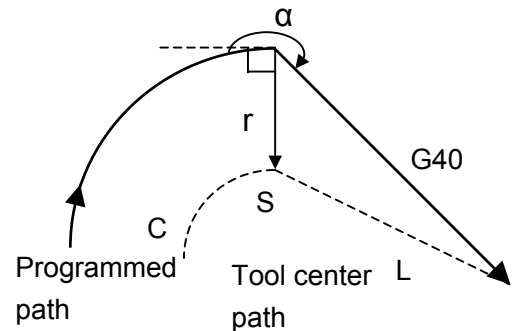


Fig.4-15b Circular to linear (inner side, offset cancellation)

(b) Tool movement along the outside of a corner at an obtuse angle ($180^\circ > \alpha \geq 90^\circ$)

1) Linear to linear

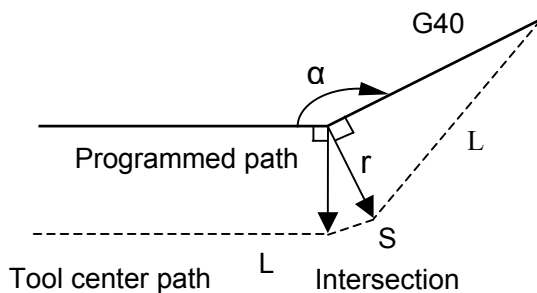


Fig.4-16a Circular to linear (obtuse, outside, offset cancellation)

2) Circular to linear

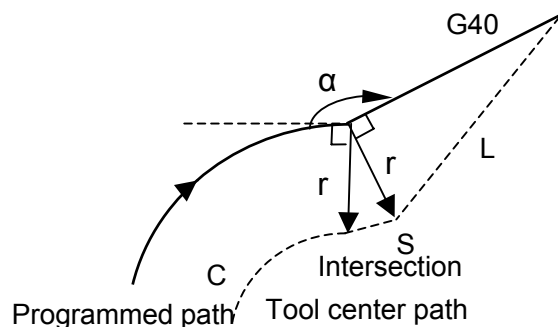


Fig.4-16b Circular to linear (obtuse, outside, offset cancellation)

(c) Tool movement along the outside of a corner at an acute angle ($180^\circ > \alpha \geq 90^\circ$)

1) Linear to linear

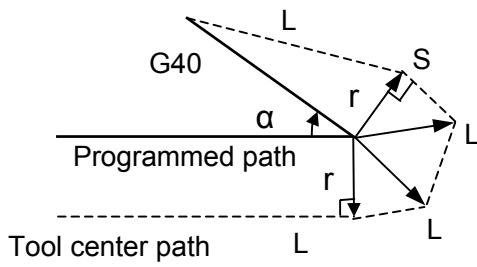


Fig.4-17a Linear to linear (acute angle, outside, offset cancellation)

2) Circular to linear

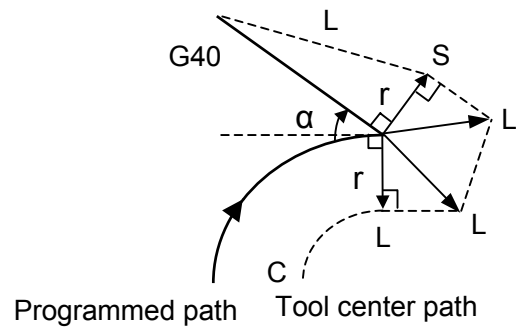


Fig.4-17b Linear to linear (acute angle, outside, offset cancellation)

(d) Tool movement along the corner outside at an acute angle less than 1 degree: linear to linear ($\alpha < 1^\circ$)

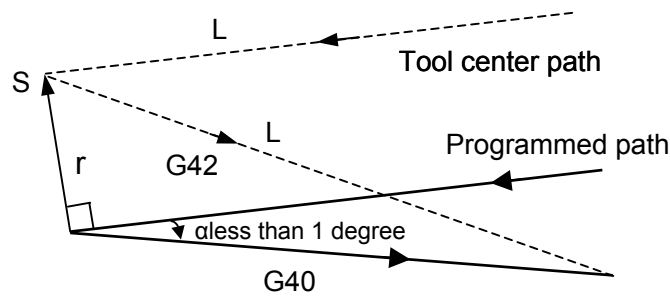


Fig.4-18 Linear to linear (the included angle less than 1 degree, outside, offset cancellation)

4.2.5 Interference check

Tool overcutting is called “interference”. The interference check function can check tool overcutting in advance. This interference check is performed even if the overcutting 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

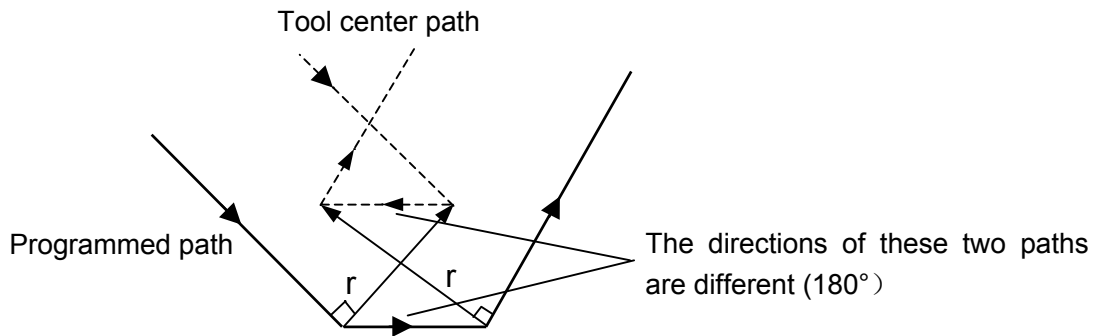


Fig.4-19a Machining interference (1)

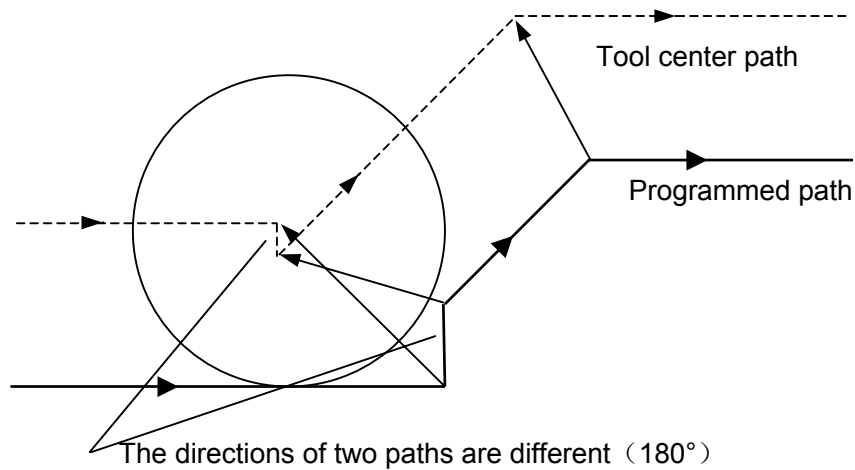


Fig.4-19b Machining interference

(2) If there is no interference actually, but it is treated as interference.

- 1) The groove depth less than the compensation value

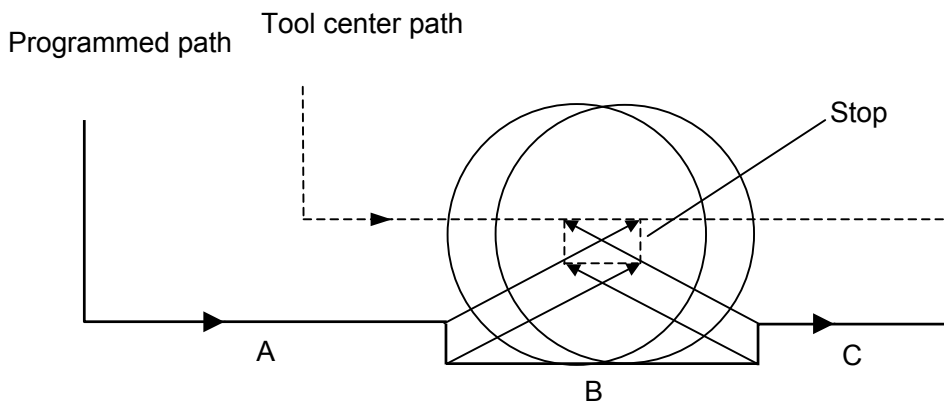


Fig.4-20 Exceptional case (1) treated as interference

There is no interference actually, but program direction in block B is opposite to the cutter radius compensation path. The cutter stops, and the alarm occurs.

2) The groove depth less than compensation value

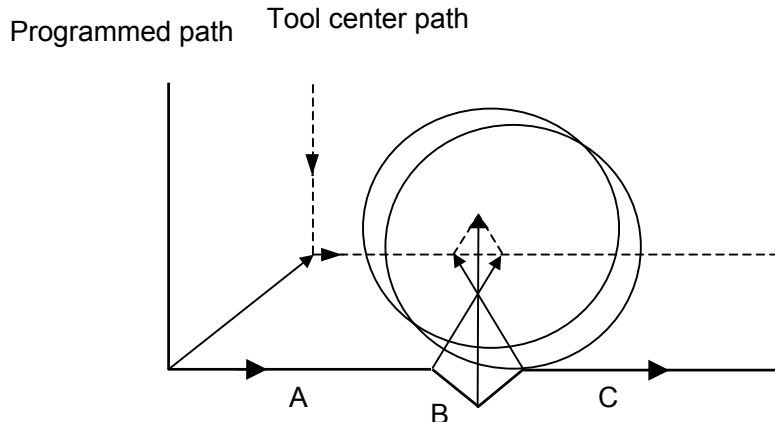


Fig.4-21 Exceptional case (2) treated as interference

There is no interference actually, but program direction in block B is opposite to the cutter radius compensation path. The cutter stops, and the alarm occurs.

4.2.6 Command of compensation vector cancel temporarily

If the following commands G92, G28, G29, coordinate command selection G54~G59 and canned cycle are specified in compensation mode, the compensation vector is temporarily cancelled and then automatically restored after these commands are executed. 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. And the tool moves to the intersection directly when the compensation mode restores.

- **Coordinate system setting command G92 and coordinate system selection command G54~G59**

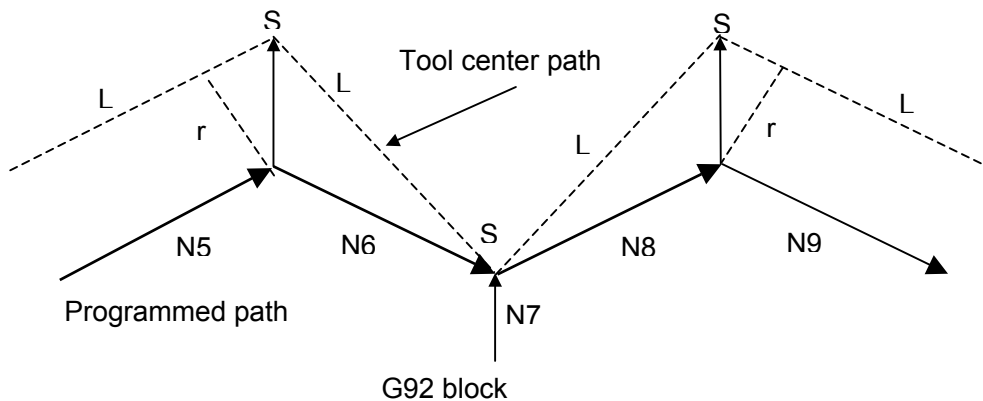


Fig.4-22 Temporary compensation vector by G92

Note: SS is indicated as the point stopped for twice in Single block mode.

- **Automatic return to the reference point G28**

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.

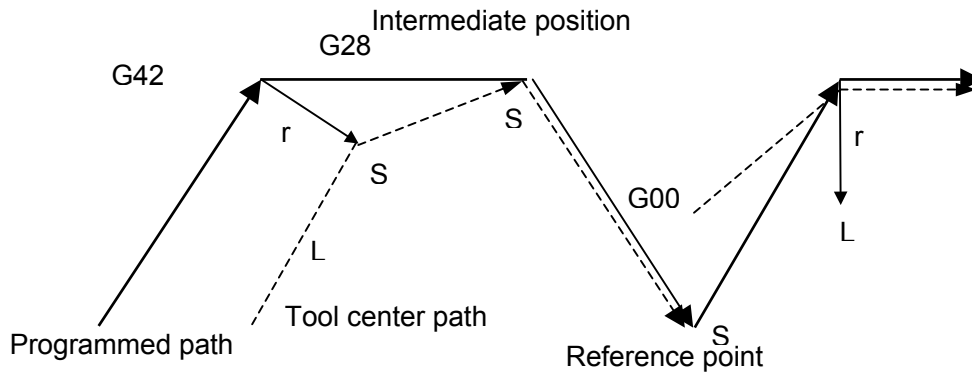


Fig.4-23 Temporarily cancel compensation vector by G28

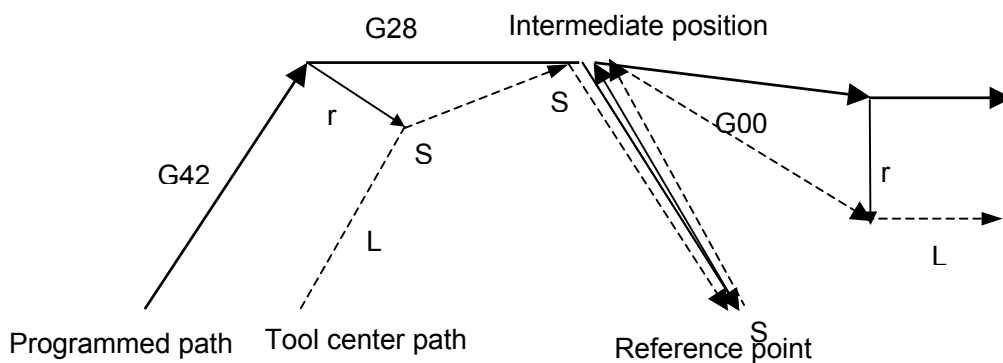


Fig. 4-24 G29 temporarily cancel compensation vector

● Canned cycle

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.

4.2.7 Exceptional case

● When the inner corner machining is less than tool radius

When the inner corner machining is less than tool radius, the inner offset of a tool will cause overcut. The tool stops and alarm occurs after moving at the beginning or at the corner in previous block. But if the switch of “Single block” is ON, the tool will be stopped at the end of the previous block.

● When a groove less than the tool diameter is machined

When the tool center moves opposite to the direction of programmed path, the overcutting 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.

● When a step less than the tool radius is machined

When 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 executed correctly and no alarm will be generated if the step is a straight line. (But the uncut part is reserved.)

● When the sub-program is contained in G code

CNC should be in compensation cancellation mode before calling the sub-program (namely, before the G98 is performed). Offset 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.

● When compensation value 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 executed again.

(b) If different compensation values are commanded in different blocks of a program, different compensation value will be compensated to the corresponding block. But if it is an arc, the alarm will be generated. For details, refer to the following explanation (c) about “arc data error in C type cutter radius compensation”.

● When the end point for the programming arc is not on the arc

When 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”.

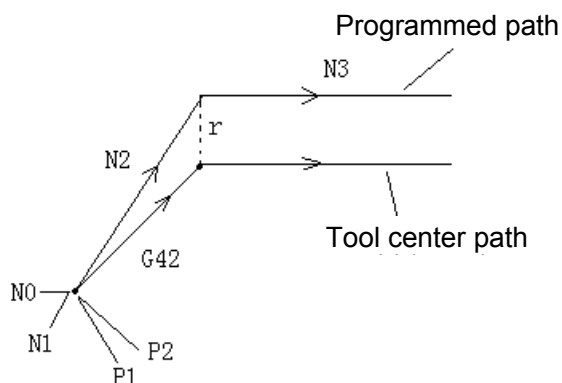
● Explanation for “two same points”

Radius compensation mode is that two blocks are read in advance, the transition point is calculated by 3 points (start, intermediate and end points) and the path movement is executed by this method. So, the following cases of “two same points” may occur:

- (a) Previous two points are same when start-up.
- (b) Posterior two points are same when start-up.
- (c) Previous two points in compensation are same.
- (d) Posterior two points in compensation are same.
- (e) Previous two points are same in cancellation.
- (f) Posterior two points are same in cancellation.

As for two same points: the point can be regarded as a straight line closing to zero, the transit points can be calculated by straight line (point) to straight line (point), straight line (point) to arc (point), arc (point) to straight line (point) and arc (point) to arc (point) when two points are same.

The following examples are the calculation for transit point that two points are same when start-up.



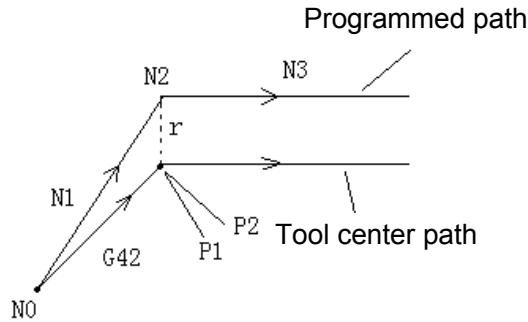
```
N0 G90 G00 X-50 Y-50
```

```
N1 G91 G1 G41 X0 Y0 D1 F800 ... no move
```

```
N2 G90 X0 Y0
```

```
N3 X50
```

“Two same points” will occur in startup of the above program, and the compensation will not be applied. Judging by the figure, the transit point of N0 and N1 is P1, transit point of N1 and N2 is P2, they share a same point.



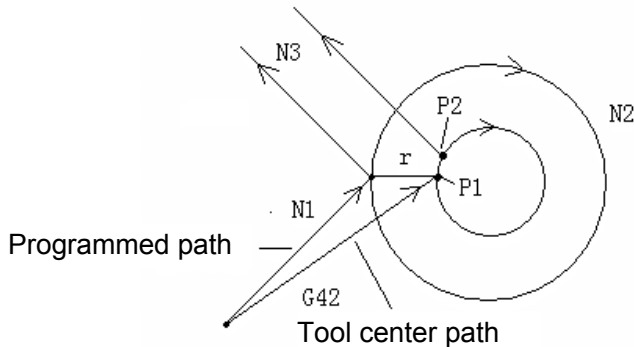
```
N0 G90 G00 X-50 Y-50
N1 G1 G41 X0 Y0 D1 F800
N2 G91 X0 Y0    no move
N3 X50
```

“Two same points” will occur in startup of the above program, and the compensation will be applied. Judging by the figure, the transit point of N1 and N2 is P1, transit point of N2 and N3 is P2, they share a same point.

So, in compensation mode, if “two same points” occur, the compensation value will be held on; in cancellation mode, the similar start-up are divided into “previous two same points” and “posterior two same points”

● The related alarms explanation about “the arc data error in C type cutter radius compensation”

(a) Example for an alarm may be issued in a full circle

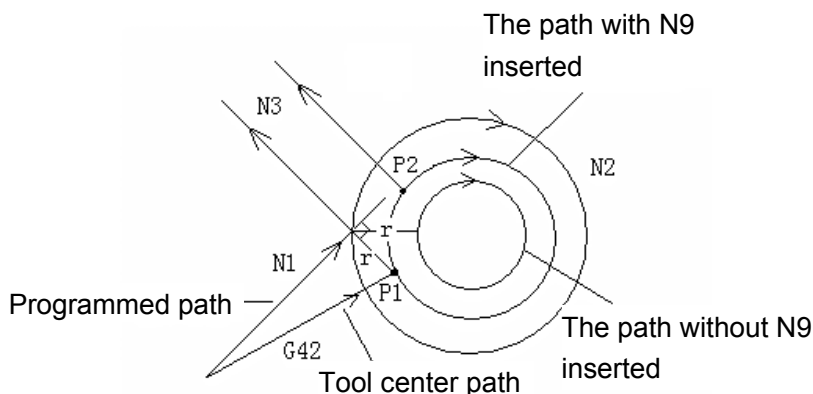


Examples:

```
N0 G90 G00 X-50 Y-50 Z50
N1 G01 G42 X0 Y0 D1 F800
N2 G02 I50
N3 G91 G01 X-50 Y-50 ...no move
```

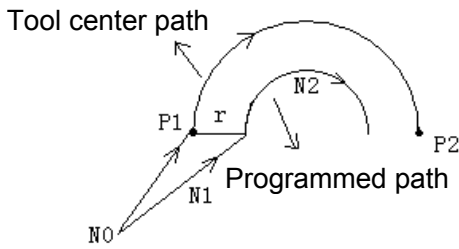
As for the figure at the left hand, the transit point of straight line N1 and arc N2 is P1, and the transit point of arc N2 and straight line N3 is P2, the radius compensation is r , the circular arc compensated is more than 360 degrees in this case.

If a block N9 G91 G0 X0 Y0 is inserted between N1 and N2, the alarm “arc data error in compensation C” occurs.



Point N9 inserted equals to point N1, they are regarded as “two same points”, so the transit point P1 is got, the P1 position is obviously different to the P1 without N9 block inserted. So the arc path cut by this transit point is different to the actual machining path. And the alarm “Arc data error in cutter radius compensation C” will be generated.

(b) Example for that an alarm may be issued for a non-full circle



Examples:

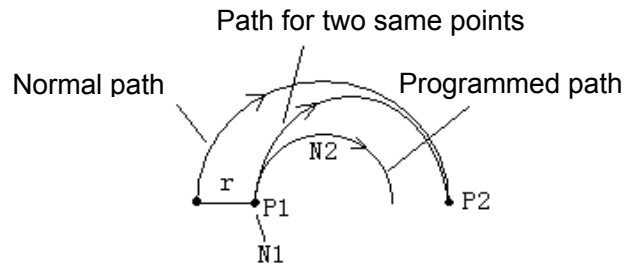
```
N0 G90 G00 X-50 Y-50 Z50
N1 G01 G41 X0 Y0 D1 F800
N2 G02 X50 R25
```

As for the figure at the left hand, The P1 and P2 are the transit points for cutter radius compensation, r is radius. This is the case of normal straight line to arc.

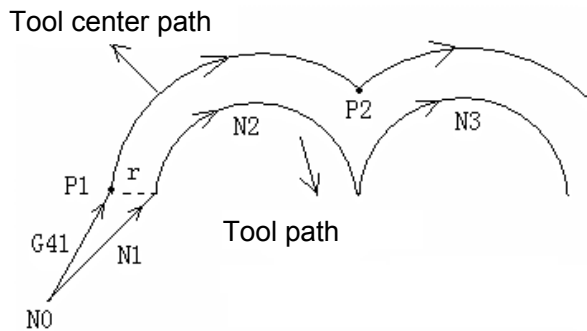
Alarm will be generated if the program is executed by

```
N0 G90 G00 X0 Y0 Z0
N1 G01 G41 X0 Y0 D1 F800 ...No move, start-up in origi
N2 G02 X50 R25
```

Because the N1 block has no move, which is equivalent to “Two same points”. They are treated as (path for two same points) “two same points” to get the transit points P1 and P2 separately. So, the arc path cut by this transit point is different to the actual machining path. So the alarm, “Arc data error in cutter radius compensation C” will be generated too.



(c) In the calculation of C type cutter radius compensation, if the compensation radius D is changed, the alarm will be generated.



Examples:

```
N0 G90 G00 X-50 Y-50 Z25
N1 G01 G41 X0 Y0 D1 F800
N2 G02 X50 R25
N3 G02 X100 R25
```

Programmed path and tool center path are performed for the above program, see the left figure.

If the compensation radius D is changed in N3 block, for example, the D2 is commanded in N3 block ($D2 \neq D1$), similar as (b), the alarm “arc data error in cutter radius compensation C” will be generated.

PART 2

OPERATION

CHAPTER 1 OPERATION MODE AND DISPLAY	I -1
1.1 Panel Division	I -1
1.1.1 State indication.....	I -2
1.1.2 Edit keypad	I -2
1.1.3 Menu display.....	I -3
1.1.4 Machine panel.....	I -4
1.2 Summary of Operation Mode	I -7
1.3 Display Interface	I -7
1.3.1 Position interface	I -10
1.3.2 Program interface.....	I -12
1.3.3 Tool offset, macro variable and tool life management interface.....	I -14
1.3.4 Alarm interface	I -23
1.3.5 Setting interface	I -25
1.3.6 BIT PARAMETER, DATA PARAMETER, SCREW-PITCH COMP interface..	I -29
1.3.7 CNC DIAGNOSIS, PLC STATE, PLC VALUE, machine soft panel, VERSION MESSAGE interface	I -30
1.3.8 LCD contrast adjustment	I -32
1.4 General Operation List.....	I -33
CHAPTER 2 POWER ON OR OFF AND PROTECTION	II -1
2.1 System Power On.....	II -1
2.2 System Power Off	II -1
2.3 Overtravel Protection	II -2
2.3.1 Hardware overtravel protection	II -2
2.3.2 Software overtravel protection.....	II -2
2.4 Emergency Operation	II -2
2.4.1 Reset.....	II -2
2.4.2 Emergency stop.....	II -3
2.4.3 Feed hold.....	II -3
2.4.4 Power off.....	II -3
CHAPTER 3 MANUAL OPERATION.....	III-1
3.1 Coordinate axis moving	III-1
3.1.1 Manual feed	III-1
3.1.2 Manual rapid traverse	III-1
3.1.3 Manual feedrate override adjustment.....	III-2
3.1.4 Manual rapid override adjustment	III-2
3.1.5 Relative coordinate clearing.....	III-2
3.2 Other Manual operations.....	III-3
3.2.1 Spindle CCW, CW, stop control.....	III-3
3.2.2 Spindle Jog.....	III-3
3.2.3 Cooling control	III-4
3.2.4 Lubrication control.....	III-4
3.2.5 Spindle override adjustment	III-4
CHAPTER 4 MPG/STEP OPERATION	IV-1
4.1 Step Feed.....	IV-1
4.1.1 Increment selection	IV-1

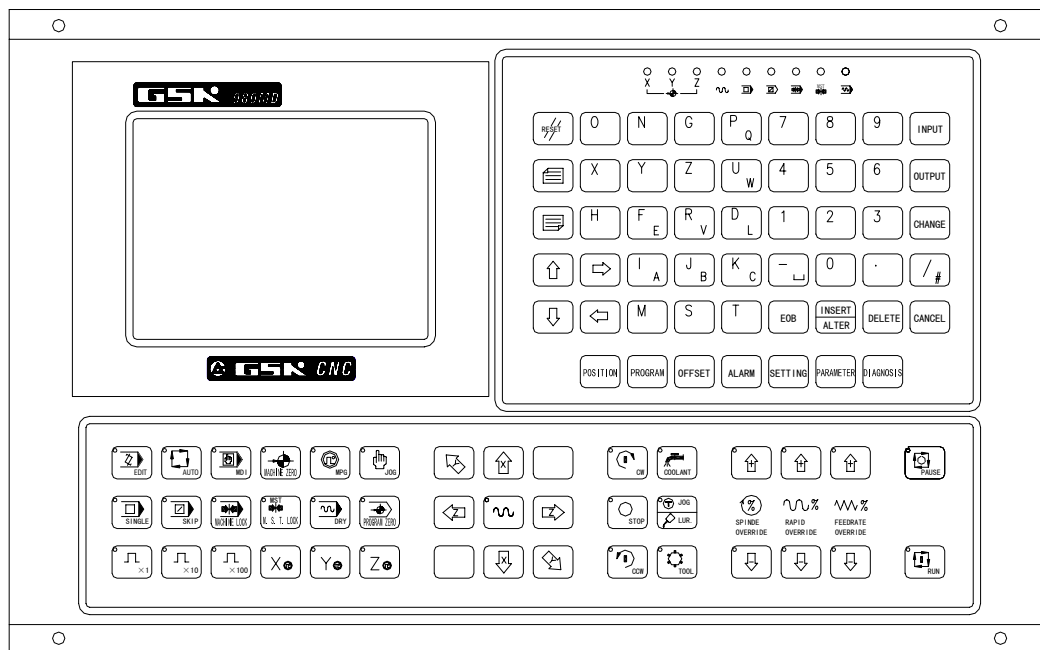
4.1.2	Moving direction selection	IV-2
4.2	Handwheel Feed	IV-2
4.2.1	Increment selection	IV-3
4.2.2	Moving axis and direction selection	IV-3
4.2.3	Explanation items	IV-4
CHAPTER 5 MDI OPERATION		V-1
5.1	Words Input.....	V-1
5.2	Words Execution.....	V-2
5.3	Parameter Setting.....	V-2
5.4	Data Modification.....	V-2
5.5	OUT Key Start	V-3
CHAPTER 6 PROGRAM EDIT AND MANAGEMENT		VI-1
6.1	Program Creation.....	VI-1
6.1.1	Creation of the block number	VI-1
6.1.2	Input of the program content	VI-1
6.1.3	Search of the character.....	VI-3
6.1.4	Insertion of the character	VI-4
6.1.5	Deletion of the character.....	VI-6
6.1.6	Modification of the character.....	VI-6
6.1.7	Deletion of a single block	VI-7
6.1.8	Deletion of the blocks	VI-7
6.1.9	Segment deletion	VI-8
6.2	Program Annotation.....	VI-9
6.2.1	Setup of the program annotation.....	VI-9
6.2.2	Modification of the program annotation.....	VI-11
6.3	Deletion of the Program.....	VI-11
6.3.1	Deletion a single program.....	VI-11
6.3.2	Deletion of all programs	VI-11
6.4	Selection of the Program	VI-11
6.4.1	Search method	VI-11
6.4.2	Scanning method	VI-12
6.4.3	Cursor method	VI-12
6.5	Execution of the Program	VI-13
6.6	Rename of the Program	VI-13
6.7	Copy of the Program.....	VI-13
6.8	Program Management.....	VI-14
6.8.1	Program list.....	VI-14
6.8.2	Software version	VI-14
6.8.3	Part program number	VI-14
6.8.4	The memory capacity and the capacity used	VI-14
CHAPTER 7 AUTO OPERATION		VII-1
7.1	Auto Run.....	VII-1
7.1.1	Selection of the program to be run	VII-1
7.1.2	Program start.....	VII-2
7.1.3	Stop of the auto run.....	VII-2
7.1.4	Auto run from an arbitrary block.....	VII-3

Contents

7.1.5	Adjustment of the feedrate override, rapid override	VII-3
7.1.6	Spindle override adjustment	VII-4
7.2	DNC running.....	VII-5
7.3	State on Running.....	VII-5
7.3.1	Single block execution.....	VII-5
7.3.2	Dry run	VII-6
7.3.3	Machine lock.....	VII-6
7.3.4	MST lock.....	VII-6
7.3.5	Block skip	VII-7
CHAPTER 8 MACHINE ZERO OPERATION		VIII-1
8.1	Machine Zero	VIII-1
8.2	Machine Zero Return Steps.....	VIII-1
CHAPTER 9 DATA SETTING, BACKUP AND RESTORE.....		IX-1
9.1	Data Setting.....	IX-1
9.1.1	Switch setting.....	IX-1
9.1.2	Graphic setting	IX-1
9.1.3	Parameter setting.....	IX-2
9.2	Data Restore and Backup.....	IX-7
9.3	The Password Setting and Modification	IX-8
9.3.1	Entry of the operation level.....	IX-9
9.3.2	Alteration of the password	IX-10
9.3.3	Set lower level.....	IX-11
CHAPTER 10 COMMUNICATION		X-1
10.1	Brief of GSK980MD TDComm Communication Software	X-1
10.1.1	Files download (PC→CNC)	X-2
10.1.2	Upload of the files (CNC→PC)	X-6
10.1.3	Setting option	X-8
10.2	Preparation Before Communication	X-8
10.3	Data Input (PC→CNC).....	X-10
10.3.1	Input of program	X-10
10.3.2	Input of the tool offset.....	X-12
10.3.3	Input of the parameter	X-13
10.4	Data Output (CNC→PC)	X-14
10.4.1	Output of a program.....	X-14
10.4.2	Output of all programs	X-16
10.4.3	Output of the tool offset.....	X-16
10.4.4	Output of the parameter	X-17
10.5	Communication between CNC and CNC	X-19

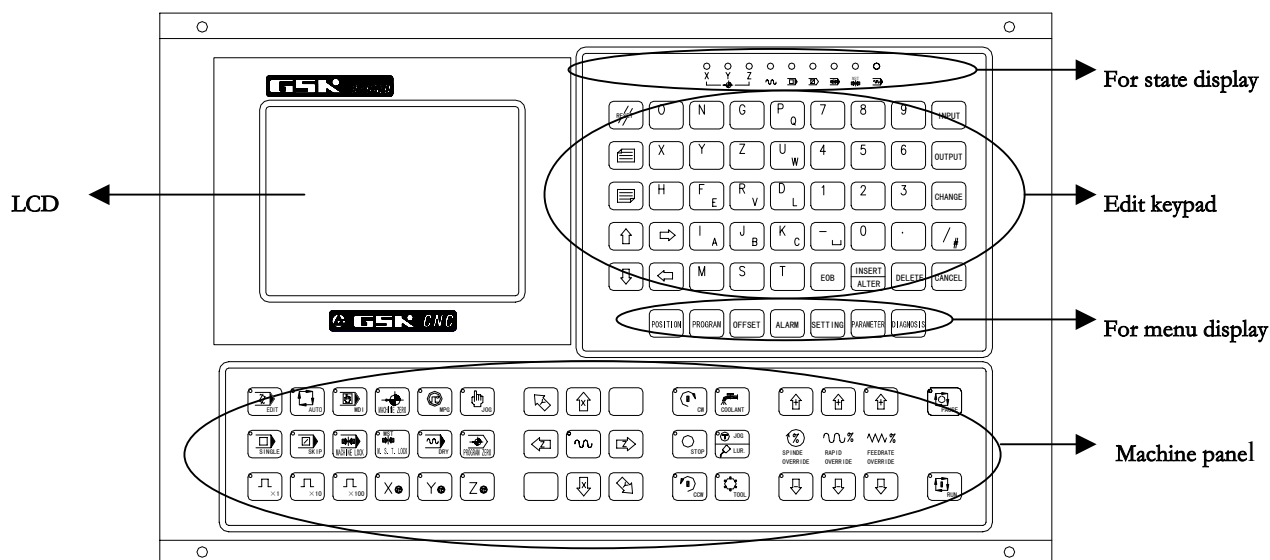
CHAPTER 1 OPERATION MODE AND DISPLAY

This GSK980MD system is employed with an aluminum alloy solid operator panel, the outline of it is shown as following:



1.1 Panel Division

This GSK980MD is employed with an integrated panel, which is divided as following:

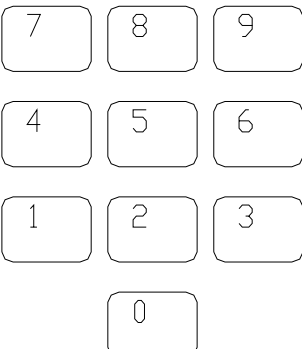







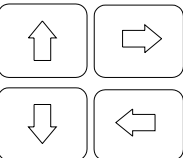



1.1.1 State indication





	X, Y, Z machine zero 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 key	For CNC reset, feed, stop output etc.
	Address key	Address input
		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, modification, 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.
	To enter program interface. There are PRG CONTENT, PRG LIST, PRG STATE pages in this interface.
	To enter TOOL OFFSET, MARRO interface(switching between two interfaces by pressing it repeatedly). OFFSET interface displays offset values; MARRO for CNC macro variables.
	To enter alarm interface. There are CNC, PLC ALARM pages in this interface.

	To enter Setting, Graphic interface (switching between two interfaces by pressing repeatedly). There are SWITCH, PARM OPERATION, PASSWORD SETTING in setting interface; there are GRAGH SET and TRACK pages in Graphic interface.
	To enter BIT PARAMETER, DATA PARAMETER, SCREW-PITCH COMP interfaces (switching between each interfaces by pressing repeatedly).
	To enter CNC DIAGNOSIS, PLC STATE, PLC VALUE, machine software panel, 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 message; machine software panel is used for machine soft keypad operation; the VERSION MESSAGE interface displays CNC software, hardware and PLC version No.











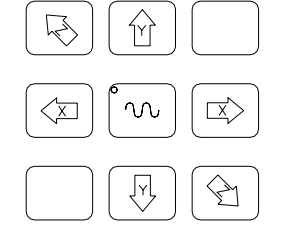






1.1.4 Machine panel












The keys function in GSK980MD 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 shown as following table:

Key	Name	Function explanation	Function mode
	Feed Hold key	Dwell commanded by program, MDI	Auto mode, MDI mode
	Cycle Start key	Cycle start commanded by program, MDI	Auto mode, MDI mode
     	Feedrate Override keys	For adjustment of the feedrate	Auto mode, MDI mode, Edit mode, Machine zero mode, MPG mode, Step mode, MANUAL mode, Program zero
   	Rapid override keys	For adjustment of rapid traverse	Auto mode, MDI mode, Machine zero mode, MANUAL mode, Program zero

Chapter 1 Operation Mode And Display

Key	Name	Function explanation	Function mode
  SPINDE OVERRIDE 	Spindle override keys	For spindle speed adjustment (spindle analog control valid)	Auto mode, edit mode, MDI mode, Machine zero mode, MANUAL mode, Step mode, MPG mode, Program zero
	JOG key	For spindle Jog ON/OFF	Machine zero mode, MANUAL mode, Step mode, MPG mode, Program zero
	Lubricating key	For machine lubrication ON/OFF	Machine zero mode, MANUAL mode, Step mode, MPG mode, Program zero
	Cooling key	For coolant ON/OFF	Auto mode, Edit mode, MDI mode, Machine zero mode, MANUAL mode, Step mode, MPG mode, Program zero
  	Spindle control keys	Spindle CCW Spindle stop Spindle CW	Machine zero mode, MANUAL mode, Step mode, MPG mode, Program zero
	Rapid traverse key	For rapid traverse /feedrate switching	Auto mode, MDI mode, Machine zero mode, MANUAL mode, Program zero
	Manual feed key	For positive/negative moving of X, Y, Z axis in Manual, Step mode	Machine zero mode, MANUAL mode, Program zero, Step 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, MANUAL mode, Step mode, MPG mode, Program zero

Key	Name	Function explanation	Function mode
	Single Block key	For switching of block/blocks execution, Single block lamp lights up if Single mode is valid	Auto mode, 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, MDI mode
	Machine Lock key	If the machine is locked, its lamp lights up, and X, Z axis output is invalid.	Auto mode, MDI mode, Edit mode, Machine zero mode, MANUAL mode, Step mode, MPG mode, Program zero
	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, 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, MDI mode
	Edit mode key	To enter Edit mode	Auto mode, MDI mode, Machine zero mode, MANUAL mode, Step mode, MPG mode, Program zero
	Auto mode key	To enter Auto mode	MDI mode, Edit mode, Machine zero mode, MANUAL mode, Step mode, MPG mode, Program zero
	MDI mode key	To enter MDI mode	Auto mode, Edit mode, Machine zero mode, MANUAL mode, Step mode, MPG mode, Program zero
	Machine zero mode key	To enter Machine zero mode	Auto mode, MDI mode, Edit mode, MANUAL mode, Step mode, MPG mode, Program zero
	Step/MPG mode key	To enter Step or MPG mode (one mode by parameter)	Auto mode, MDI mode, Edit mode, Machine zero mode, MANUAL mode, Program zero
	Manual mode key	To enter Manual mode	Auto mode, MDI mode, Edit mode, Machine zero mode, Step mode, MPG mode, Program zero

Key	Name	Function explanation	Function mode
	DNC mode key	To enter DNC mode	To enter DNC mode by pressing this key in Auto mode

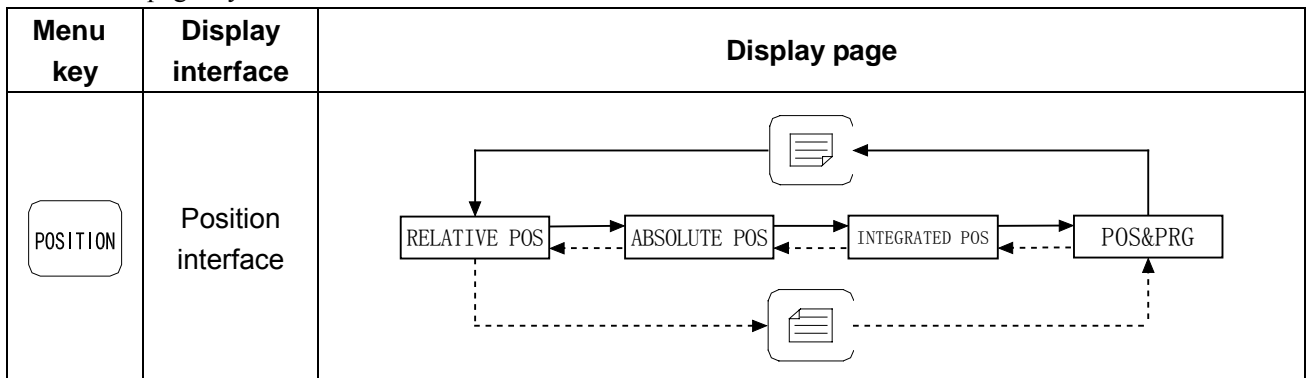
1.2 Summary of Operation Mode

There are 7 modes that include Edit, Auto, MDI, Machine zero, Step/MPG, Manual, Program Zero modes in this GSK980MD.

- **Edit mode**
In this mode, the operation of part program setup, 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, Z 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 9 interfaces such as Position, Program etc., and there are multiple pages in each interface. Each interface (page) is separated with the operation mode. See the following figures for the display menu, display interface and page layers:






Menu key	Display interface	Display page
PROGRAM	Program interface	<pre> graph LR PRG_CONTENT[PRG CONTENT] --> PRG_STATE[PRG STATE] PRG_STATE --> PRG_LIST[PRG LIST] PRG_LIST --> PRG_STATE PRG_STATE --> PRG_CONTENT PRG_CONTENT -.-> Doc1[Document Icon] Doc1 -.-> PRG_LIST PRG_LIST -.-> Doc2[Document Icon] Doc2 -.-> PRG_STATE </pre>
OFFSET	TOOL OFFSET interface	<pre> graph LR TOOL_OFFSET_1[TOOL OFFSET 1] --> TOOL_OFFSET_i[TOOL OFFSET i] TOOL_OFFSET_i --> TOOL_OFFSET_5[TOOL OFFSET 5] TOOL_OFFSET_5 --> TOOL_OFFSET_i TOOL_OFFSET_i --> TOOL_OFFSET_1 TOOL_OFFSET_1 -.-> Doc1[Document Icon] Doc1 -.-> TOOL_OFFSET_5 </pre>
	MACRO interface	<pre> graph LR PLC_DATA_1[PLC DATA 1] --> PLC_DATA_i[PLC DATA i] PLC_DATA_i --> VERSION_MESSAGE[VERSION MESSAGE] VERSION_MESSAGE --> PLC_DATA_i PLC_DATA_i --> PLC_DATA_1 PLC_DATA_1 -.-> Doc1[Document Icon] Doc1 -.-> VERSION_MESSAGE </pre>
	Tool life interface	<pre> graph LR TOOL_LIFE_1[TOOL-LIFE 1] --> TOOL_LIFE_i[TOOL-LIFE i] TOOL_LIFE_i --> TOOL_LIFE_n[TOOL-LIFE n] TOOL_LIFE_n --> TOOL_LIFE_i TOOL_LIFE_i --> TOOL_LIFE_1 TOOL_LIFE_1 -.-> Doc1[Document Icon] Doc1 -.-> TOOL_LIFE_n </pre>
ALARM	CNC alarm	CNC ALARM

Menu key	Display interface	Display page
	PLC alarm/warn	PLC ALARM/WARN
	Alarm log	ALARM LOG
SETTING	Setting interface	
	G54 setting	SET (G54~G59)
	Graphic interface	
PARAMETER	Bit parameter	
	Data parameter	
	Screw-pitch parameter	
DIAGNOSIS	CNC diagnosis	

Menu key	Display interface	Display page
	PLC state	
	PLC data	
	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  keys.

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.

ABSOLUTE POS	
00000	N00000
X	0.000
Y	0.000
Z	0.000
PRG.F : 100	G CODE: G00 ,G94
ACT.F : 0	PART CNT: 0
FED OVRI: 150%	CUT TIME: 0:00:00
RAP OVRI: 100%	S0000 T00 H00
MDI	

PRG. F: a rate specified by F code in program

Note It displays “PRG. F” In Auto, MDI mode; “JOG F. in Machine zero,Manual mode”;“HNDL INC”in MPG mode; “STEP INC”in Step mode.

ACT. F: actual speed after feedrate override in a machining.

FED OVRI: an override by feedrate override switch.

G CODE: modal value of 01 group G code and 03 group G code

PART CNT: part number plusing 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 following:

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 is a must.



T0100: Current tool No. and tool offset No.



2) RELATIVE POS display page

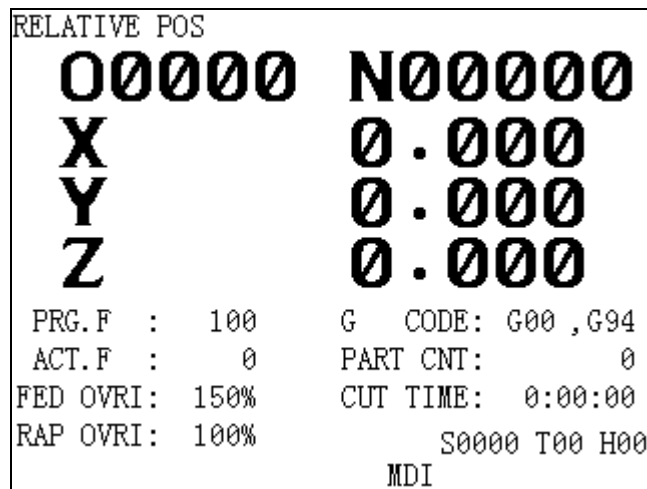
The X, Y, Z axis coordinates displayed are the current position relative to the relative reference point, and they are held on at CNC power on. They can be cleared at any time. If X, Y, Z axis relative coordinates are cleared, the current position will be the relative reference point. When CNC parameter 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  key till the “X” in the page blinks, press  key to clear X coordinate;



In RELATIVE POS page, press and hold  key till the “Y” in the page blinks, press  key to clear Y coordinate;



In RELATIVE POS page, press and hold  key till the “Z” in the page blinks, press  key to clear Z coordinate;



The method for X, Y, Z axis relative coordinates divided by 2:

In RELATIVE POS page, press and hold  key till the “X” in the page blinks, press  key, X coordinate will be divided by 2;

In RELATIVE POS page, press and hold  key till the “Y” in the page blinks, press  key, Y coordinate will be divided by 2;

In RELATIVE POS page, press and hold  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 of the target position by block or MDI command to the current position.

The display page is as following:


INTEGRATED POS		00000 N00000	
(RELATIVE)		(ABSOLUTE)	
X	0.000	X	0.000
Y	0.000	Y	0.000
Z	0.000	Z	0.000
(MACHINE)		(DIST TO GO)	
X	0.000	X	0.000
Y	0.000	Y	0.000
Z	0.000	Z	0.000
		S0000 T00 H00	
		MDI	



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.

POS & PRG		00000 N00000	
(RELATIVE)		(ABSOLUTE)	
X	0.000	X	0.000
Y	0.000	Y	0.000
Z	0.000	Z	0.000
00000;			
G0 G54 G90 X0 Y0 Z0;			
X10 Y10;			
X-10 Y-10;			
M99;			
		S0000 T00 H00	
		MDI	

1.3.2 Program interface



Press  to enter Program interface, which has three pages such as PRG CONTENT, PRG STATE, PRG

LIST in non-Edit modes, and they can be viewed by  or  keys. There is only PRG CONTENT page

in Edit mode, all the blocks of the current program can be shown by pressing  or  keys.

1) PRG CONTENT page

In this page, the PRG CONTENT including current block can be displayed. In Edit mode, the PRG CONTENT

can be viewed forward or backward by pressing  or  keys.

```
PRG CONTENT ROW2 COL1 00000 N00000
00000;
G0 G54 G90 X0 Y0 Z0;
X10 Y10;
X-10 Y-10;
M99;
%

S0000 T00 H00
EDIT
```

2) PRG STATE page

In this page, it displays the G, M, S, T, F codes. In Auto and MDI mode, it displays the current block.

```
PRG STATE (MDI) 00000 N00000
(MODAL)
X M G00 X
Y S G17 Y
Z T G90 Z
U H G94 V
V D G98 W
W L G54 R
I G21 F 100
J G40 M
K G49 S 00
R P 0
F Q
P
Q

S0000 T00 H00
MDI
```

3) PRG LIST page

In this page it displays:

(a) VERSION NO.: CNC current software version

(b) PART-PRG NO.: numbers of the programs that can be saved and programs saved by CNC (including subprogram)

(c) MEMORY SIZE: the max. capacity for the programs that can be saved and the capacity that has been taken up by programs.

(d) PRG LIST: number of the program saved by name size order

```

PRG LIST                                00000 N00000
VERSION NO. :GSK980MD                  V07.10.20
PART-PRG NO. :   23
MEMORY SIZE: 20.0 MB; USED: 287 KB
PROGRAM LIST:
00000 00001 00002 00003 00004 00006
00008 00010 00020 00022 00032 00041
00042 00088 00099 00100 00111 00232
00444 01114 04142 07777 07998

PRG SIZE: 53B   NOTE:
                                S0000 T00 H00
                                MDI
    
```

1.3.3 Tool offset, macro variable and tool life management interface

OFFSET

is a compound key, press

OFFSET

key once in other page, it enters the TOOL OFFSET page, press

OFFSET

key again, it enters the MACRO interface.

1 OFFSET interface

There are 5 tool offset pages in this interface, and 33 offset No. (No.000~No.032) available for user, which

can be shown as following by pressing



or



keys.

```

TOOL OFFSET & WEAR                    00000 N00000
NO.   OFT(H)  WEAR(H)  OFT(D)  WEAR(D)
01    0.000   0.000   0.000   0.000
02    0.000   0.000   0.000   0.000
03    0.000   0.000   0.000   0.000
04    0.000   0.000   0.000   0.000
05    0.000   0.000   0.000   0.000

RELATIVE POS
X    0.000  Y    0.000  Z    0.000
NO.  01                                S0000 T00 H00
                                MDI
    
```

2 Macro variable interface

There are 3 pages in this interface, which can be shown by pressing



or



keys. In Macro page

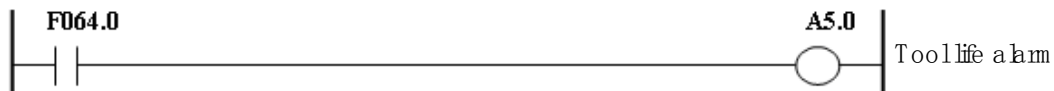
there are 48 (No.200~No.231 and No.500~No.515) macro variables which can be specified by macro command or set by keypad. The macro variable values are held on after power down.

MACRO		00000 N00000	
NO.	DATA	NO.	DATA
100	0.000	108	0.000
101	0.000	109	0.000
102	0.000	110	0.000
103	0.000	111	0.000
104	0.000	112	0.000
105	0.000	113	0.000
106	0.000	114	0.000
107	0.000	115	0.000
RELATIVE			
X	0.000	Y	0.000
Z	0.000		
NO. 100		S0000 T00 H00	
MDI			

3 Tool life management

Note The tool change signal TLCH: F064#0 should be added for PLC when using this function.


Ladder example:




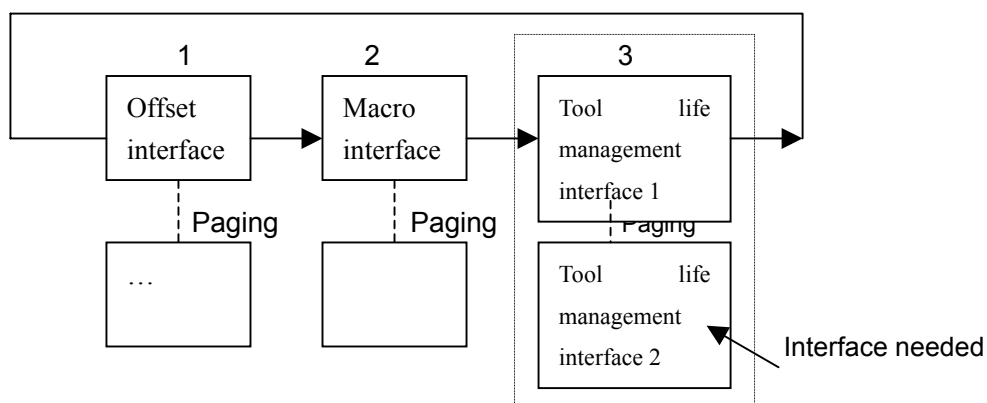
- **Using of tool life management function**

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**

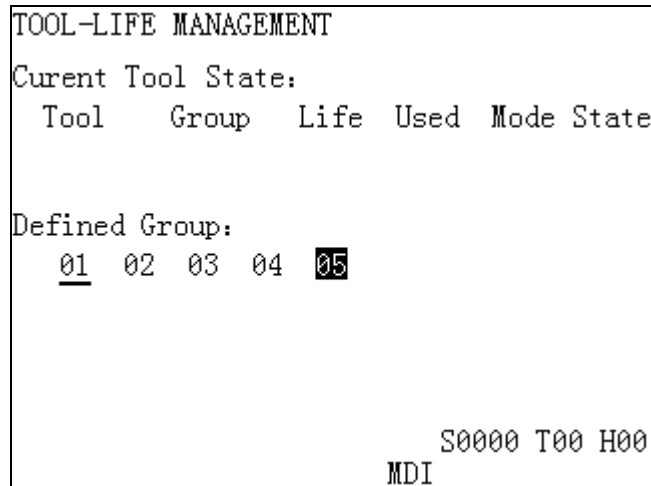
The tool life management is controlled by  key, which is displayed in the third sub-interface, and it is composed by 2 pages (paging by page keys).

Interface shown by pressing  key repeatedly



Tool life management display (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 next page.



i . Display explanation



<Current Tool State>: It displays the life data of the current tool 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)

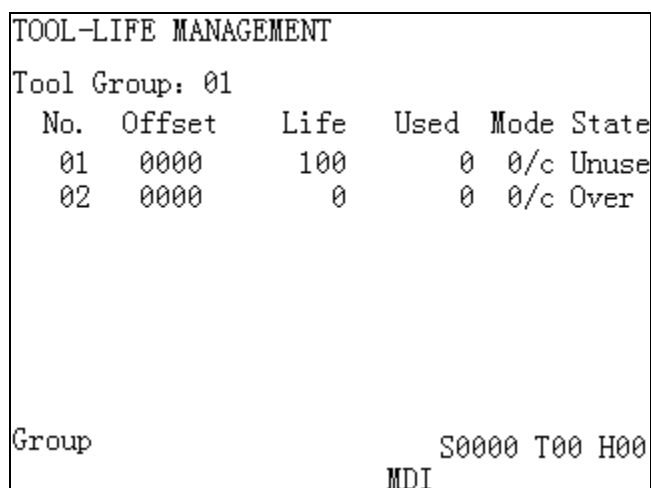
< Defined Group >: It only displays the group numbers which have been defined, and the undefined are not shown. The group number with the backlight means that all the tool life in that group has expired.

ii . Deletion of all data defined

In that page, press  +  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:

- Directly input the group number in the “Tool Group ”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: 22, and the 1st tool will be defined by system automatically:

TOOL-LIFE MANAGEMENT					
Tool Group: 05					
No.	Offset	Life	Used	Mode	State
01	0000	0	0	0/c	Over
Group					
S0000 T00 H00					
MDI					

- ii. Move the cursor to select the group number in the “Defined Group” of the 1st page, and it displays the group content as paging for 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 when paging next page.

1. Definition of tool life data

The setting of tool life data has 2 types: ① NC programming and program execution ② Directly input from the tool life management interface

① NC programming setting Example:

```

O0020 (CUTTER LIFE DATA)
G10 L3;           Life data setting start
P01;              Group No.
T0101 L500 N0;    Tool No., life, mode setting
T0201 L600 N1;
P02;              Next group No.
T0303 L200 N0;    Tool No., life, mode setting
T0304 L300 N0;
G11;              Data setting over
M30;
    
```

P: Tool group number 1 – 32

L: each tool life 0 – 999999 minutes or 0 – 999999 times

N: mode selection (0—time, 1—times)

T: Tool and tool offset number


Notice of programming



- i. The group number specified by P may be not continuous, but should be in ascending sequence that can be seen clearly in monitoring interface.
- ii. If life data L_ is omitted, the tool life is 0; if mode N_ is omitted, the tool mode is 0(min). In this case it only counts with no alarm output.
- iii. The words between G10 L3 and G11 are all ignored.

② Input from tool life management interface

To set 3 levels operation authority in MDI mode, page to the 2nd page of the tool life management interface:

A. Data modification:

i. Press  key to enter modification mode. This mode supports up and down, left and right cursor moving, as well as data input (tool offset, tool life, used, mode).

ii. In modification mode, input data from the input column, and press  key to confirm it. Then press  key, it may exit this modification mode.

TOOL-LIFE MANAGEMENT						
Tool Group: 02						
No.	Offset	Life	Used	Mode	State	
01	0103	3500	920	0/c	Unuse	
02	0202	4000	2331	0/c	Unuse	
03	0101	4000	178	0/c	Unuse	
04	0302	5000	20	1/t	Unuse	
05	0102	5000	566	1/t	Unuse	
06	0304	5000	100	0/c	Unuse	
Life = 999999_ S0000 T00 H00						
MDI						

Before modification

TOOL-LIFE MANAGEMENT					
Tool Group: 02					
No.	Offset	Life	Used	Mode	State
01	0103	999999	920	0/c	Unuse
02	0202	4000	2331	0/c	Unuse
03	0101	4000	178	0/c	Unuse
04	0302	5000	20	1/t	Unuse
05	0102	5000	566	1/t	Unuse
06	0304	5000	100	0/c	Unuse
Life = S0000 T00 H00					
MDI					

After modification

B. Data insertion

Insert any sequence number in current page, press [N] —> [01~08] —> [INPUT] key to insert the new line, so the initial val defin ue ised as follows:

No.	Offset	Life	Used	Mode	State
N	0000	0	0	0	Over

i. To insert backward, press [N] —> [Sequence No.] —> [INPUT] :

TOOL-LIFE MANAGEMENT					
Tool Group: 02					
No.	Offset	Life	Used	Mode	State
01	0103	999999	920	0/c	Unuse
02	0202	4000	2331	0/c	Unuse
03	0101	4000	178	0/c	Unuse
04	0302	5000	20	1/t	Unuse
05	0102	5000	566	1/t	Unuse
06	0304	5000	100	0/c	Unuse
No. = 7_ S0000 T00 H00					
MDI					

Before insertion

TOOL-LIFE MANAGEMENT					
Tool Group: 02					
No.	Offset	Life	Used	Mode	State
01	0103	999999	920	0/c	Unuse
02	0202	4000	2331	0/c	Unuse
03	0101	4000	178	0/c	Unuse
04	0302	5000	20	1/t	Unuse
05	0102	5000	566	1/t	Unuse
06	0304	5000	100	0/c	Unuse
07	0000	0	0	0/c	Over
Life = S0000 T00 H00					
MDI					

After insertion

ii. To insert in between, press [N] —> [Sequence No.] —> [INPUT] : the original sequence No. data move backward.

TOOL-LIFE MANAGEMENT						
Tool Group: 02						
No.	Offset	Life	Used	Mode	State	
01	0103	999999	920	0/c	Unuse	
02	0202	4000	2331	0/c	Unuse	
03	0101	4000	178	0/c	Unuse	
04	0302	5000	20	1/t	Unuse	
05	0102	5000	566	1/t	Unuse	
06	0304	5000	100	0/c	Unuse	
07	0000	0	0	0/c	Over	
No. = 5_ S0000 T00 H00						
MDI						

Before insertion

TOOL-LIFE MANAGEMENT						
Tool Group: 02						
No.	Offset	Life	Used	Mode	State	
01	0103	999999	920	0/c	Unuse	
02	0202	4000	2331	0/c	Unuse	
03	0101	4000	178	0/c	Unuse	
04	0302	5000	20	1/t	Unuse	
05	0000	0	0	0/c	Over	
06	0102	5000	566	1/t	Unuse	
07	0304	5000	100	0/c	Unuse	
08	0000	0	0	0/c	Over	
Life = S0000 T00 H00						
MDI						

After insertion

C. Data deletion:

- i. To delete all group data, press [CANCEL] + [G] keys

TOOL-LIFE MANAGEMENT						
Curent Tool State:						
Tool	Group	Life	Used	Mode	State	
Defined Group:						
01	03	04	05			
S0000 T00 H00						
MDI						

Before deletion

TOOL-LIFE MANAGEMENT						
Curent Tool State:						
Tool	Group	Life	Used	Mode	State	
Defined Group:						
—						
S0000 T00 H00						
MDI						

After deletion

- ii. To delete any group data, press [P] —> [group No.] —> [DELETE] keys;

TOOL-LIFE MANAGEMENT						
Tool Group: 02						
No.	Offset	Life	Used	Mode	State	
01	0103	999999	920	0/c	Unuse	
02	0202	4000	2331	0/c	Unuse	
03	0101	4000	178	0/c	Unuse	
04	0302	5000	20	1/t	Unuse	
05	0102	5000	566	1/t	Unuse	
06	0304	5000	100	0/c	Unuse	
07	0000	0	0	0/c	Over	
Group = 2_ S0000 T00 H00						
MDI						

Before deletion

TOOL-LIFE MANAGEMENT						
Tool Group: 03						
No.	Offset	Life	Used	Mode	State	
01	0000	100	0	0/c	Unuse	
Group = S0000 T00 H00						
MDI						

After deletion

- iii. To delete any sequence number in current page, press [N] —> [01~08] —> [DELETE] .

TOOL-LIFE MANAGEMENT					
Tool Group: 02					
No.	Offset	Life	Used	Mode	State
01	0103	999999	920	0/c	Unuse
02	0202	4000	2331	0/c	Unuse
03	0101	4000	178	0/c	Unuse
04	0302	5000	20	1/t	Unuse
05	0000	0	0	0/c	Over
06	0102	5000	566	1/t	Unuse
07	0304	5000	100	0/c	Unuse
08	0000	0	0	0/c	Over
No.	=	5_	S0000 T00 H00		
MDI					

Before deletion

TOOL-LIFE MANAGEMENT					
Tool Group: 02					
No.	Offset	Life	Used	Mode	State
01	0103	999999	920	0/c	Unuse
02	0202	4000	2331	0/c	Unuse
03	0101	4000	178	0/c	Unuse
04	0302	5000	20	1/t	Unuse
05	0102	5000	566	1/t	Unuse
06	0304	5000	100	0/c	Unuse
07	0000	0	0	0/c	Over
Life = S0000 T00 H00					
MDI					

After deletion

D. New group definition:

Press [P] ——> [group No.] ——> [INPUT] keys to set up the new group and new line.

E. Illegal date treatment

If the data input is illegal, the input is invalid, and an alarm will be issued.

Cautions:

- All tool life data are reserved after power down.
- If the tool life preset program is executed (such as O0020 above), it will completely clear all original life data and the system will preset those life data based on the program requirement.
- Manual modification of life data is prohibited during the program execution. They can't be modified till the execution is quitted.(except the execution of tool life preset program)

2. Usage of tool life function

Format:

```
O0050;
...
Txx99;          Use the tool life management of XX group
...
Txx88;          Cancel the offset of the Group xx
...
M30;
```

Using example:

T0199; Use the tool management of 01 group

...

T0188; Cancel the offsets of the Group 01

...

T0405; Use No.04 tool, offset 05, no management

...

T0400; Cancel the offset of the No.04 tool

...

T0299; Use the tools of No.02 group

...

T0199; Use the tools of No.01 group, use next tool if it contains multiple tools

Counting of tool life:

There are 2 types for tool life counting, time and times.

If the counting result is that the using life is more than or equal to the setting value, the next group number selection command will choose the spared tools in the group and count the tools selected. If all the tool lives in the group expired with no spared tools left in the group, it continues to count and an alarm will be issued to PLC.

Counting in MDI mode is defined by parameter (No.002 # 3) .

1. Counting by time

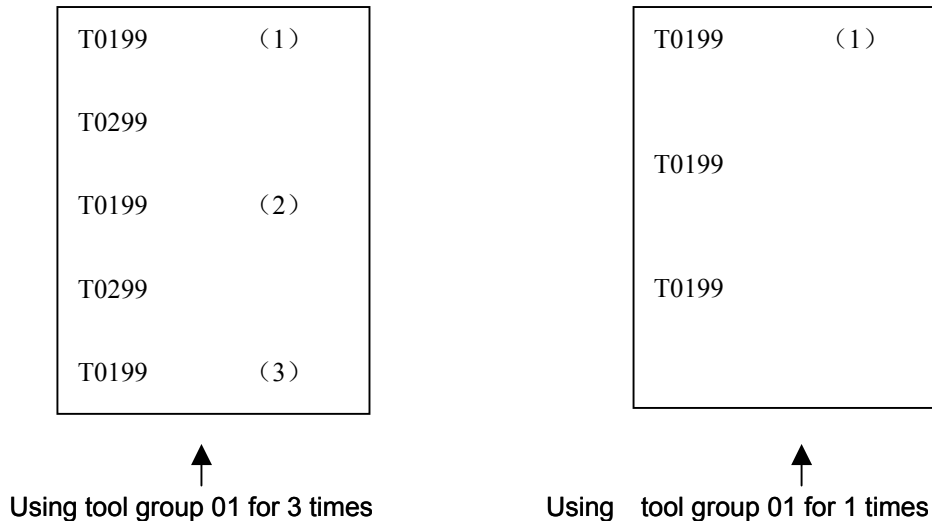
In cutting mode (G01,G02,G03,G32,G33,G34 etc.), the tool using time is counted by unit of minute, but in G04 dwell, feed hold, machine lock, MST lock, dry run modes, no counting will be done.

2. Counting by times(2 types)

a) Pattern 1 (No.002#2=0)

For the tool number change by tool group selection command (Txx99) and cutting mode(except the machine lock, MST lock, dry run modes), it counts. If the system doesn't enter the cutting mode with the number changed, it doesn't count.

Example:



b) Pattern 2 (No.002#2=1) (No.002#2=1)

i. It increases 1 from the part program beginning to M30(M99) for the tool group in cutting, if it is reset in the midway, the run times is not accumulated.

ii. It doesn't count in machine lock, MST lock, dry run modes.

3. CNC-PLC signal of tool life function

Tool change signal TLCH:	F064#0
New tool selection signal TLNW:	F064#1
Tool change reset signal TLRST:	G048#7
Tool skip signal TLSKP:	G048#5
Tool group selection signal TL01~TL16:	G047#0~#4 (0~31 means No.1~32 group tools)

Tool change signal

TLCH (F064 # 0)

[Type] Output signal

[Function] It notifies PLC of the expiration of the last tool in this group.

[Output condition] The signal is set to 1 while as :

- Life of the last tool in a group expires and all the tool lives in that group expire.
- While as the signal is set to 0:
- Life of the last tool in a group doesn't expire.
- TLRST signal for tool change reset is set to 1.

Note: All the tool groups applied in the program will be checked at the program end (M02, M30, and M99). TLCH signal will be given if all tool lives in a group expire.

Reset signal for tool change

TLRST (G048#7) TLRST (G048#7)

[Type] Input signal

[Function] It clears all the data which have been executed.

[Action] When this signal is set to 1, action of Control Unit is as follows :

The life data of tool used in all groups are cleared; and the state of tools is reset for unused.

Note: TLRST signal for tool change reset effects only when the automatic operation signal OP is "0".

New tool selection Signal TLNW (F064#1)

TLNW (F064#1)

[Type] Output signal

[Function] It notifies that a new tool is selected from a group.

It may be used under some conditions, for example, when a new tool is selected and offsetting is automatic measured.

[Output condition] The signal is set to 1 while as:

- A new tool from a group is selected.

The signal is set to 0 while as:

- The FIN signal of miscellaneous functions is set to 1.

Tool skipping signal TLSKP (G048#5)

TLSKP (G048#5)

[Type] Input signal

[Function] Change the unexpired tool by one of the two methods as follows :

1) When the parameter TLFEJMPGUP (2#4) is 1, the tool group number is specified by group number selection signal, and change the TSKP signal for tool skipping for "1", and the next T code will skip the current group tool being used to use the first unexpired tool in the specified group.

2) When the parameter TLFEJMPGUP (2#4) is 0, no group number will be specified. Change the TSKP signal for tool skipping for "1", the machine will use the next tool in current group .

[Action] When the signal is set to 1, the operation of Control Unit is as follows:

Next tool in current or specified group is selected by the tool group number selection signal specified together with skipping command.

Note: The signals for cycle start indicator and feed hold indicator should be "0" before inputting TSKP signal.

Tool group number selection signals TL01~TL16 (G047#0~#4)

TL01~TL16 (G047#0~#4)

[Type] Input signal

[Function] Tool group number must be given in advance by using tool group number selection signal T01~16 while inputting TLSKP signal .

Specify the following value by the binary system:

Tool group number specified +1

[Action] Select the specified tool group.

4. Updated alarm number for tool life function

019 Tool group number exceeding range (1~32) in Tool Life Management

024 No G11 in program

025 No tool in current group in Tool Life Management


026 No definition for the current group in Tool Life Management



027 Tool number of the current group is over 8 in Tool Life Management

028 Tool Life Management invalid, G10 L3 command unallowed

029 G11 before G10 unallowed

1.3.4 Alarm interface

Press  key to enter Alarm interface, there are CNC ALARM, PLC ALARM 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. may be viewed by moving the cursor. If there are 2 alarms in current page, it displays as following:

```

PLC ALARM /WARN          00000 N00000
CNC ALM  0, PLC ALM  2 / PLC WARN  0
1001 1032

ALM NO. 1001 BIT ADDRES: A0000.1
spindle alarm

ALM          JOG
    
```

Page as the cursor locates at the alarm No.1001

```

PLC ALARM /WARN          00000 N00000
CNC ALM  0, PLC ALM  2 / PLC WARN  0
1001 1032

ALM NO. 1032 BIT ADDRES: A0004.0
M code is unpermitted

          JOG
    
```

Page as the cursor locates at the alarm No.1032

2) CNC ALARM: It displays the numbers of CNC alarm, PLC alarm, and the current CNC alarm No.. It may display 24 CNC alarm No. together. The details for the respective alarm No. may be viewed by moving the cursor.




```

CNC ALARM                                00000 N00000
CNC ALM  2, PLC ALM  0 / PLC WARN  0
000  421

ESP WARN:000
CNC has received a emergency stop signal
, which caused a warning

ESP.                                     MDI
    
```

Page as the cursor locates at the alarm No.000



3) WARN LOG: Press  key to enter Alarm interface, then press it again to enter the WARN LOG page, which records the latest history alarm message including alarm date, alarm time, alarm No. and alarm content. 200 pieces warn log message can be viewed by  or  key.




```

WARN LOG          PAGE:1          00000 N00000
00002:10:55  301# 0 00000          N0000
Parameter switch has been opened
00000:37:58  302# 0 PlcValue.ENG N0000
Fail to have the system file
initialized
00000:26:53  301# 0 00000          N0000
Parameter switch has been opened
00000:26:24  302# 0 PlcValue.CHI N0000
Fail to have the system file
initialized

S0000 T00 H00
MDI
    
```

① Warn log order: the latest alarm log ranks at the head of the 1st page, following by others. If the alarm log exceeds 200, the last history log will be cleared.

② Alarm log manual clearing: under the 2 level authority, press  +  key, it may clear all the warn logs.

4) Alarm clearing: If multiple alarms are issued, only one alarm where the cursor locates could be cleared by pressing  key each time (In alarm interface, it clears all alarms and warnings by pressing  and  keys).

```

CNC ALARM                00000 N00000
CNC ALM  2, PLC ALM  0 / PLC WARN  0
000  421

ESP WARN:000
CNC has received a emergency stop signal
,which caused a warning

ESP.                      MDI
    
```

Current page





```

CNC ALARM                00000 N00000
CNC ALM  1, PLC ALM  0 / PLC WARN  0
421




CTR WARN:421
The X axis driver is not prepared

MDI
    
```



Page after pressing RESET key

5) 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 Graphic interface. Press  key repeatedly, it switches between Setting and Graphic 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 the parameter, program, auto sequence No. on-off state.

PARM SWT: when it is turned for ON, the parameters are allowed to be modified; it is turned for OFF, the parameters are unallowed to be modified.

PROG SWT: when it is turned for ON, the programs are allowed to be edited; it is turned for OFF, the programs are unallowed to be edited.

AUTO SEG: when it is turned for ON, the block No. is created automatically; it is turned for OFF, the block No. is not created automatically, but manually if needed.

```

SWITCH SETTING          00000 N00000

▶ PARM SWT:  *OFF  ON
  PROG SWT:  *OFF  ON
  AUTO SEG:  *OFF  ON

S0000 T00 H00
MDI
    
```

2) PARM OPERATION: In this page, the CNC data (such as bit parameter, data parameter, screw-pitch parameter, tool offset) can be backup and restored.

Backup PAR. (User): For CNC data backup by user (save)

Resume PAR. (User): For backup data restore by user (read)

Resume Default PAR.1(Test): For original parameter data reading of CNC test by user

Resume Default PAR. 2 (Step): For original parameter data reading of suited step drive by user.

Resume Default PAR. 3 (Servo): For original parameter data reading of suited servo drive by user.

```

PARM. OPERATION          00000 N00000
▶ Backup PAR. ( User )
Resume PAR. ( User )
Resume Default PAR.1(Test)
Resume Default PAR.2(Step)
Resume Default PAR.3(Servo)
PRESS[IN]+[P]TO CONFIRM(POWER ON AGAIN)

                                S0000 T00 H00
                                MDI
    
```

Page of 3rd, 4th, 5th level user

```

PARM. OPERATION          00000 N00000
▶ Backup PAR. (Machine Tool Corp.)
Resume PAR. (Machine Tool Corp.)
Resume Default PAR.1(Test)
Resume Default PAR.2(Step)
Resume Default PAR.3(Servo)
PRESS[IN]+[P]TO CONFIRM(POWER ON AGAIN)

                                S0000 T00 H00
                                MDI
    
```

Page of 2nd level user

3) PASSWORD SETTING: for user operation level display and setting

The password of GSK980MD is classified for 4 levels, by descending sequence, which are machine builder (2nd) level, equipment management (3rd) level, technician (4th) level, machining operation (5th) level.

Machine builder (2nd) level: the CNC bit parameter, data parameter, screw-pitch parameter, tool offset data, part program edit(including macro), PLC ladder editing and modification, ladder upload and download operations are allowed;

Equipment management (3rd) level: initial password 12345, the CNC bit parameter, data parameter, screw-pitch parameter, tool offset data, part program edit operations are allowed;

Technician (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, screw-pitch parameter operations are unallowed.

Operation level: no password. Only the machine panel operation is allowed, and the operations of part program edit and selection, the modification operations of CNC bit parameter, data parameter, screw-pitch parameter, tool offset data are unallowed.

```


PASSWORD SETTING          00000 N00000
      CURRENT LEVEL: 3
      SET LOWER LEVEL
▶ INPUT  PASSWORD:
      UPDATE  PASSWORD:


      Modify parameter and edit program

                                S0000 T00 H00
                                MDI
    
```

2. Setting page of G54~G59

Page location

Under the Setting interface, press  key twice, this page is displayed. (The graphic page if pressing

 key twice)

SET (G54~G59)				00000 N00000	
EXT	X	0.000	G54	X	0.000
ZERO	Y	0.000		Y	0.000
	Z	0.000		Z	0.000
G55	X	0.000	G56	X	0.000
	Y	0.000		Y	0.000
	Z	0.000		Z	0.000
POS (MACHINE)					
X	0.000	Y	0.000	Z	0.000
DATA				S0000	T00 H00
MDI					

SET (G54~G59)				00000 N00000	
G57	X	0.000	G58	X	0.000
	Y	0.000		Y	0.000
	Z	0.000		Z	0.000
G59	X	0.000	WORKP	X	0.000
	Y	0.000	MOVE	Y	0.000
	Z	0.000		Z	0.000
POS (MACHINE)					
X	0.000	Y	0.000	Z	0.000
DATA				S0000	T00 H00
MDI					

The zero of the coordinate system: workpiece coordinate system zero offset, G54, G55, G56, G57, G58, G59

Page operation


1). Moving of the cursor

The cursor moves at the data of each coordinate system axis. And the data where the cursor locates are backlighted.

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 are backlighted.

2). Absolute data input

After“data+ key”is keyed in by user, the data where the cursor locates is changed for 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.

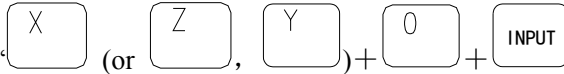
3). Relative data input



After “data+ key” is keyed in by user, the original data where the cursor locates is changed by the sum of “data” input by user pulsing that original data.

The validity judgement of user input data is the same as that of 980TD coordinate data input in MDI mode.

4). Auto measurement input



After “ (or ,) + + 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



keys.

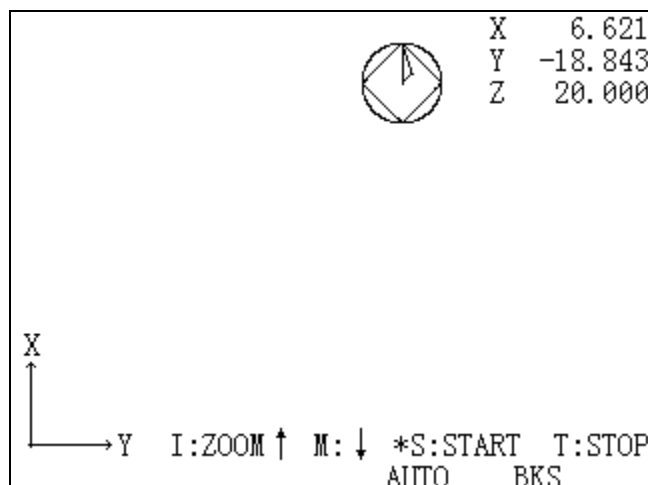
1) GRAPH SET page

In this page, the coordinate system, scaling and scope for graphic display can be selected.

GRAPH		00008 N00000
► COORD OPT	1	(XY0 YX1 ZX2 XYZ3
SCALE	50.000	YZ4 ZY5 XZ6 XZY7)
CENTER	-76.000	(X axis value)
CENTER	-36.000	(Y axis value)
CENTER	-8.000	(Z axis value)
X MAX.	44.000	
Y MAX.	64.000	
Z MAX.	106.000	
X MIN.	-196.000	
Y MIN.	-136.000	
Z MIN.	-122.000	
		S0000 T00 H00
		MDI

2) GRAPH TRACK page

In this page, it displays the path within the parameters range (referred by absolute coordinate) of GRAPH SET page.



1.3.6 BIT PARAMETER, DATA PARAMETER, SCREW-PITCH COMP interface



is a compound key, it enters BIT PARAMETER, DATA PARAMETER and SCREW-PITCH COMP interfaces by pressing this key repeatedly.

1 BIT PARAMETER interface



Press key, it enters BIT PARAMETER interface, there are 30 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 shown as following:

As is shown in this page, there are 2 parameter rows at the page bottom, 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				00008 N00000	
NO.	DATA	NO.	DATA		
_001	00000000	009	00000000		
_002	00100011	010	00011111		
003	00000000	011	00000000		
004	01000000	012	00011011		
005	00010001	013	10000011		
006	00000000	014	00000111		
007	00000000	164	10000000		
008	00000111	168	00000000		
BIT7:1/0:Reserved					
*** ** ACS HWL XRC *** **					
NO. 001				S0000 T00 H00	
				AUTO BKS	

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 shown as following:

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				00008 N00000	
NO.	DATA	NO.	DATA		
_015	1	023	7600		
_016	1	024	7600		
017	1	025	100		
018	1	026	100		
019	1	027	100		
020	1	028	200		
021	0	029	100		
022	7600	030	50		
Command multiplier for X ax.					
NO. 015				S0000 T00 H00	
				AUTO BKS	


3 SCREW-PITCH COMP interface

Press  key repeatedly, it enters SCREW-PITCH COMP interface, there are 256 screw-pitch parameters




which are displayed by 32 pages in this interface, and they can be viewed by pressing  or  key.



SCREW-PITCH COMP				00008 N00000
NO.	X	Y	Z	
000	0	0	0	
001	200	200	0	
002	200	200	0	
003	0	0	0	
004	1	1	0	
005	3	3	0	
006	0	0	0	
007	0	0	0	
Units: 0.001 (mm)				
NO. 000				S0000 T00 H00
				AUTO BKS

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




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 diaognsis No. bits where the cursor locates.

CNC DIAGNOSIS				00008 N00000
NO.	DATA	NO.	DATA	
000	00000000	008	00000111	
001	00000000	009	00000111	
002	00000000	010	00000000	
003	00000000	011	00000000	
004	00000000	012	00000000	
005	00000000	013	00000000	
006	00000000	014	00000000	
007	10000100	015	00000000	
BIT7:*TCP/low pressure detect (mach->PLC				
TCP DIQP XDEC BDT T04 T03 T02 T01				
NO. 000				S0000 T00 H00
				AUTO BKS

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.

PLC STATE				00008 N00000			
NO.	DATA	NO.	DATA				
X0000	00000000	X0008	00000000				
X0001	00000000	X0009	00000000				
X0002	00000000	X0010	00000000				
X0003	00000000	X0011	00000000				
X0004	00000000	X0012	00000000				
X0005	00000000	X0013	00000000				
X0006	00000000	X0014	00000000				
X0007	00000000	X0015	00000000				
Bit0: T03							
TCP DIQP ESP T05 XDEC BDT T04 T03							
NO.	X0000			S0000	T00	H00	
				AUTO	BKS		


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 VALUE				00008 N00000			
NO.	DATA	NO.	DATA				
DT000	0	DT008	0				
DT001	0	DT009	0				
DT002	0	DT010	0				
DT003	0	DT011	0				
DT004	0	DT012	0				
DT005	0	DT013	0				
DT006	0	DT014	0				
DT007	0	DT015	0				
保留							
NO.	DT000			S0000	T00	H00	
				AUTO	BKS		

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. As is shown in the following figure:

```









VERSION MESSAGE                00000 N00000

PRODUCT TYPE      : GSK980MD
SOFTWARE VER. NO. : V07.10.20
HARDWARE VER. NO. : 2.00.000--06.03.09

LADDER DESIGN CORP: GSK OF CHINA
LADDER VER. NO.   : V2.0-07.05.10
LADDER VERIFY CODE: CE80
NOTE: Standard Ladder of GSK980MD

                                S0000 T01 H00
                                JOG
    
```

1.3.8 LCD contrast adjustment



























It enters the RELATIVE POS interface by pressing  key (press  or  key if necessary), then press  or  or  key to make them to blink, and the LCD contrast decreases (darkening) each time by pressing  key, or increases (brightening) each time by pressing  key.






















```

RELATIVE POS
00000 N00000
X      0.000
Y      0.000
Z      0.000












PRG.F : 100    G CODE: G00 ,G94
ACT.F : 0      PART CNT: 0
FED OVRI: 150% CUT TIME: 0:00:00
RAP OVRI: 100%                S0000 T00 H00
                                MDI
    
```


















1.4 General Operation List

Classification	Function	Operation	Operation mode	Page	Password level	Program switch	Parameter switch	Remark
Clearing	Relative coordinate clearing of X axis	 , 		RELATIVE POS				Section 1.3.1 of the 2 nd part
	Relative coordinate clearing of Y axis	 , 		RELATIVE POS				
	Relative coordinate clearing of Z axis	 , 		RELATIVE POS				
	Parts clearing	 + 		RELATIVE POS or ABSOLUTE POS				
	Cut time clearing	 + 						
	X offset clearing	 , 		TOOL OFFSET	2 nd level, 3 rd level, 4 th level			Section 7.4.3 of the 2 nd part
	Z offset clearing	 , 		TOOL OFFSET	2 nd level, 3 rd level, 4 th level			Section 7.4.3 of the 2 nd part
Data setting	Bit parameter	Parameter value, 	MDI mode	BIT PARAMETER	2 nd level, 3 rd level		ON	Section 7.4.3 of the 2 nd part
	Data parameter	Parameter value, 	MDI mode	DATA PARAMETER	2 nd level, 3 rd level		ON	
	X screw-pitch parameter input	 , offset value, 	MDI mode	SCREW-PITCH COMP	2 nd level		ON	Section 10.1.3 of the 2 nd part
	Z screw-pitch parameter input	 , offset value, 	MDI mode	SCREW-PITCH COMP	2 nd level		ON	Section 10.1.3 of the 2 nd part
	Macro variable	Macro variable value, 		MACRO	2 nd level, 3 rd level, 4 th level			
	Incremental input of X axis offset	 , 		TOOL OFFSET	2 nd level, 3 rd level, 4 th level			Section 7.4.2 of the 2 nd part
	Incremental input of Z axis offset	 , 		TOOL OFFSET	2 nd level, 3 rd level, 4 th level			Section 7.4.2 of the 2 nd part
Search	Downward search from cursor current location	Character, 	Edit mode	PRG CONTENT	2 nd level, 3 rd level, 4 th level	ON		Section 6.1.3 of the 2 nd part

Classification	Function	Operation	Operation mode	Page	Password level	Program switch	Parameter switch	Remark
	Upward search from cursor current location	Character, 	Edit mode	PRG CONTENT	2 nd level, 3 rd level, 4 th level	ON		Section 6.1.3 of the 2 nd part
	Downward search from current program	 , 	Edit mode or Auto mode	PRG CONTENT, PRG LIST, PRG STATE	2 nd level, 3 rd level, 4 th level			Section 6.4.1 of the 2 nd part
	Upward search from current program	 , 			2 nd level, 3 rd level, 4 th level			Section 6.4.1 of the 2 nd part
	Search of a specified program	 , program name, 			2 nd level, 3 rd level, 4 th level			Section 6.4.2 of the 2 nd part
	Search of bit parameter, data parameter or screw-pitch parameter	 , parameter No., 		Corresponding data page				Section 10.1.3 of the 2 nd part
	PLC state, PLC data search	 , address No., 		PLC state, PLC data				
Deletion	Deletion of a character cursor locating at		Edit mode	PRG CONTENT	2 nd level, 3 rd level, 4 th level	ON		Section 6.1.6 of the 2 nd part
			Edit mode	PRG CONTENT	2 nd level, 3 rd level, 4 th level	ON		
	Deletion of a single block	Cursor homing, 	Edit mode	PRG CONTENT	2 nd level, 3 rd level, 4 th level	ON		With block No., Section 6.1.7 of the 2 nd part
	Deletion of blocks	  , sequence No., 	Edit mode	PRG CONTENT	2 nd level, 3 rd level, 4 th level	ON		Section 6.1.8 of the 2 nd part
	Segment deletion	 , Character, 	Edit mode	PRG CONTENT	2 nd level, 3 rd level, 4 th level	ON		Section 6.1.9 of the 2 nd part
	Deletion of a program	 , program name, 	Edit mode	PRG CONTENT	2 nd level, 3 rd level, 4 th level	ON		Section 6.3.1 of the 2 nd part

Chapter 1 Operation Mode And Display

Classification	Function	Operation	Operation mode	Page	Password level	Program switch	Parameter switch	Remark
	Deletion of all programs	0,  999, 	Edit mode	PRG CONTENT	2 nd level, 3 rd level, 4 th level	ON		Section 6.3.2 of the 2 nd part
Rename	Rename of a program	0, 	Edit mode	PRG CONTENT	2 nd level, 3 rd level, 4 th level	ON		2 nd level authority needed if block No. more than or equal to 9000. Section 6.6 of the 2 nd part
Copy	Copy of a program	0, 	Edit mode	PRG CONTENT	2 nd level, 3 rd level, 4 th level	ON		2 nd level authority needed if block No. more than or equal to 9000. Section 6.7 of the 2 nd part
CNC→CNC (Sending)	Tool offset		Edit mode	TOOL OFFSET	2 nd level, 3 rd level		ON	Section 11.6 of the 2 nd part
	Bit parameter		Edit mode	BIT PARAMETER	2 nd level, 3 rd level		ON	
	Data parameter		Edit mode	DATA PARAMETER	2 nd level, 3 rd level		ON	
	Screw-pitch parameter		Edit mode	SCREW-PITCH H COMP	2 nd level		ON	
	Sending of a program	0, program name, 	Edit mode	PRG CONTENT	2 nd level, 3 rd level, 4 th level	ON		
	Sending of all programs	0,  999, 	Edit mode	PRG CONTENT	2 nd level, 3 rd level, 4 th level	ON		
CNC→CNC (Receiving)	Tool offset		Edit mode		2 nd level, 3 rd level, 4 th level		ON	Section 11.6 of the 2 nd part
	Bit parameter		Edit mode		2 nd level, 3 rd level		ON	
	Data parameter		Edit mode		2 nd level, 3 rd level		ON	
	Screw-pitch parameter		Edit mode		2 nd level		ON	

Classification	Function	Operation	Operation mode	Page	Password level	Program switch	Parameter switch	Remark
	Part program		Edit mode		2 nd level, 3 rd level, 4 th level	ON		
CNC→PC (Upload)	Tool offset		Edit mode	TOOL OFFSET	2 nd level, 3 rd level, 4 th level		ON	Section 11.5.3 of the 2 nd part
	Bit parameter		Edit mode	BIT PARAMETER	2 nd level, 3 rd level, 4 th level		ON	
	Data parameter		Edit mode	DATA PARAMETER	2 nd level, 3 rd level		ON	Section 11.5.4 of the 2 nd part
	Screw-pitch parameter		Edit mode	SCREW-PITCH COMP	2 nd level		ON	
	Sending of a program	 , program name, 	Edit mode	PRG CONTENT	2 nd level, 3 rd level, 4 th level	ON		Section 11.5.1 of the 2 nd part
	Sending of all programs	 ,  , 	Edit mode		2 nd level, 3 rd level, 4 th level	ON		Section 11.5.2 of the 2 nd part
PC→CNC (Download)	Tool offset		Edit mode		2 nd level, 3 rd level, 4 th level		ON	Section 11.4.2 of the 2 nd part
	Bit parameter		Edit mode		2 nd level, 3 rd level		ON	Section 11.4.3 of the 2 nd part
	Data parameter		Edit mode		2 nd level, 3 rd level		ON	
	Screw-pitch parameter		Edit mode		2 nd level		ON	Section 11.4.3 of the 2 nd part, 2 nd level needed
	Part program		Edit mode		2 nd level, 3 rd level, 4 th level	ON		Section 11.4.1 of the 2 nd part, 2 nd level needed if block No. more than or equal to 9000
LCD contrast	Contrast increasing(brightening)	( , ) or ( , )	Edit mode	RELATIVE POS				Section 1.3.8 of the 2 nd part
	Contrast decreasing(darkening)	( , ) or ( , )	Edit mode	RELATIVE POS				

Chapter 1 Operation Mode And Display

Classification	Function	Operation	Operation mode	Page	Password level	Program switch	Parameter switch	Remark
Switch setting	Parameter switch ON			SWITCH SETTING	2 nd level, 3 rd level			Section 10.1.1 of the 2 nd part
	Program switch ON			SWITCH SETTING	2 nd level, 3 rd level, 4 th level			
	Auto sequence No. switch ON			SWITCH SETTING				
	Parameter switch OFF			SWITCH SETTING	2 nd level, 3 rd level			
	Program switch OFF			SWITCH SETTING	2 nd level, 3 rd level, 4 th level			
	Auto sequence No. switch OFF			SWITCH SETTING				

Explanation: In the Operation column, “,” sign between two adjacent keys means the operation is performed by sequence; “+” sign between two adjacent keys means the operations are performed simultaneously.

Example: , means to press key first, then to press key; + means to press two keys simultaneously.

CHAPTER 2 POWER ON OR OFF AND PROTECTION

2.1 System Power On

Before this GSK980MD power 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 GSK980MD is powered on:



The current position (RELATIVE POS) page is displayed after system auto detection and initiation are finished.

```

RELATIVE POS
00000  N00000
X      0.000
Y      0.000
Z      0.000
PRG.F : 100    G CODE: G00 ,G94
ACT.F : 0      PART CNT: 0
FED OVRI: 150%  CUT TIME: 0:00:00
RAP OVRI: 100%      S0000 T00 H00
                     MDI
  
```

2.2 System Power Off

Before power is off, ensure that:

- 1 The X, Y, Z axis of the CNC is at halt;
- 2 Miscellaneous functions (spindle, pump etc.) are off;
- 3 Cut off CNC power prior to machine power cutting off.

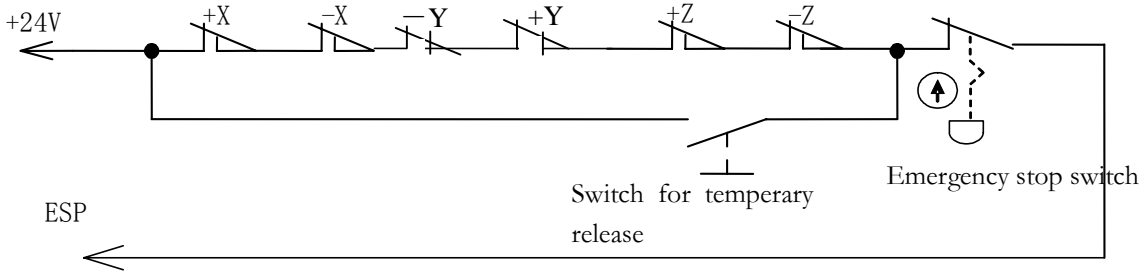
Note: Please see the machine builder's manual for the machine power cut-off operation.

2.3 Overtravel Protection

Overtravel protection should be employed to prevent the damage to the machine due to the overtravel of the X, Y, or Z axis.

2.3.1 Hardware overtravel protection

The stroke switches are fixed at the positive and negative maximum travel of the machine X, Y, Z axis respectively, they are connected by the following figure. And the BIT3 (MESP) of bit parameter No.172 must be set to 0. If the overtravel occurs, the stroke switch acts to make the machine to stop, and the emergency alarm is issued.



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 BIT4 of bit parameter No.172 is set to 0, the software limit is valid.


The software travel strokes are set by data parameter NO.045,NO.046,NO.047,NO.048, NO.049,NO.050, they refer to machine coordinate. As following figure shows, X, Y, Z are the machine coordinate system axes;No.045,NO.048 are for X axis positive and negative stroke, No.046,NO.049 are for Y axis positive and negative stroke, No.047,NO.050 are for Z axis positive and negative stroke.


If the machine position (coordinate) exceeds the setting range, overtravel alarm will be issued. The steps to eliminate this alarm is: press RESET key to clear the alarm display, 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 etc.So this GSK980MD should be stopped immediately for these incidents. This section mainly describes the resolutions that this GSK980MD are capable of under the emergency situation. Please see the relative explanation on these resolutions under the emergency by machine builder.

2.4.1 Reset

Press  key to reset this GSK980MD system if there are abnormal output and axis action in it:

- 1 All axes movement stops;
- 2 M, S function output is invalid(which can be set by parameter whether automatically cut off signals such as spindle CCW/CW, lubrication, cooling by pressing  key, defined by PLC ladder);
- 3 Auto run ends, modal function and state is 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. All the outputs such as the spindle running, coolant are cut off. 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: Repperform 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 BIT3 (MESP) of the bit parameter No.172 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 to pause. But in threading, 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. But 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.

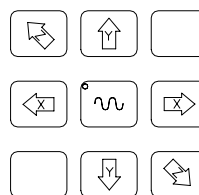
The keys functions of this 980MD machine panel are defined by ladders, please refer to the materials by the respective machine builder for the function significance.


Please note that the following function introduction is described based on the 980MD standard ladder!



3.1 Coordinate axis moving

In Manual mode, the coordinate axis can be moved manually for feeding and rapid traverse.

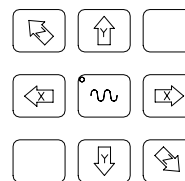
3.1.1 Manual feed







Press and hold 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; also the 3 axes direction keys of X, Y, Z axis can be held at a time to make the 3 axes to move simultaneously.

In Manual mode, press  key to make the indicator  to light up, and the rapid traverse is valid, it enters the Manual rapid traverse mode.

3.1.2 Manual rapid traverse



Press and hold feed axis and  key in the 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 the axis direction key, and the axis stops moving if releasing the key; also the 3 axes direction keys of X, Y, Z axis can be held at a time to make the 3 axes to move simultaneously.




In Manual rapid mode, press  key to make the indicator  to 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 BIT0 of the bit parameter No.012.

Note 2: In Edit/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 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, it can press  or  key in  (also by , ,  key with the respective override F0, 50%, 100%) to modify the Manual rapid override, and there are 4 gears of F0, 20%, 50%, 100% for the override. (F0 set by data parameter No.032)

3.1.5 Relative coordinate clearing





1) Press  key to enter Position interface, then press  or  key to select the RELATIVE POS page;


```

RELATIVE POS
00000 N00000
X      3.024
Y      3.024
Z      4.704
JOG.F : 1260 G CODE: G00 ,G94
ACT.F : 0 PART CNT: 0
FED OVRI: 150% CUT TIME: 0:00:00
RAP OVRI: 100% S0000 T00 H00
JOG

```

- 2) Press  key to make the “X” in the page to blink, then press  key;

```

RELATIVE POS
00000 N00000
X      0.000
Y      3.024
Z      4.704
JOG.F : 1260 G CODE: G00 ,G94
ACT.F : 0 PART CNT: 0
FED OVRI: 150% CUT TIME: 0:00:00
RAP OVRI: 100% S0000 T00 H00
JOG

```

- 3) The clearing operations of Y, Z coordinate are the same as above.

3.2 Other Manual operations

Note: The following operations are also valid in Machine zero, MPG/Step etc. mode.

3.2.1 Spindle CCW, CW, stop control



: In Manual mode, the spindle rotates forward if pressing this key;




: In Manual mode, the spindle stops if pressing this key;



: In Manual mode, the spindle rotates backward if pressing this key;

3.2.2 Spindle Jog

Press and hold  key, the spindle rotates forward, 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 overridden.



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 980MD machine panel are defined by ladders, please refer to the materials by the respective machine builder for the function significance.

Please note that the following function introduction is described based on the 980MD standard PLC program!




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:

```

RELATIVE POS
00008  N00000
X      0.000
Y      0.000
Z     -37.842
STEP INC: 0.001   G CODE: G00 ,G94
ACT.F :    0     PART CNT:    6
FED OVRI: 150%   CUT TIME:  0:02:01
RAP OVRI: 100%           S0000 T00 H00
                        STEP
  
```

4.1.1 Increment selection

Press ,  or  key to select the move increment, the increment will be shown in the page.

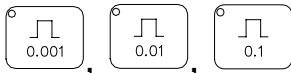
When the BIT1 (SINC) of the bit parameter No.173 is 1, step  is invalid; when the BIT1 is 0, , ,  are all valid. For example, to press  key, the page is shown as following:

```

RELATIVE POS
00008 N00000
X      0.000
Y      0.000
Z     -37.842



STEP INC: 0.001   G CODE: G00 ,G94
ACT.F :    0     PART CNT:    6
FED OVRI: 100%   CUT TIME:  0:02:01
RAP OVRI: 100%           S0000 T00 H00
                        STEP
    
```




Note: In the modes other than Edit and MPG/Step, the Rapid override can be changed by pressing




key with the corresponding override F0, 50%, 100%.


4.1.2 Moving direction selection

Press  or  key once, it can move the X axis negatively or positively by a step increment; press

 or  key once, it can move the Y axis negatively or positively by a step increment; press  or

 key once, it can move the Z axis negatively or positively by a step increment.

4.2 Handwheel Feed

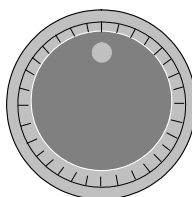
Set the BIT3 of the bit parameter No.001 to 1, and press  key to enter the MPG mode, it displays as following:

```

RELATIVE POS
00008 N00000
X      0.000
Y      0.000
Z     -37.842




HNDL INC: 0.001   G CODE: G00 ,G94
ACT.F :    0     PART CNT:    6
FED OVRI: 100%   CUT TIME:  0:02:01
RAP OVRI: 100%           S0000 T00 H00
                        HNDL
    
```




The handwheel figure is as following:



The handwheel figure

4.2.1 Increment selection

Press ,  or  key to select the move increment, the increment will be shown in the page.

When the BIT1 (SINC) of the bit parameter No.173 is 1, step  is invalid; when BIT1 is 0, , ,


 are all valid. For example, to press  key, the page is shown as following:

```

RELATIVE POS
00008  N00000
X      0.000
Y      0.000
Z     -37.842
HNDL INC: 0.001  G CODE: G00 ,G94
ACT.F :    0    PART CNT:    6
FED OVRI: 150%  CUT TIME:  0:02:01
RAP OVRI: 100%                S0000 T00 H00
                                HNDL
  
```

4.2.2 Moving axis and direction selection

In MPG mode, press  or  axis, the corresponding axis will be selected. For example, to press

 key, the page is shown as following:

```

RELATIVE POS
00008  N00000
X      0.000
Y      0.000
Z     -37.842
HNDL INC: 0.001  G CODE: G00 ,G94
ACT.F :    0    PART CNT:    6
FED OVRI: 150%  CUT TIME:  0:02:01
RAP OVRI: 100%                S0000 T00 H00
                                HNDL X-AXIS
  
```

The handwheel feed direction is defined by its rotation direction. Generally, the handwheel CW is for positive feed, and CCW 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.

4.2.3 Explanation items

1.The correspondence of the handwheel scale to the machine moving amount is as following table:

	Moving amount of each handwheel scale		
Handwheel increment	0.001	0.01	0.1
Specified coordinate value	0.001mm	0.01mm	0.1mm

2.The rotation speed of the handwheel should be less than 5 r/sec, if it is over that, the scale may be not coincide with the moving amount;

3.The ,  and  keys are only valid 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 980MD machine panel are defined by ladders, 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 980MD standard PLC programs!





5.1 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  key to enter MDI mode;

2 Press  key ( or  key if needed) to enter PRG STATE page:

PRG STATE		00000 N00000	
(MDI)		(MODAL)	
X	M	G00	X
Y	S	G17	Y
Z	T	G90	Z
U	H	G94	V
V	D	G98	W
W	L	G54	R
I		G21	F 100
J		G40	M
K		G49	S 00
R			P 0
F			Q
P			
Q			
		S0000 T00 H00	
		MDI	

3 Key in address key , numerical key ,  and  key by sequence, the page is shown as following:


4 Key in address key , numerical key , ,  and  key by sequence;

5 Key in address key , numerical key ,  and  key by sequence;

The page is shown as following after the above operations are completed:

PRG STATE				00008 N00000			
(MDI)				(MODAL)			
G92	X	50.000	M	G00	X		
	Y		S	G17	Y		
	Z	100.000	T	G90	Z		
	U		H	G94	V		
	V		D	G98	W		
	W		L	G54	R		
	I			G21	F	100	
	J			G40	M		
	K			G49	S	00	
	R				P	0	
	F				Q		
	P						
	Q						
				S0000 T00 H00			
				MDI			

5.2 Words Execution

After the words are input, these MDI words are executed after the  key is pressed. During the execution,









and Emergency Stop button may be pressed to terminate the MDI words execution.

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 10 of this part.

5.4 Data Modification

In the PRG STATE page of MDI mode, for the input data to be executed, if there is an error in input words, it may press  key to clear all the input, then reinput the correct ones; or reinput the correct part to replace the false one. As for the false input data“X50” in Section 5.1 of this chapter, it may press the address key , numerical key , ,  and  key to replace it . And the page is shown as following after the operation:

PRG STATE				O0008 N00000			
(MDI)				(MODAL)			
G92	X	100.000	M	G00	X		
	Y		S	G17	Y		
	Z	100.000	T	G90	Z		
	U		H	G94	V		
	V		D	G98	W		
	W		L	G54	R		
	I			G21	F	100	
	J			G40	M		
	K			G49	S	00	
	R				P	0	
	F				Q		
	P						
	Q						
				S0000 T00 H00			
				MDI			

5.5 OUT Key Start

When the BIT2 of the system parameter No.004 is set to 1, the current words input may be executed by pressing



key in MDI mode.


CHAPTER 6 PROGRAM EDIT AND MANAGEMENT

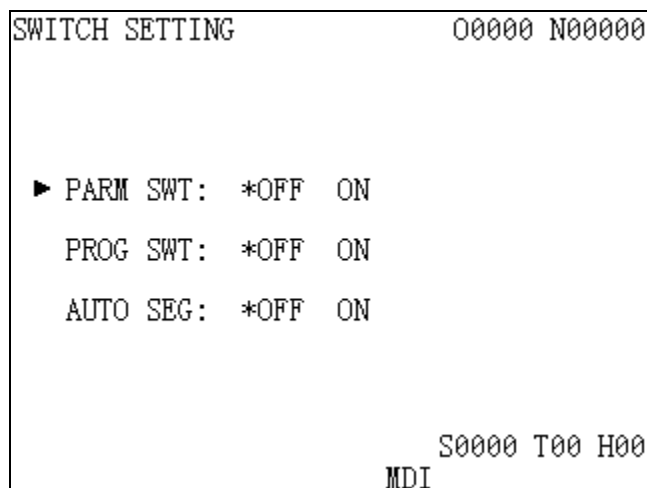
In Edit mode, the program can be created, selected, modified, copied and deleted, and the bidirectional communication between CNC and CNC, or CNC and PC can also be achieved. To prevent the program to be modified or deleted accidentally, a program switch is set for this GSK980MD system. And it must be 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 the program switch be opened for program editing.

6.1 Program Creation


6.1.1 Creation of the block number



In the program, the block number can be added or not, the program is executed by the block numbered sequence (except the calling). When the “AUTO SEG” switch in setting page is OFF, CNC doesn’t generate the block number automatically, but the blocks may be numbered manually.

When the “AUTO SEG” switch in setting page is ON, CNC generates the block number automatically, it automatically generates the next block number by pressing  key in editing. The block number increment is set by the CNC data parameter No.042.



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 ROW2 COL1 00000 N00000
00000;
G0 G54 G90 X0 Y0 Z0;
X10 Y10;
X-10 Y-10;
M99;
%

S0000 T00 H00
EDIT
```

3 Key in address key , numerical key , , and key by sequence (example by program O0001 creation);

```
PRG CONTENT ROW2 COL1 00000 N00000
00000 (00000);
G0 G54 G90 X0 Y0 Z0;
X10 Y10;
X-10 Y-10;
M99;
%

00001 S0000 T00 H00
EDIT
```

4 Press key to setup the new program;

```
PRG CONTENT ROW2 COL1 00001 N00000
00001 (00001);
;
%



S0000 T00 H00
EDIT
```



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 key to terminate it.



6 Other blocks input may be finished by step 5 above.


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;


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;

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;

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;


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;


Press  key to page upward, the cursor moves to the 1st line and 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;


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 is as following:

1) Press  key to enter Edit mode;



2) Press  key to enter the PRG CONTENT page;


3) Press  key to enter Search mode, input the characters to be searched with max. 10 bytes, the character over 10 bit will cover the 10th bit. e.g. to move the cursor to G2, it displays as following:

```
PRG CONTENT ROW9 COL1 00008 N00000
00008 (CNC PROGRAM 2005020);
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;
FINDG2_ S0000 T00 H00
EDIT
```

4) Press  key ( or  by the location relation of the character to be searched and the character where the cursor locates), it displays as following:

```
PRG CONTENT ROW6 COL1 00008 N00000
00008 (CNC PROGRAM 2005020);
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;
FINDG2_ S0000 T00 H00
EDIT
```


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 cue 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 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;

- 2) Press  key to enter the Insert state (the underline is cursor), the page is as following:

```
PRG CONTENT ROW6 COL1 00008 N00000
00008 (CNC PROGRAM 2005020);
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;
FINDG2_ S0000 T00 H00
EDIT
```

- 3) Input the character to be inserted(as G98 code before G2 in the above figure, input ,

, , )

```
PRG CONTENT ROW6 COL5 00008 N00000
00008 (CNC PROGRAM 2005020);
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;
S0000 T00 H00
EDIT
```

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 the Insert mode, if the previous bit before the cursor is a decimal point and the cursor is not located at the line end, input an address word, the "0" will be added automatically following the decimal point;



Note 3 In the Insert mode, if the previous bit before the cursor is a decimal point and the cursor is not located at the line end, the "0" will be added automatically following the

decimal point by pressing  key.

6.1.5 Deletion of the character

Steps:

1) Select the PRG CONTENT page in Edit mode;

2) Press  key to delete the character before the cursor; press  key to delete the character where the cursor locates.


6.1.6 Modification of the character

There are two methods for the character modification:

Modification by insertion: first delete the character modified by the methods in Section 6.1.5.

Direct modification:




1) Select the PRG CONTENT page in Edit mode

1) Press  key to enter the modification state(the cursor is a backlight rectangle), the page is as following:

```
PRG CONTENT ROW5 COL4 00008 N00000
00008 (CNC PROGRAM 2005020);
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;

S0000 T00 H00
EDIT
```

3) Key in the characters modified (for example, to modify the X20 in above page for U89, key in

, , ) , the page is as following:

```
PRG CONTENT ROW5 COL7 00008 N00000
00008 (CNC PROGRAM 2005020);
G40 G49 G80;
G0 G90 G54 X0 Y0 Z0;
Z50;
G1 U89Z20 F1500;
G98 G2 I-20;
G3 I-20;
G4 X5;
G1 X0 Y20 Z0 F1000;
X-20 Y0;

S0000 T00 H00
EDIT
```


Note 1 In modification state, the current character where the cursor locates is altered for the input one, and the cursor moves a bit forward;

Note 2 In modification state, if the cursor is located at the “;” sign, the input character will substitute this sign, and the next block will shift upward a line. As following figure shows, key in “0”, the page is as following:

```

PRG CONTENT ROW5 COL18 00008 N00000
00008 (CNC PROGRAM 2005020);
G40 G49 G80;
G0 G90 G54 X0 Y0 Z0;
Z50;
G1 U89 Z20 F15000; G98 G2 I-20;
G3 I-20;
G4 X5;
G1 X0 Y20 Z0 F1000;
X-20 Y0;
X0 Y-20 Z-10;

S0000 T00 H00
EDIT


```

6.1.7 Deletion of a single block

This function is only applied to the block which has a block No. and the block No. is headed the line or there is only space before the block No..

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), then press  key.

Note If the block has no block No., key in “N” at the head of the block, and move the cursor to

“N”, then press  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. (retrieving downward), and the specified block must have a block No..

```

PRG CONTENT ROW3 COL1 00008 N00000
00008 (CNC PROGRAM 2005020);
G40 G49 G80;
G0 G90 G54 X0 Y0 Z0;
Z50;
G1 U89 Z20 F15000;
N10 G98 G2 I-20;
G3 I-20;
G4 X5;
G1 X0 Y20 Z0 F1000;
X-20 Y0;

S0000 T00 H00
EDIT


```

Steps:

- 1) Select the PRG CONTENT page in Edit mode;

- 2) Press  key to enter the FIND state, and key in the block No.;

```
PRG CONTENT ROW3 COL1 00008 N00000
00008 (CNC PROGRAM 2005020);
G40 G49 G80;
G0 G90 G54 X0 Y0 Z0;
Z50;
G1 U89 Z20 F15000;
N10 G98 G2 I-20;
G3 I-20;
G4 X5;
G1 X0 Y20 Z0 F1000;
X-20 Y0;
FINDN10_ S0000 T00 H00
EDIT
```

- 3) Press  key, it displays as following:

```
PRG CONTENT ROW3 COL1 00008 N00000
00008 (CNC PROGRAM 2005020);
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;
S0000 T00 H00
EDIT
```


6.1.9 Segment deletion

It deletes the content downward from the current character where the cursor locates to the word specified.


```
PRG CONTENT ROW3 COL4 00008 N00000
00008 (CNC PROGRAM 2005020);
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;
S0000 T00 H00
EDIT
```

Steps:

- 1) Select the PRG CONTENT page in Edit mode;

- 2) Press  key to enter the FIND state, and key in the characters;

```
PRG CONTENT ROW3 COL4 00008 N00000
00008 (CNC PROGRAM 2005020);
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;
FINDF1000_ S0000 T00 H00
EDIT
```

- 3) Press  key, it displays as following:

```
PRG CONTENT ROW3 COL4 00008 N00000
00008 (CNC PROGRAM 2005020);
G40 G49 G80;
G3 _
X-20 Y0;
X0 Y-20 Z-10;
X20 Y0 Z-20;
X5 Y5 Z-50;
M99;
%
S0000 T00 H00
EDIT
```

Note 1 If the specified character is not found or the specified character is located before the current cursor, the cue 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

6.2.1 Setup of the program annotation

Steps:

- 1) Select the PRG CONTENT page in Edit mode;

- 2) Press  key, it displays as following:

```

PRG CONTENT ROW3 COL4 00008 N00000
00008 (CNC PROGRAM 2005020);
G40 G49 G80;
G3 ;
X-20 Y0;
X0 Y-20 Z-10;
X20 Y0 Z-20;
X5 Y5 Z-50;
M99;
%

FIND S0000 T00 H00
EDIT
    
```

3) Key in program annotation characters in the cue line (20 characters at most, parenthesis excluded), it displays as following:

```

PRG CONTENT ROW3 COL4 00008 N00000
00008 (CNC PROGRAM 2005020);
G40 G49 G80;
G3 ;
X-20 Y0;
X0 Y-20 Z-10;
X20 Y0 Z-20;
X5 Y5 Z-50;
M99;
%

FINDCNC PRO 20060312_ S0000 T00 H00
EDIT
    
```

INPUT

4) Press key to finish the annotation, it displays as following:

```

PRG CONTENT ROW3 COL4 00008 N00000
00008 (CNC PRO 20060312);
G40 G49 G80;
G3 ;
X-20 Y0;
X0 Y-20 Z-10;
X20 Y0 Z-20;
X5 Y5 Z-50;
M99;
%

S0000 T00 H00
EDIT
    
```

Note 1 If the annotation is not added after the program creation, CNC system defaults that the program name is the annotation.

Note 2 The annotation added to the CNC could only be English, but this CNC system supports the Chinese annotation(except the Chinese decimal point).

6.2.2 Modification of the program annotation






Its operation steps are the same as the setup of the program annotation in Section 6.2.1 of this part.

6.3 Deletion of the Program

6.3.1 Deletion a single program

Steps:

- 1) Select the PRG CONTENT page in Edit mode;






- 2) Key in address key , numerical key , , ,  by sequence(by program O0001);

- 3) Press  key, program O0001 will be deleted.

6.3.2 Deletion of all programs

Steps:

- 1) Select the PRG CONTENT page in Edit mode;

- 2) Key in address key , symbol key , numerical key , ,  by sequence;


- 3) Press  key, all the programs will be deleted.

6.4 Selection of the Program



When there are multiple programs in CNC system, a program can be selected by the following 3 methods:

6.4.1 Search method

- 1) Select Edit or Auto mode;

- 2) Press  key to enter the PRG CONTENT page;

- 3) Press address key  and key in the program No.;


- 4) Press  or  key, the retrieved program will be displayed. If the program does not exist, an alarm will be issued.


Note In step 4, if the program does not exist, a new program will be created by CNC system after



 key is pressed.

6.4.2 Scanning method

1) Select Edit or Auto mode;

2) Press  key to enter the PRG CONTENT page;


3) Press address key ;

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) Select Auto mode (must be in non-running state);





2) Press  key to enter the PRG LIST page;


```

PRG LIST                                00011 N00000
VERSION NO. :GSK980MD                   V07.10.20
PART-PRG NO. :   24
MEMORY SIZE: 20.0 MB; USED: 288 KB
PROGRAM LIST:
00000 00001 00002 00003 00004 00006
00008 00010 00011 00020 00022 00032
00041 00042 00088 00099 00100 00111
00232 00444 01114 04142 07777 07998

PRG SIZE:  61B   NOTE:00000

                                S0000 T00 H00
                                AUTO   BKS
    
```

3) Press , , ,  key to move the cursor to the program name to be selected ("PRG SIZE" "NOTE" content changed as the cursor moves);

4) Press  key.

```

PRG LIST                                00008 N00000
VERSION NO.:GSK980MD                   V07.10.20
PART-PRG NO.: 24
MEMORY SIZE: 20.0 MB; USED: 288 KB
PROGRAM LIST:
00000 00001 00002 00003 00004 00006
00008 00010 00011 00020 00022 00032
00041 00042 00088 00099 00100 00111
00232 00444 01114 04142 07777 07998

PRG SIZE: 733B    NOTE:00011
                                S0000 T00 H00
                                AUTO    BKS

```



```


PRG LIST                                00011 N00000
VERSION NO.:GSK980MD                   V07.10.20
PART-PRG NO.: 24
MEMORY SIZE: 20.0 MB; USED: 288 KB
PROGRAM LIST:
00000 00001 00002 00003 00004 00006
00008 00010 00011 00020 00022 00032
00041 00042 00088 00099 00100 00111
00232 00444 01114 04142 07777 07998

PRG SIZE: 733B    NOTE:00011
                                S0000 T00 H00
                                AUTO    BKS

```


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, 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;


4) Press  key.

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 Management

6.8.1 Program list

In non-Edit mode, press  key to enter the PRG LIST page. In this page, it lists the program names saved in CNC system, and it can display max. 36 names in a page, if the programs saved exceed 36,

it may press  key to display the names in the next page.

```

PRG LIST                                00011 N00000
VERSION NO. :GSK980MD                  V07.10.20
PART-PRG NO. : 24
MEMORY SIZE: 20.0 MB; USED: 288 KB
PROGRAM LIST:
00000 00001 00002 00003 00004 00006
00008 00010 00011 00020 00022 00032
00041 00042 00088 00099 00100 00111
00232 00444 01114 04142 07777 07998

PRG SIZE: 733B    NOTE:00011
                                S0000 T00 H00
                                AUTO    BKS
    
```

6.8.2 Software version

It shows the current CNC software version.

6.8.3 Part program number

It shows the total numbers of the part programs (max. 384) that can be saved in CNC system and the current part programs that have been saved currently.

6.8.4 The memory capacity and the capacity used

It shows the total capacity (22MB) of the CNC and the current capacity that has been used.

CHAPTER 7 AUTO OPERATION

Note !

The keys functions of this 980MD machine panel are defined by ladders, please refer to the respective materials by the machine builder for these functions significance.


Please note that the following functions are described based on the 980MD standard ladder!

7.1 Auto 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 CONTENT 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  key to enter the PRG LIST page;

c) Press , ,  or  key to move the cursor to the name of the program to be selected;

d) Press  key.


7.1.2 Program start

- 1、 Press  key to select the Auto mode;
- 2、 Press  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  key, make a check whether the cursor is located at the block to be executed.

7.1.3 Stop of the auto run

● Stop by command (M00)

After 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 slows down 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 Mode switching

When the Auto mode is switched to the Machine zero, MPG/Step, Manual, Program zero mode, 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.

Note 4 Only the BIT3(ESP) of the bit parameter No.172 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, then press  or  key to select the PRG CONTENT page:
- 2、 Move the cursor to the block to be executed (for example, move the cursor to the 4th line head if it executes from the 4th line);

```
PRG CONTENT ROW5 COL1 00008 N00000
00008 (CNC PROGRAM 2005020);
G40 G49 G80;
G0 G90 G54 X0 Y0 Z0;
Z50;
G1 U89 Z20 F1500;
G98 G2 I-20;
G3 I-20;
G4 X5;
G1 X0 Y20 Z0 F1000;
X-20 Y0;

S0000 T00 H00
EDIT
```




- 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, only the corresponding modal function is executed, could next step be continued.


- 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



FEEDRATE
OVERRIDE

It can realize the 4-level real time rapid override adjustment by pressing the  or  key in .

Press the  key each time, the rapid override ascends a gear level till 100%;

Press the  key each time, the rapid override decends a gear level till F0.

Note 1 The max. rapid traverse speeds of X, Y, Z axis are set by bit parameter No.022, No.023, No.024 respectively;

X axis actual rapid traverse feedrate = value set by parameter No.022×rapid override

Y axis actual rapid traverse feedrate = value set by parameter No.023×rapid override

Z axis actual rapid traverse feedrate = value set by parameter No.024×rapid override




Note 2 When the rapid override is F0, the rapid traverse feedrate is set by bit parameter No.032.


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.



SPINDE
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 gear level till 120%;

Press the  key each time, the rapid override decends a gear level till 50%.

Note 1 The actual output analog voltage=analog voltage by parameter×spindle override

Example: When the bit parameter No.037 is set for 9999, No.021 for 645, to execute S9999 command to select the spindle override 70%, the actual output analog voltage≈10×70%=7V

Note 2 The variation of the spindle is shown under the screen left bottom, the shift between rapid

override and spindle override is via the  key.

```

RELATIVE POS
00008  N00000
X      0.000
Y      0.000
Z     -37.842
JOG.F : 1260   G CODE: G00 ,G94
ACT.F :    0   PART CNT:    6
FED OVRI: 150% CUT TIME: 0:02:01
RAP OVRI: 100% S0000 T00 H00
REF

```



```

RELATIVE POS
00008  N00000
X      0.000
Y      0.000
Z     -37.842
PRG.F : 100   G CODE: G00 ,G94
ACT.F :    0   PART CNT:    6
FED OVRI: 150% CUT TIME: 0:02:01
SPI OVRI: 100% S0000 T00 H00
MDI

```

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 run 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 see it for details in the DNC communication software.


7.3 State on Running

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 valid single block function is as following:

Press the  key to make the single block indicator  in State area to light up, it means that the single block function is 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 stop is valid 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 valid dry run function is as following:

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 program in dry run mode is invalid, the machine runs by a speed in the following table:



	Program command	
	Rapid traverse	Cutting feed
Rapid traverse switch ON	Rapid traverse	Max. manual feedrate
Rapid traverse switch OFF	Manual feedrate or rapid traverse (see note)	Manual feedrate

Note 1 The rapid traverse by manual feedrate or rapid feedrate is set by the BIT6 of the bit parameter No.004.

Note 2 The shift of rapid switch in Dry run mode doesn't affect the speed of the current block being executed, but that of the next block.

7.3.3 Machine lock

In Auto mode, the valid machine lock function is as following:

Press the  key to make the machine lock indicator  in State area to light up, it means that the machine lock function is selected.



While in the machine lock mode:

1 the machine carriage doesn't move, the "MACHINE" in the INTEGRATED POS page of the POSITION interface doesn't 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 valid MST lock function is as following:



Press the  key to make the MST lock indicator  in State area to light up, it means that the MST lock function is selected. And the carriage move is not performed by M, S, T commands.

Note When the MST lock is valid, the execution of M00, M30, M98,M99 command is not affected by it.

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 valid block skip function is as following:

Press the  key to make the block skip indicator  in State area to light up, it means that the block skip function is selected.

Note: While the block skip function is invalid, the blocks headed with “/” signs are executed normally in Auto mode.


CHAPTER 8 MACHINE ZERO OPERATION

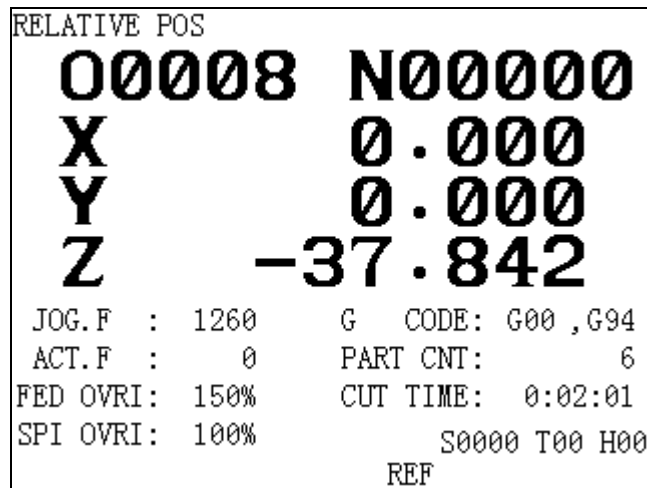
8.1 Machine Zero




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 switch fixed on the machine. Usually the switch is fixed on the positive max. strokes of X, Y, Z axes.

8.2 Machine Zero Return Steps

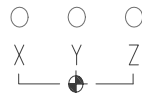


Press  key, it enters the Machine zero mode, the bottom line of the screen page shows “REF”, as following figure shows:



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 indicator

Note 1 If the machine zero is not fixed on the machine, machine zero operation B/C is unallowed;

Note 2 While the coordinate is moved out from the machine zero, the machine zero finish indicator is gone out.

Note 3 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.183;

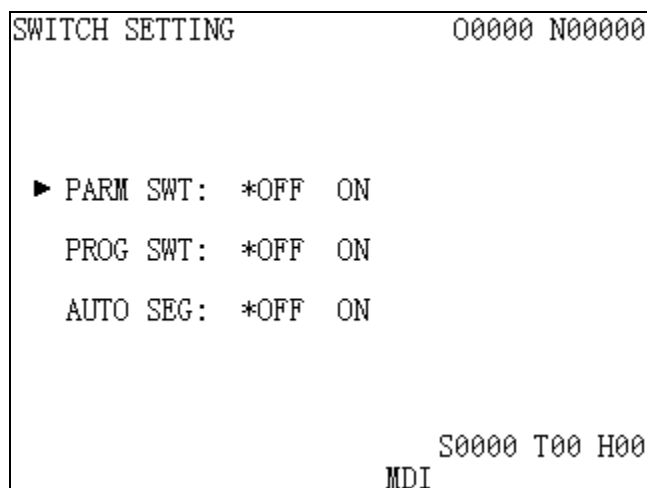
Note 4 See details in the 3rd part INSTALLATION AND CONNECTION for the parameters concerning with the machine zero.




CHAPTER 9 DATA SETTING, BACKUP AND RESTORE



9.1 Data 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, as is shown in following figure:





1 Press  key to enter the Setting interface, then press  or  key to enter the 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  key, “*” moves to the left to set the switch for OFF, press  key, “*” moves to the right to set the switch for ON.

Only the PARM SWT is set for ON, could the parameter be modified; so are PROG SWT and AUTO SEG.

Note: When the PARM SWT is shifted from “OFF” to “ON”, an alarm will be issued by CNC system.

By pressing the ,  key together, the alarm can be cancelled. If the PARM SWT is shifted again, no alarm is issued. For security it should set the PARM SWT for “OFF” after the parameter modification is finished.

9.1.2 Graphic setting




Press  key to enter the Graphic interface, by pressing the  or  key, the following graphic parameter page is shown:

```

GRAPH                                00008 N00000
▶COORD OPT      1  (XY0 YX1 ZX2 XYZ3
SCALE          50.000 YZ4 ZY5 XZ6 XZY7)
CENTER         -76.000 (X axis value)
CENTER         -36.000 (Y axis value)
CENTER         -8.000  (Z axis value)
X MAX.         44.000
Y MAX.         64.000
Z MAX.        106.000
X MIN.        -196.000
Y MIN.        -136.000
Z MIN.        -122.000

                                S0000 T00 H00
                                MDI
    
```

A: Graphic parameter setting

- 1 In MDI mode, press ,  key to move the cursor to the parameter to be set;
- 2 Key in the corresponding value;
- 3 Press  key to finish the setting.

B: Graphic parameter meaning

- 1 SCALE: to set the ratio of the drawing
- 2 CENTER: to set the coordinates of the LCD center in workpiece coordinate system
- 3 MAX, MIN: after the axis maximum and minimum value is set, the scaling, and graphic center will be automatically set by CNC system.

X MAX: the maximum value of X axis in program (unit: mm)

X MIN: the minimum value of X axis in program (unit: mm)



Y MAX: the maximum value of Y axis in program (unit: mm)



Y MIN: the minimum value of Y axis in program (unit: mm)

Z MAX: the maximum value of Z axis in program (unit: mm)




Z MIN: the minimum value of Z axis in program (unit: mm)

C: Scaling up and down of graphic

In Graphic page, the graphic path can be scaled up and down by the keys ,  in the Edit keypad.




Press  key each time, the graphic path is scaled up $\sqrt{2}$ fold, press  key each time, the graphic path is scaled down $\sqrt{2}$ fold.

D: The start, stop and clearing of the graphic path

In Graphic page, press the  key once, it starts the drawing, and “*”sign is moved before S; press the  key once, it stops drawing, and “*”sign is moved before T; press  key once, it clears the current graphic path.

9.1.3 Parameter setting

By the parameter setting, the characteristics of the driver and machine can be adjusted. See Appendix 1 for their significance.





Press  key to enter the Parameter interface, then press  or  key to switch the parameter page, as is shown in the following figure:



BIT PARAMETER				00008 N00000			
NO.		DATA		NO.		DATA	
001		00011000		009		00000000	
002		00100011		010		00011111	
003		00000000		011		00000000	
004		01000000		012		00011011	
005		00010001		013		10000011	
006		00000000		014		00000111	
007		00000000		164		10000000	
008		00000111		168		00000000	
BIT4:1/0:C comp.is radius/diameter value							
*** RDRN DECI ORC *** DCS PROD SCW							
NO. 004 =				S0000 T00 H00			
MDI							


A Modification of the bit parameter

1 字节修改: Byte modification

- 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  key in parameter No., then press  key.

- 4) Key in the new parameter value;
- 5) Press  key, the parameter value is entered and displayed.
- 6) After all parameters setting is finished, the PARM SWT needs to be set for OFF for security.

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 cue line, as is shown in following figure:

BIT PARAMETER				00008 N00000			
NO.		DATA		NO.		DATA	
001		00000000		009		00000000	
002		00100010		010		00001111	
003		00100000		011		00000000	
004		01000000		012		00011011	
005		00010001		013		10000000	
006		00000000		014		00000111	
007		00000000		164		00000000	
008		00000111		168		00000000	
BIT7:1/0:Reserved							
*** RDRN DECI ORC *** DCS PROD SCW							
NO. 004 = 01100000				S0000 T00 H00			
MDI							

INPUT




Press key to finish the parameter modification. The page is shown as following:


BIT PARAMETER		00008 N00000	
NO.	DATA	NO.	DATA
001	00011000	009	00000000
002	00100011	010	00011111
003	00000000	011	00000000
004	01100000	012	00011011
005	00010001	013	10000011
006	00000000	014	00000111
007	00000000	164	10000000
008	00000111	168	00000000



BIT4:1/0:C comp.is radius/diameter value
 *** RDRN DECI ORC *** DCS PROD SCW
 NO. 004 = S0000 T00 H00
 MDI





2 Bit modification

- 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 modified, 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

CHANGE

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 0, and the other bits unchanged.

Move the cursor to “No.004” by the steps above, press and hold  key for 2 seconds or press

CHANGE

key to skip to a bit of the parameter, as is shown in the following figure;

BIT PARAMETER		00008 N00000	
NO.	DATA	NO.	DATA
001	00011000	009	00000000
002	00100011	010	00011111
003	00000000	011	00000000
004	01100000	012	00011011
005	00010001	013	10000011
006	00000000	014	00000111
007	00000000	164	10000000
008	00000111	168	00000000
BIT7:1/0:Reserved			
*** RDRN DECI ORC *** DCS PROD SCW			
NO. 004		S0000 T00 H00	
		MDI	

Move the cursor to “BIT5” by pressing  or  key, as is shown in the following figure:

BIT PARAMETER		00008 N00000	
NO.	DATA	NO.	DATA
001	00011000	009	00000000
002	00100011	010	00011111
003	00000000	011	00000000
004	01100000	012	00011011
005	00010001	013	10000011
006	00000000	014	00000111
007	00000000	164	10000000
008	00000111	168	00000000
BIT5:1/0:*DEC signal is low/high level			
*** RDRN DECI ORC *** DCS PROD SCW			
NO. 004		S0000 T00 H00	
		MDI	

Key in “0” to finish the modification.

BIT PARAMETER		00008 N00000	
NO.	DATA	NO.	DATA
001	00011000	009	00000000
002	00100011	010	00011111
003	00000000	011	00000000
004	01000000	012	00011011
005	00010001	013	10000011
006	00000000	014	00000111
007	00000000	164	10000000
008	00000111	168	00000000
BIT5:1/0:*DEC signal is low/high level			
*** RDRN DECI ORC *** DCS PROD SCW			
NO. 004		S0000 T00 H00	
		MDI	

B Modification of the data parameter, screw-pitch data

1 Data parameter modification

- 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 for OFF for security.
- Example 1: to set the data parameter No.022 to 4000.

Move the cursor to “No.022” by the steps above, key in “4000” by sequence in the cue line, as is shown in the following figure;

DATA PARAMETER				00008 N00000	
NO.	DATA		NO.	DATA	
015	1		023	7600	
016	1		024	7600	
017	1		025	100	
018	1		026	100	
019	1		027	100	
020	1		028	200	
021	0		029	100	
_022	7600		030	50	
Max. speed of rapid traverse in X(mm/min)					
NO. 022 = 4000_ S0000 T00 H00					
MDI					

Press key to finish the modification. The page is shown as following:

DATA PARAMETER				00008 N00000	
NO.	DATA		NO.	DATA	
015	1		023	7600	
016	1		024	7600	
017	1		025	100	
018	1		026	100	
019	1		027	100	
020	1		028	200	
021	0		029	100	
_022	4000		030	50	
Max. speed of rapid traverse in X(mm/min)					
NO. 022 = S0000 T00 H00					
MDI					

Example 2: to set the X axis value of the screw-pitch data No.000 to 12, Z axis value of that to 30.

Move the cursor to screw-pitch data No.000 by the steps above, key in “X12” by sequence in the cue line, as is shown in the following figure;

SCREW-PITCH COMP				00008 N00000
NO.	X	Y	Z	
000	0	0	0	
001	200	200	0	
002	0	200	0	
003	0	0	0	
004	1	1	0	
005	3	3	0	
006	0	0	0	
007	0	0	0	
Units: 0.001 (mm)				
NO. 002	X 12_			S0000 T00 H00
MDI				

Press key to finish the modification. The page is shown as following:

SCREW-PITCH COMP				00008 N00000
NO.	X	Y	Z	
000	0	0	0	
001	200	200	0	
002	12	200	30	
003	0	0	0	
004	1	1	0	
005	3	3	0	
006	0	0	0	
007	0	0	0	
Units: 0.001 (mm)				
NO. 002				S0000 T00 H00
MDI				

The same as above, key in “Z30” by sequence in the cue line, press key to finish the modification. The page is as following:

SCREW-PITCH COMP				00008 N00000
NO.	X	Y	Z	
000	0	0	0	
001	200	200	0	
002	12	200	30	
003	0	0	0	
004	1	1	0	
005	3	3	0	
006	0	0	0	
007	0	0	0	
Units: 0.001 (mm)				
NO. 002				S0000 T00 H00
MDI				

9.2 Data Restore and Backup

The user data (such as bit parameter, data parameter, tool offset and screw-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 shown as following:





```

PARM. OPERATION                                00000 N00000

▶ Backup PAR. ( User )
Resume PAR. ( User )
Resume Default PAR.1(Test)
Resume Default PAR.2(Step)
Resume Default PAR.3(Servo)
PRESS[IN]+[P]TO CONFIRM(POWER ON AGAIN)

S0000 T00 H00
MDI
    
```

1、 Turn on the parameter switch;

2、 Press  key to enter the MDI mode, then press  key ( or  key if necessary) to enter Backup PAR. page;

3、 Move the cursor to the desired item;

4、 Press ,  keys together.

Note 1: 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.

Note 2: The backup and restore data are different depending on the current password level: the 3rd, 4th, 5th password level user can performed the backup and restore operation of the bit parameter, data parameter but the screw-pitch parameter; the 2nd password level user can perform that of the bit parameter, data parameter and screw-pitch parameter.

Example: to restore the CNC parameter to 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 “Resume Default PAR. 2 (Step)”, as following figure shows:

```

PARM. OPERATION                                00008 N00000

Backup PAR. ( User )
Resume PAR. ( User )
Resume Default PAR.1(Test)
Resume Default PAR.2(Step)
▶ Resume Default PAR.3(Servo)
PRESS[IN]+[P]TO CONFIRM(POWER ON AGAIN)

SERVO PAR BACKUP RECOVERED (TURN OFF&ON
S0000 T00 H00
MDI
    
```

Press ,  keys together, the CNC system prompts “SERVO PAR BACKUP RECOVERED (TURN OFF&ON)”.

9.3 The Password Setting and Modification

To protect the part programs, CNC parameters from malignant modification, this GSK980MD provides an

authority setting function that is graded for 4 levels. By descending sequence, they are machine builder (2nd) level, equipment management (3rd) level, technician (4th) level, machining operation (5th) level.

- 2nd level: the CNC bit parameter, data parameter, screw-pitch data, tool offset data, part program edit, PLC ladder transmission etc. are allowed;
- 3rd level: initial password 12345, the CNC bit parameter, data parameter, tool offset data, part program edit operations are allowed;
- 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, screw-pitch data operations are unallowed.
- 5th level: no password. Only the machine panel operation is allowed, and the operations of part program edit and selection, the modification operations of CNC bit parameter, data parameter, screw-pitch data, tool offset data are unallowed.

```

PARM. OPERATION                                00008 N00000

Backup PAR. ( User )
Resume PAR. ( User )
Resume Default PAR. 1 (Test)
Resume Default PAR. 2 (Step)
▶ Resume Default PAR. 3 (Servo)
PRESS[IN]+[P] TO CONFIRM (POWER ON AGAIN)

SERVO PAR BACKUP RECOVERED (TURN OFF&ON
                                S0000 T00 H00
                                MDI
    
```

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.

- a) 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 “ALTER PASSWORD:”line (end line);
- b) Press key once, the cursor shifts a line downward. If the current cursor locates at the end line, by pressing key once, the cursor moves to the 1st line.

9.3.1 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 4th level, as the following page shows. The 3rd level password of CNC is 12345, please alter the current level to 3rd level.

```

PASSWORD SETTING          00008 N00000

      CURRENT LEVEL: 4
      SET LOWER LEVEL

▶ INPUT  PASSWORD:  *****
      UPDATE  PASSWORD:

      Can edit prog,input macro var&offset

                                S0000 T00 H00
                                MDI
    
```

INPUT

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 shown as following:

```

PASSWORD SETTING          00008 N00000

      CURRENT LEVEL: 3
      SET LOWER LEVEL

▶ INPUT  PASSWORD:
      UPDATE  PASSWORD:

      Modify parameter and edit program
      PASSWORD PASSED

                                S0000 T00 H00
                                MDI
    
```

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.

9.3.2 Alteration of the password

Steps for password modification:

- 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;

INPUT

- 3 Key in the new password, and press key;

- 4 The CNC system prompts “PLEASE INPUT USER PASSWORD AGAIN”, the page is shown as following:

```

PASSWORD SETTING          00008 N00000
      CURRENT LEVEL: 3
      SET LOWER LEVEL
      INPUT  PASSWORD:
      ► UPDATE  PASSWORD:

      Modify parameter and edit program
      PLEASE INPUT  USER PASSWORD AGAIN
                        S0000 T00 H00
                        MDI

```

INPUT

5 After reinputting the password, press key, if the two passwords input are identical, CNC prompts “PASSWORD UPDATED.”. So the password modification is successful.

```

PASSWORD SETTING          00008 N00000
      CURRENT LEVEL: 3
      SET LOWER LEVEL
      INPUT  PASSWORD:
      ► UPDATE  PASSWORD:

      Modify parameter and edit program
      PLEASE INPUT  USER PASSWORD AGAIN
                        S0000 T00 H00
                        MDI

```

6 If the two passwords input are not identical, CNC prompts “PASSWORD CHECKOUT ERROR.”, the page is as following:

```

PASSWORD SETTING          00000 N00000
      CURRENT LEVEL: 3
      SET LOWER LEVEL
      INPUT  PASSWORD:
      ► UPDATE  PASSWORD:

      Modify parameter and edit program
      PASSWORD CHECKOUT ERROR.
                        S0000 T01 H00
                        MDI

```

9.3.3 Set lower level

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 following:

```

PASSWORD SETTING          00000 N00000
      CURRENT LEVEL: 3
      SET LOWER LEVEL
      ► INPUT  PASSWORD:
      UPDATE  PASSWORD:

      Modify parameter and edit program

                                S0000 T00 H00
                                MDI
    
```

3 Press key, the CNC prompts "CURRENT LEVEL TO 4, MAKE SURE? "; the page is as following:

```

PASSWORD SETTING          00008 N00000
      CURRENT LEVEL: 3
      ► SET LOWER LEVEL
      INPUT  PASSWORD:
      UPDATE  PASSWORD:

      Modify parameter and edit program
      CURRENT LEVEL TO4, MAKE SURE?

                                S0000 T00 H00
                                MDI
    
```

4 Press key again, if the demotion is successful, the page is as following:

```

PASSWORD SETTING          00008 N00000
      CURRENT LEVEL: 4
      ► SET LOWER LEVEL
      INPUT  PASSWORD:
      UPDATE  PASSWORD:

      Can edit prog,input macro var&offset

                                S0000 T00 H00
                                MDI
    
```

Note: If the current level is the 5th level, the demotion operation is unallowed.

CHAPTER 10 COMMUNICATION

10.1 Brief of GSK980MD TDComm Communication Software

The files upload and download between PC and CNC system is via the TDComm software which is easy to operate and has a high communication efficiency and reliability.

•System configuration (PC) of TDComm software

Hardware: a universal PC with RS232 serial port, serial port communication cable (3-wire system)

Operation system: Microsoft Windows 98/2000/XP/2003

•Software interface

The TDComm software interface is easy to operate for user, the following figure is the interface after the software is executed:

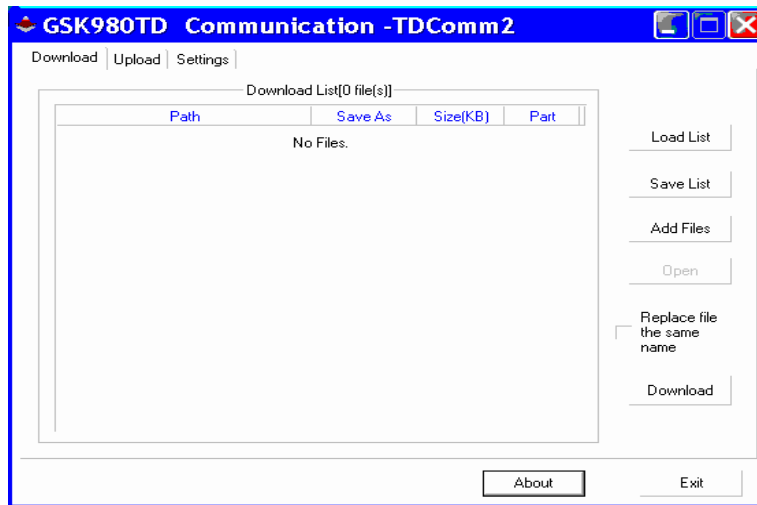


Fig.10-1 Files download interface (PC→CNC)

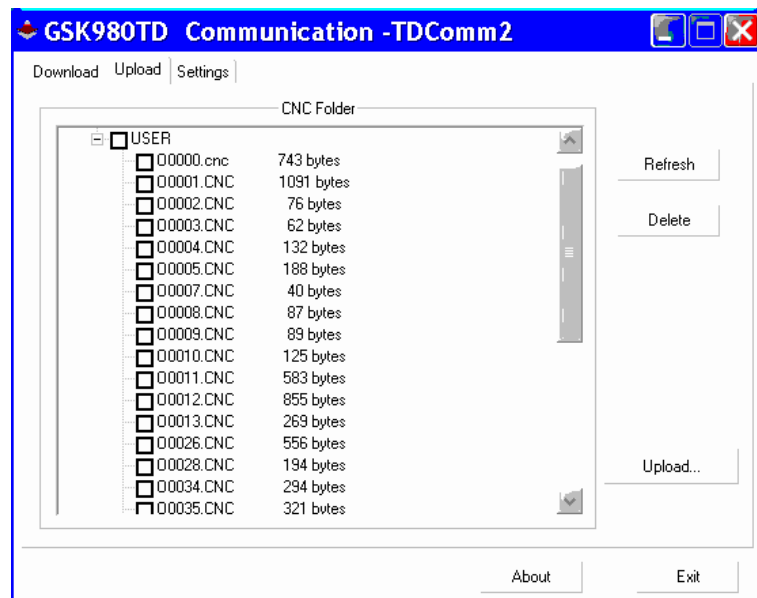


Fig.10-2 Files upload interface (CNC→PC)

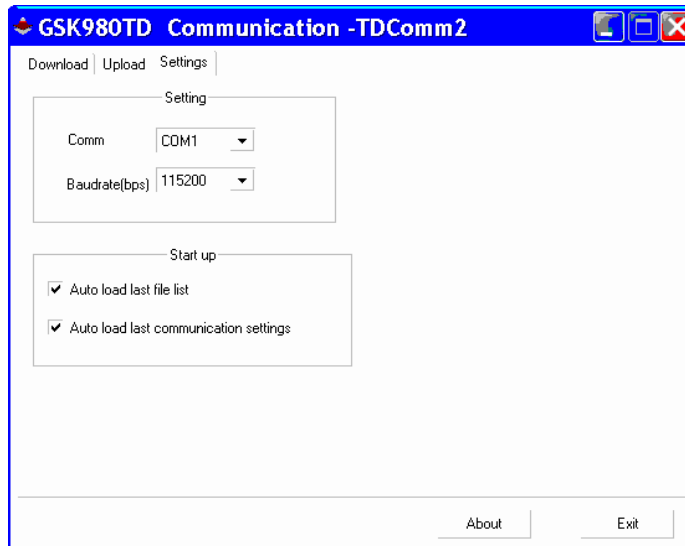


Fig.10-3 Setting option interface

10.1.1 Files download (PC→CNC)

For downloading, click **【Add Files】** button, choose the files to be sent to CNC to the list box, the messages of files paths, file names saved in CNC system, file length and the CNC storage area etc. will be listed. The list can be saved into a file which can be opened with no need to choose them repeatedly when transferring the same files next time.

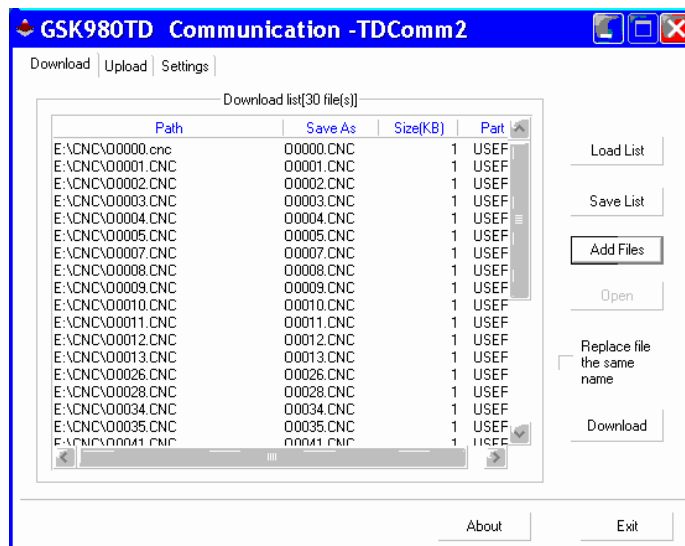


Fig. 10-4

As the Fig. 10-4 shows: the left side is files list box, the right side is **【Load List】**, **【Save List】**, **【Add Files】**, **【Open】**, **【Download】** 5 buttons and an option of **【Replace file the same name】**.

Files list box: The list box used for files to be sent to CNC lists the messages of files paths, file names saved in CNC, file length, CNC storage area etc.. And this list can be saved into a file which can be opened with no need to choose them repeatedly when transferring the same files next time by program.

【Load List】 : It is used to load the list of files saved in the harddisk.

【Save List】 : It is used to save the current files list into the harddisk.

【Add Files】 : It is used to choose a file from the harddisk to add to the list of the files to be sent.

【Download】 : It is used to start the transferring of the files that are chosen.

【Open】 : It is used to view the files chosen by text type.

【Replace file the same name】 : It is used to replace the file that has a same name with the file to be transferred in CNC without inquiring the user when transferring files.

▲ Files selection dialogue box

Click the **【Add Files】** button where the cursor locates in Fig.10-4, it pops up“please select the files to be sent” dialogue box, it may select the files to be sent in this box, or click **【All NC Files】** button to select all the CNC files under the current list into the files list box. It defaults that the file names saved in CNC are identical with the original file names. If the file name length is over 8 characters, it is automatically shortened for 8.

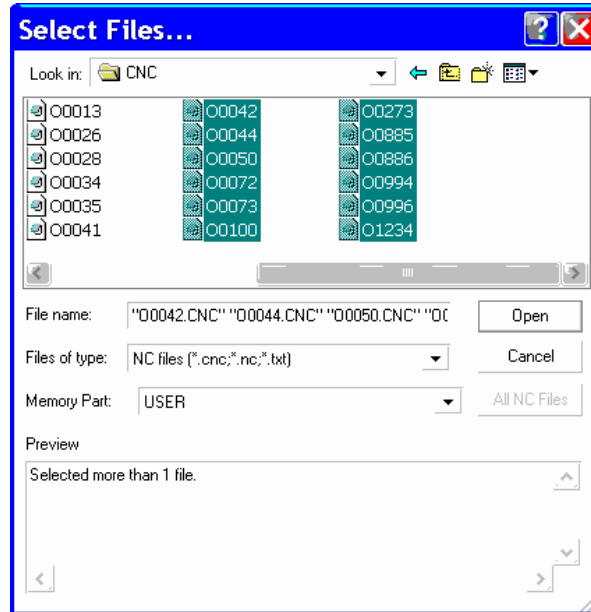


Fig. 10-5

When a single file is selected, it may view the file content at the bottom of the dialogue box. As is shown in Fig. 10-6:

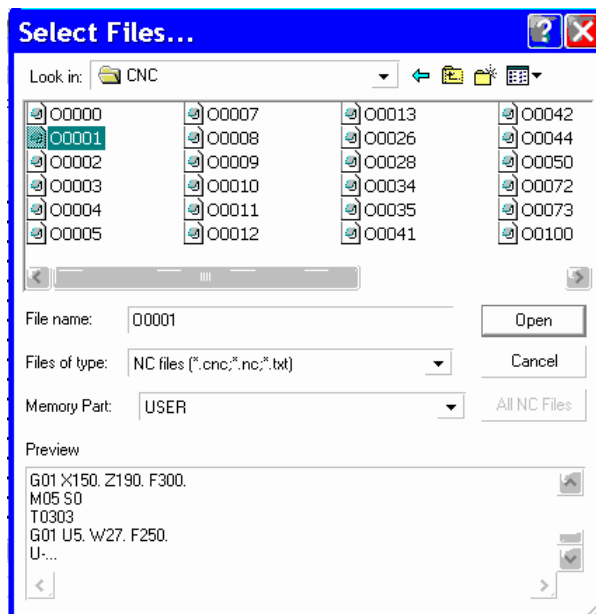


Fig. 10-6

▲ File list attribute modification

If the attribute (file path, name and storage area of the file saved) of the listed file item is needed to be modified, it may double click the item to pop up the setting dialogue box, as the Fig.10-7, Fig 10-8, Fig.10-9 show:

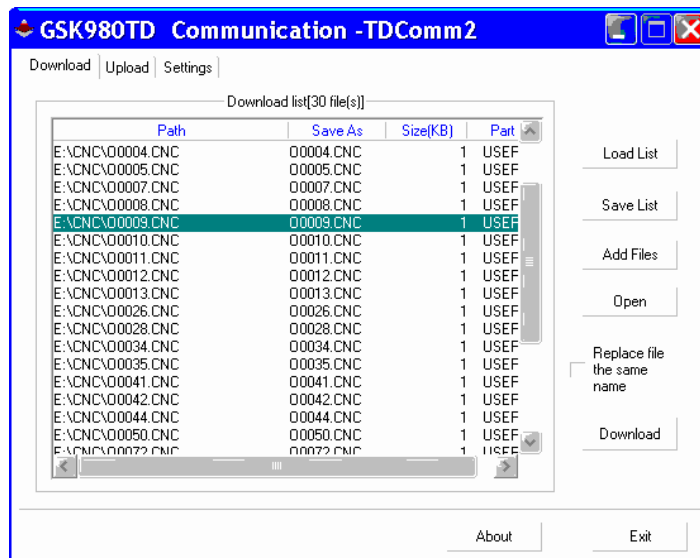


Fig. 10-7

For example, to modify the name of the highlighted item in the list for "O0001", it may perform the following operations:

Move the cursor to the file listed item, as above figure 10-7 shows, double click the item, it pops up the setting dialogue box as Fig. 10-8, it may modify the file path and file name for saving (as Fig.10-9).

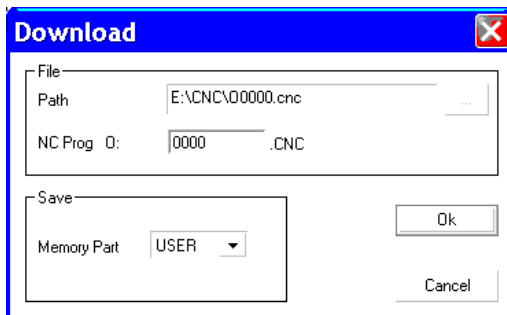


Fig. 10-8

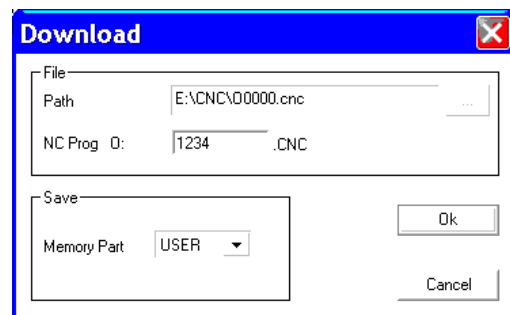


Fig. 10-9

Click the OK button to confirm the setting. As is in Fig. 10-10

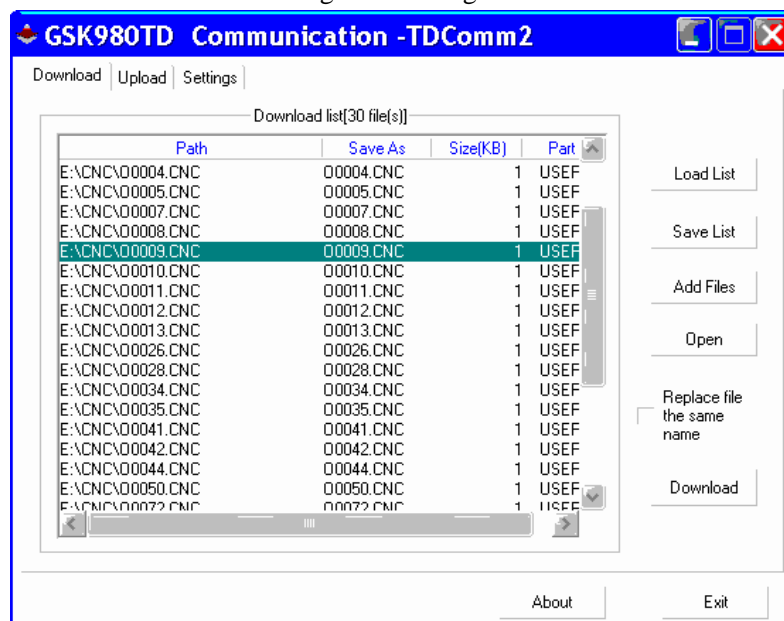


Fig. 10-10

By this means, it can add the file to be sent into the list one by one.

In addition, it may double click an item in the list to pop up the file setting dialogue box as Fig. 10-7 to change the file name, storage area etc.; after the item is selected, click the item by the right key of the mouse, a menu will be popped up for Remove or Remove All operation.

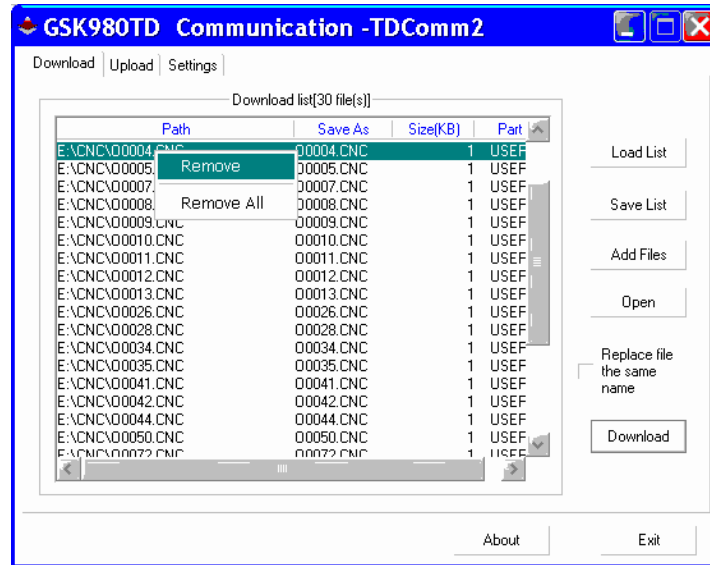


Fig. 10-11

After the adding, it may click 【Save List】 to save the list as a file which can be used by clicking 【Load List】 to load it for downloading without setting the file list one by one next time, as the following Fig.10-12 shows:

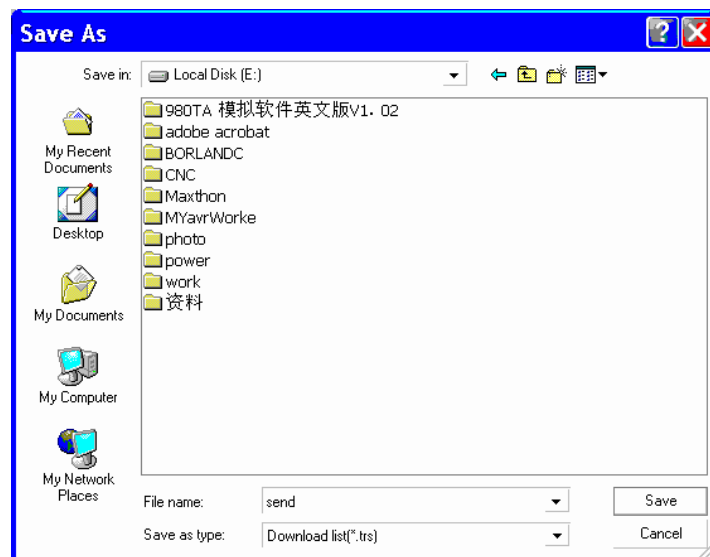


Fig.10-12

Note: It only supports the 8.3 format CNC file names(i.e. 8 English or numerical number characters for the file name, 3 English or numerical number characters for expansion name), Chinese or other characters are not supported. This should be observed for CNC file name setting when downloading or renaming the files. If the file name doesn't conform to this rule, the item will be listed by red color. Please do the modification by this rule.

After the file list is set, click the 【Download】 button to start the files downloading. And it pops up the communication state dialogue box, in this box the messages of current file transferring, process and communication state can be viewed (Fig.10-13).

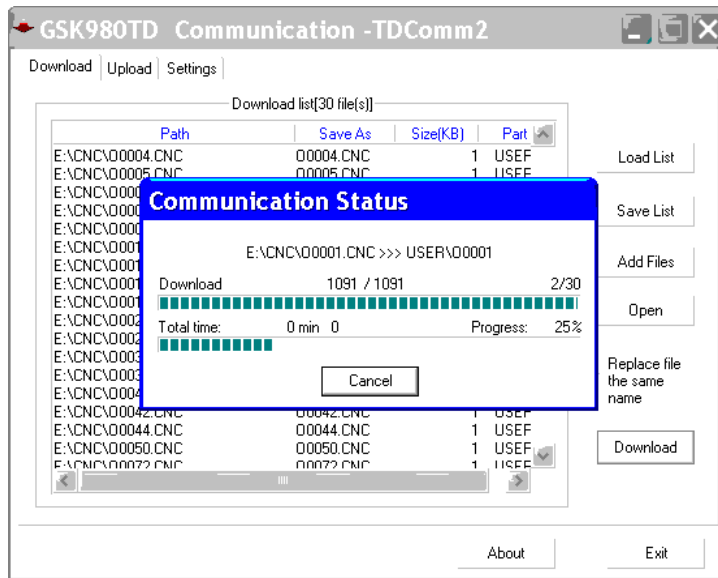


Fig. 10-13

If there is a file that has the same name with the file being transferred in CNC system, a dialogue box will be popped up. It may continue the transferring by selecting the Replace or Skip operation in the box to replace or skip the file.

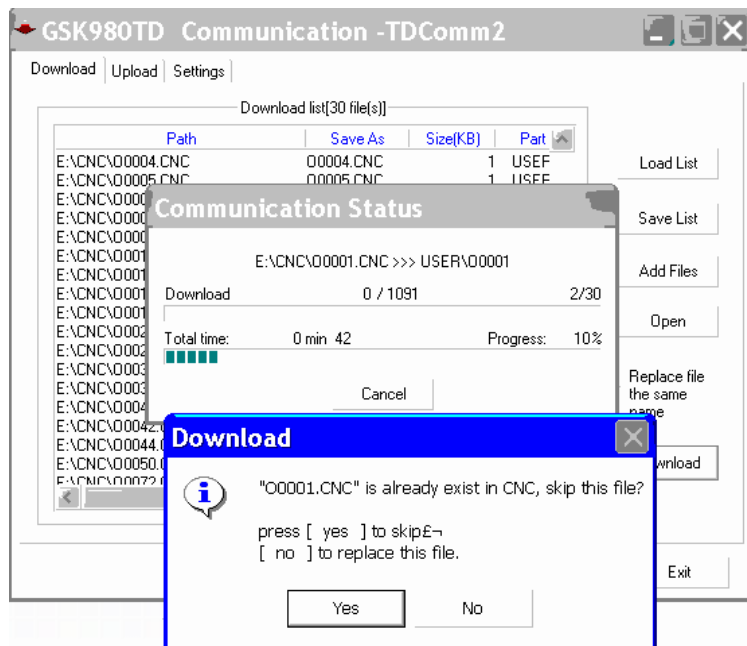


Fig. 10-14

10.1.2 Upload of the files (CNC→PC)

List refresh: Select **【Upload】** to look up the files lists of the CNC zones.

Files deletion: Select **【Upload】** to delete the files selected from CNC.

File rename: Select **【Upload】** to rename the files in CNC user storage area.

1.Operation at PC side

Click **【Upload】** to select the interface as the following figure shows, click **【Refresh】** button, the files list will be shown in the file list box of the CNC main interface. Click the little box of the item left side to choose the file to be transferred. The red tick means that the file is chosen.

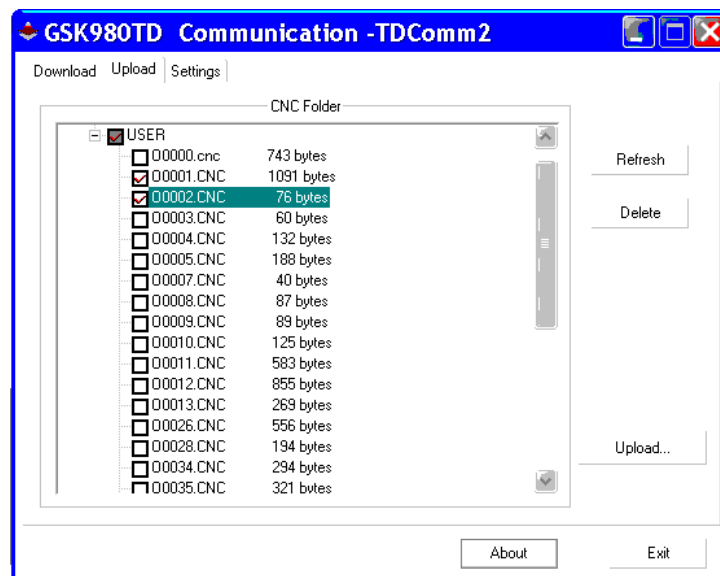


Fig. 10-15

After the file selection, click the **【Upload...】** button to select a saving list for receiving the files from CNC. There may be a communication box as Fig. 10-13 during the transmission, and it disappears when the transmission is over.

2.Operation at CNC side

After the connection with CNC is done, the software receives the files uploaded by CNC during its free time. When CNC starts to upload the files, the program starts data uploading immediately. After the receiving is over, it prompts the user to save the files.

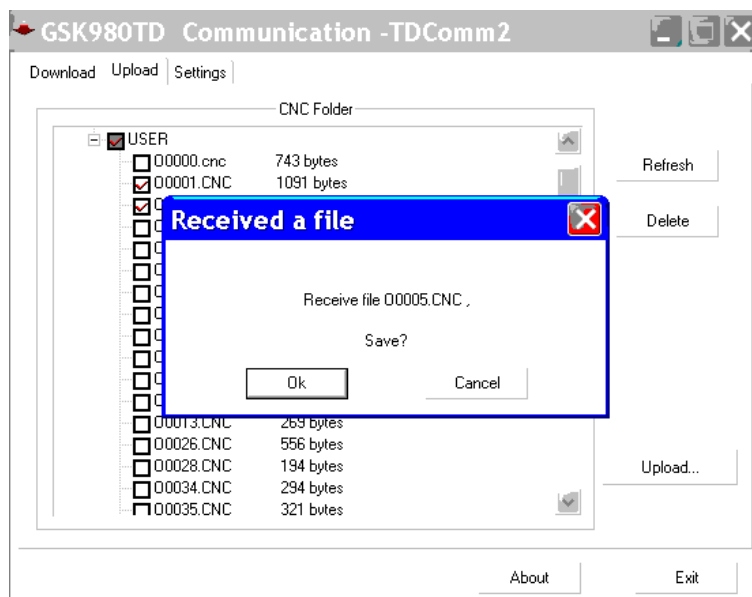


Fig. 10-16

3.Deletion of CNC files

In the **【Upload】** mode of the dialogue box attribute page, choose the file to be deleted in the file list, then click the **【Delete】** button in the communication control button area, the chosen file will be deleted (multiple chosen files can be deleted at a time).

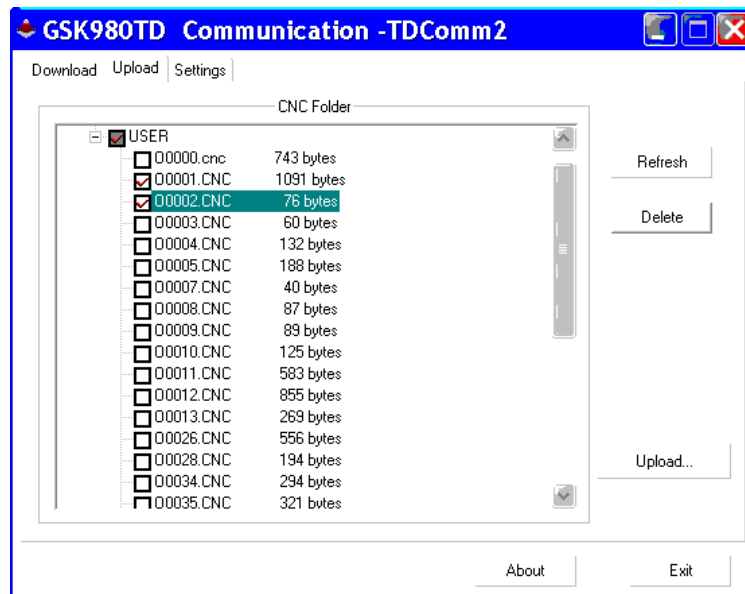
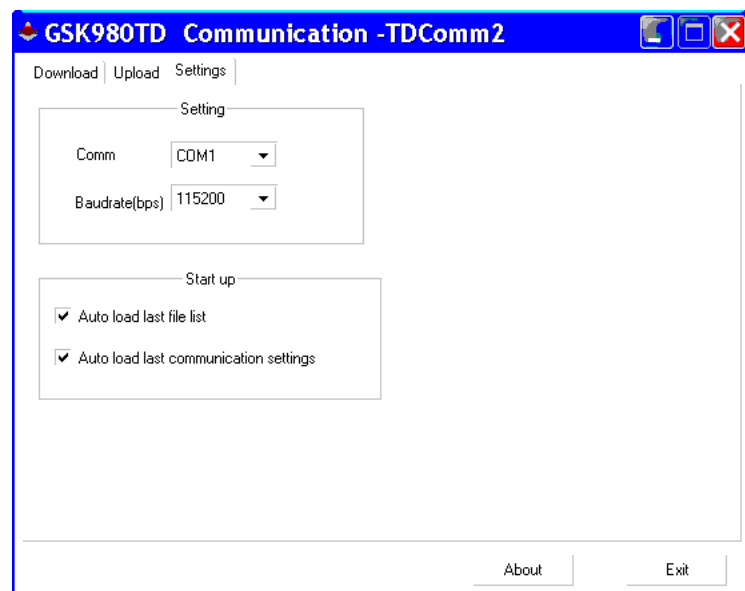


Fig. 10-17

10.1.3 Setting option



1.Selection box of communication setting

Comm port selection: For choosing the PC as the COM port of the communication, COM1~COM4 are optional.

Baudrate: For the communication baudrate selection, 4800~115200 available.

2.Selection box auto loading as program start

Auto load last file list: as the program is started next time, load/not load the file list in the file download interface (Fig.10-1) of the last time.

Auto load last communication settings: As the program is started next time, load/not load the communication setting of the last time.

10.2 Preparation Before Communication

1.To connect communication cable as both PC and CNC are power off:

Connection of PC to CNC: DB9 male plug into the XS36 communication interface of CNC, DB9 female plug into the 9-male serial interface of PC (COM0 or COM1)

Connection of CNC to CNC: two DB9 male plugs into the XS36 communication interface of CNC separately.

2. Set the BIT5 (RS232) (see Appendix 1) of the CNC bit parameter No.002 to 1;

3. Set the baudrate of the communication to make the baudrates of PC and CNC, CNC and CNC to be consistent;

● Baudrate setting of CNC

The communication baudrate of CNC serial interface of this GSK980MD turning machine is set by data parameter No.044, and its setting range is 50~115200 (Unit: bps) . While the data is to be transferred between CNC and PC, the setting value should be not less than 4800. Its factory setting: 115200

● Baudrate setting of PC

After the communication software is run, click the “Settings” with the mouse left key, its interface is shown as following:




Comm port selection: to select the port for communication (COM1, COM2, COM3, COM4)

Baudrate: to select the communication baudrate 4800,9600,19200,38400,57600,115200 (Unit:bps)

Note 1: If the part program is needed to be transferred, the program switch should be turned on; if the parameters, tool offset etc. are needed to be transferred, the parameter switch should be turned

on. If an alarm is issued after turning on the switch, it may press  and  key together to cancel this alarm.

Note 2: If the machining is being performed, to ensure a reliable communication, please stop the machining. While the data files are to be sent by CNC initiatively, please change the current mode to the Edit mode.

Note 3: If the transmission is needed to be stopped, it may press the  key.

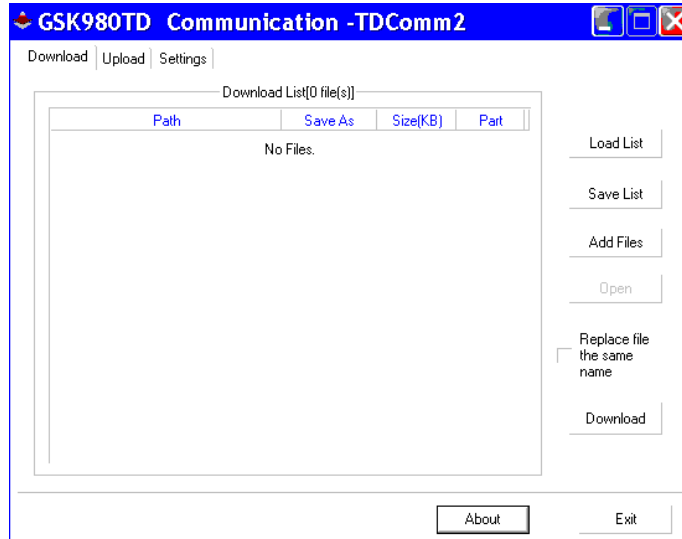
Note 4: Don't cut off the power during the data transmission, or the data transmission error may be issued.

10.3 Data Input (PC→CNC)

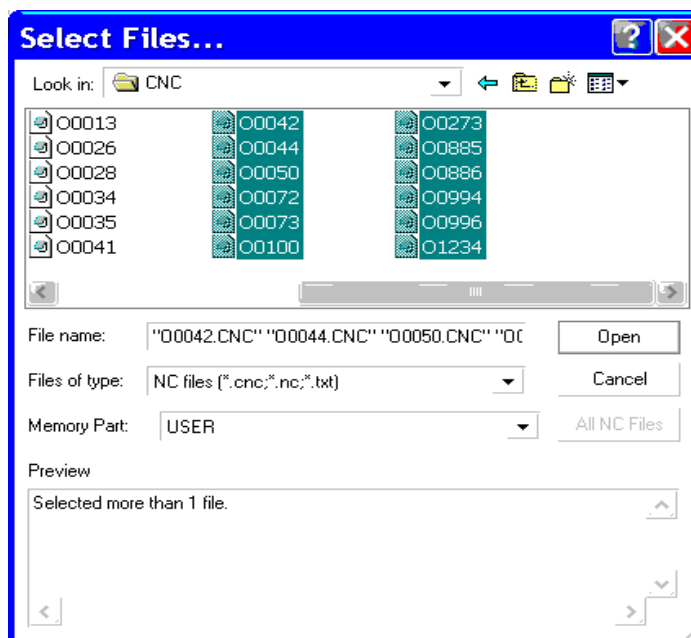
The data file in PC can be input to CNC by performing the input function, the data receivable for CNC includes part program, parameter, tool offset, screw-pitch offset etc..

10.3.1 Input of program

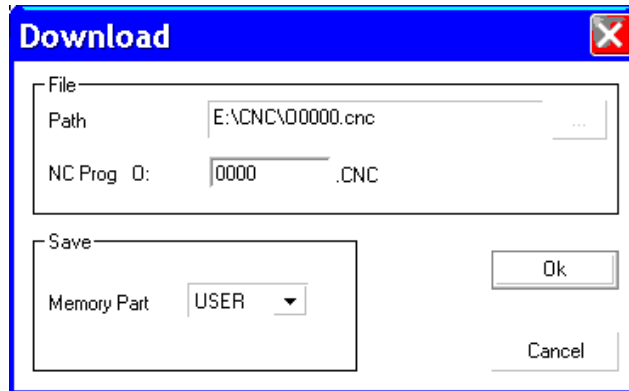
- 1.Set the corresponding authority in CNC(2nd level for macro), and set the program switch for ON;
- 2.Edit the part program (files with the expansion name *.cnc,*.nc,*.txt supported) in PC and save it into the harddisk;
- 3.Run the communication software, then click the **【Download】** button, the page is as following:



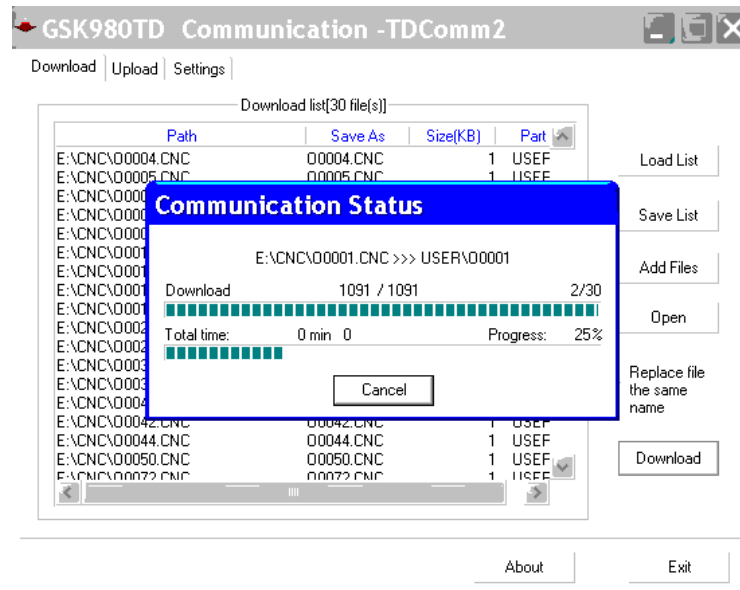
- 4.In the software interface of above figure, click **【Add Files】** button, it pops up the file adding dialogue box, select the part program edited, as the following figure shows:



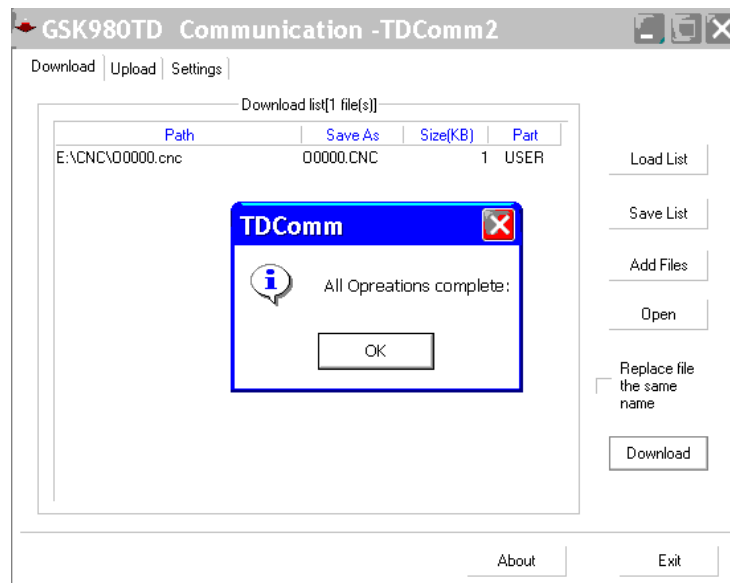
- 5.In the software interface of above figure, it may double click a part program to rename it:



6. Click the **【Download】** button, it shows as following:



7. After the transmission is finished, click the **【OK】** button in the popped up window, it shows as following:



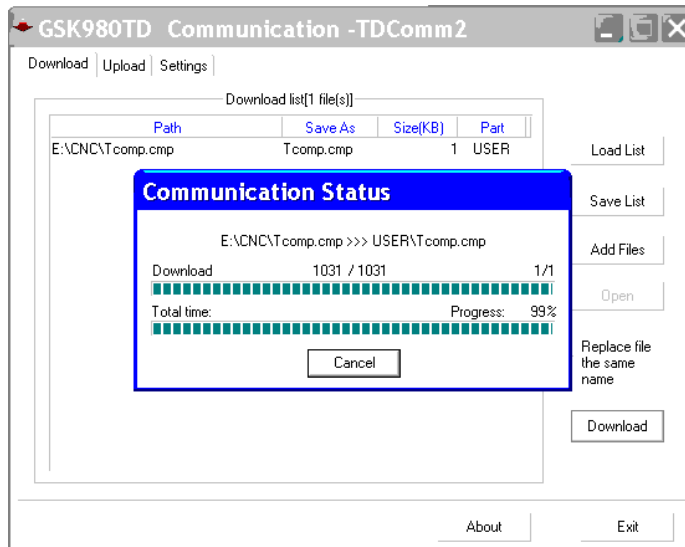
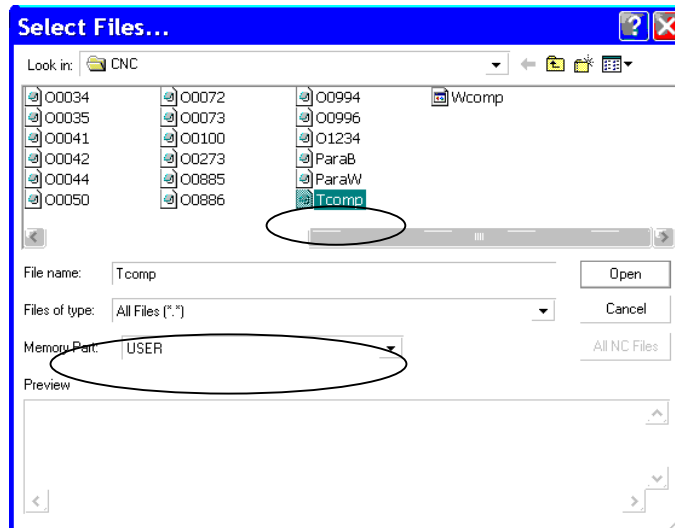
8. By the steps from 1 to 7, other programs can be transferred. And one program or multiple programs may be transferred once.

Note: This operation is performed under the 3rd level of CNC side.

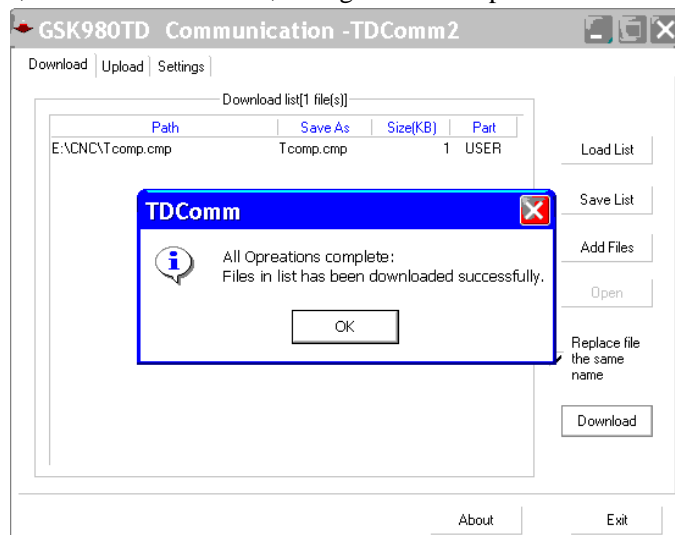
10.3.2 Input of the tool offset

1.Set the corresponding authority in CNC (2nd LEVEL for macro), and set the parameter switch for ON, select the Edit mode;

2.Run the communication software, select the Download option, then click the **【Add Files】** button to add the tool offset file (with the expansion name .cmp, or transferring a tool offset file in advance from CNC if there is no such file) to be transferred, click **【Download】** button, and the pages are as following:



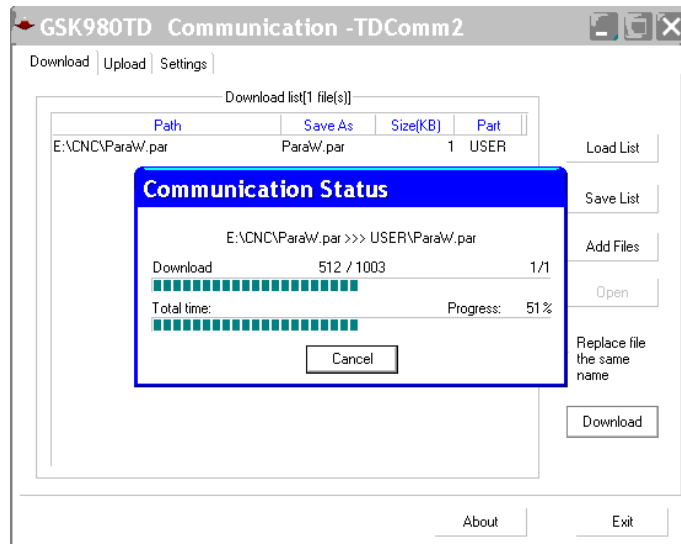
3.After the transmission, click **【OK】** button, then go on other operations.



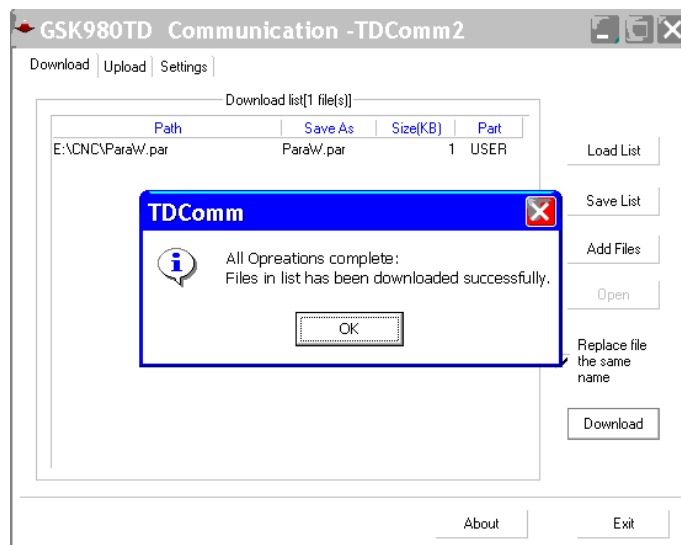
10.3.3 Input of the parameter

1. Set the corresponding authority in CNC (2nd LEVEL for screw-pitch data), and set the PARM SWT for ON, select the Edit mode;

2. Run the communication software in PC side, select the “Download “ option, then click the 【Add Files】button to add the parameter file (with the expansion name .par, or transferring an OFFSET file in advance from CNC if there is no such file) to be transferred, click 【Download】 button to start downloading, and the page is as following:



3. After the transmission, click 【OK】 button, then go on other operations.



Note 1: The parameter file includes bit parameters, data parameters and screw-pitch data, which can be operated by the user requirement.

Note 2: The bit parameters and data parameters sequence No. shown on PC begin from zero, which correspond to those in CNC one by one.

Note 3: If the bit parameters and data parameters are needed to be transferred from PC, the CNC operation authority must be 3rd level or above.

Note 4: If the screw-pitch parameters are needed to be transferred from PC, the CNC operation authority must be 2nd level or above.

10.4 Data Output (CNC→PC)

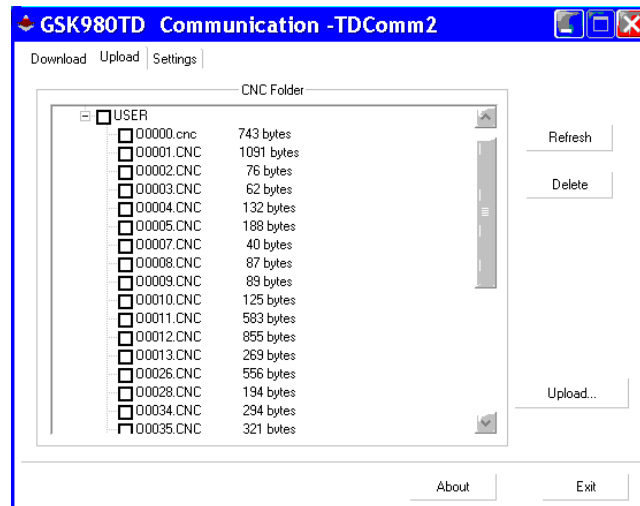
The data file in CNC can be input to PC by performing the output function, the data receivable for PC includes part program, parameter, tool offset, screw-pitch offset etc..

10.4.1 Output of a program

A program may be output to PC from CNC, the outputting steps are as following:

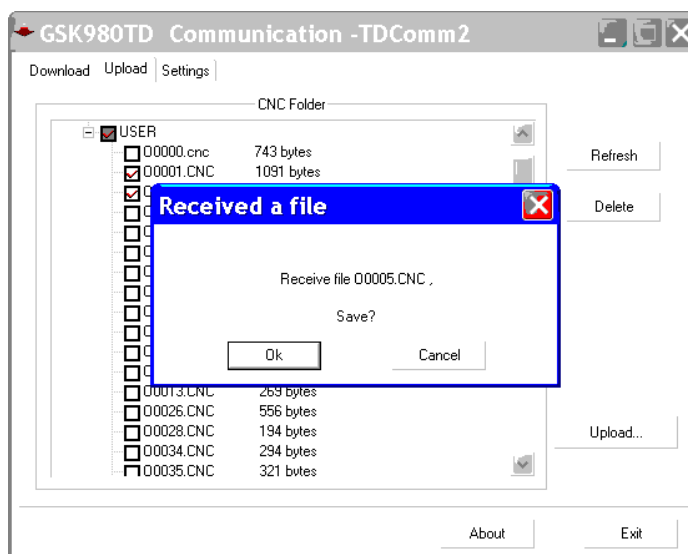
Method 1: Operation at CNC side

1. Select Edit mode and enter the PRG CONTENT page;
2. Run the communication software at the PC side, then switch to the "Upload" page;

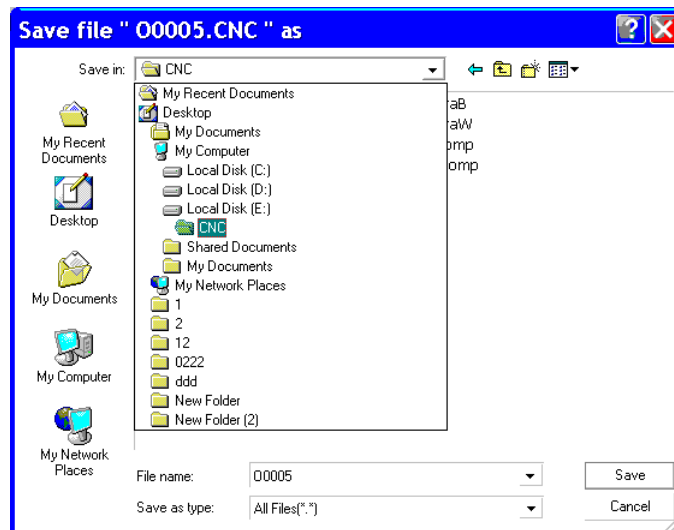


3. Key in the address key and the name of the program to be transferred (this step may be omitted if transferring the current program);

4. Press key to start the output, the characters "OUTPUT" will be shown and flickered at the right bottom of the CNC display page, and the page of PC side is shown as following after the transmission is over:

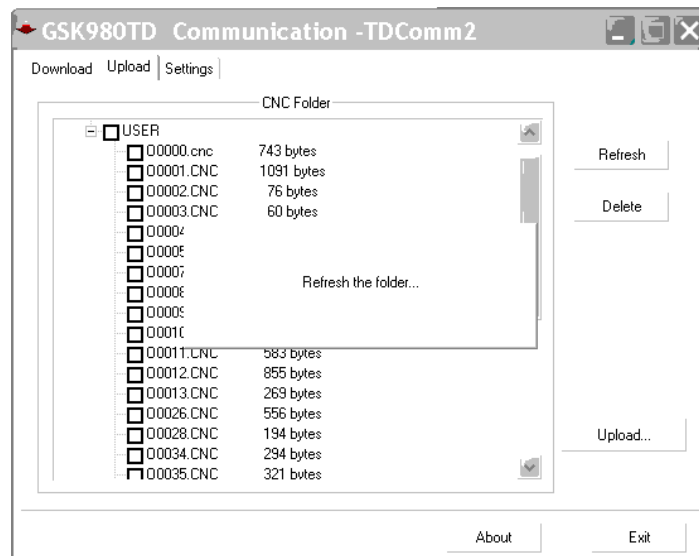


5. If the file doesn't need to be saved, click **【Cancel】** button to exit the dialogue box; if it needs to be saved, click **【OK】** button, a location dialogue box will be popped up, select a saving path, then click **【OK】** button.

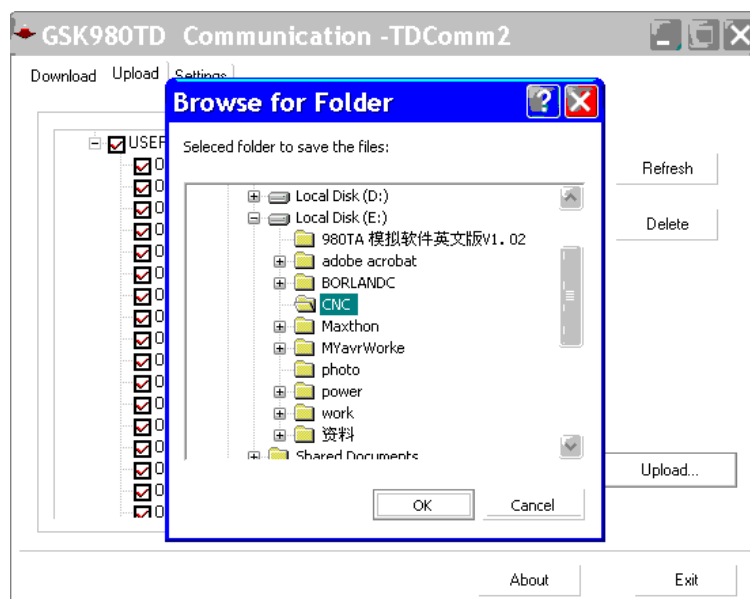


Method 2: Operation at PC side

1. Select Edit mode and enter the PRG CONTENT page;
2. Run the communication software at the PC side, then switch to the “Upload” page and click **【Refresh】** button;



3. Select the program to be saved, click **【Upload...】**, as the following figure shows (to select No.10 program for saving):



4. Select a path for saving and then click **【OK】** button.

10.4.2 Output of all programs

All the programs that are saved in CNC memory can be output to PC by user, the steps are as following:

1. Select Edit mode and enter the PRG CONTENT page;

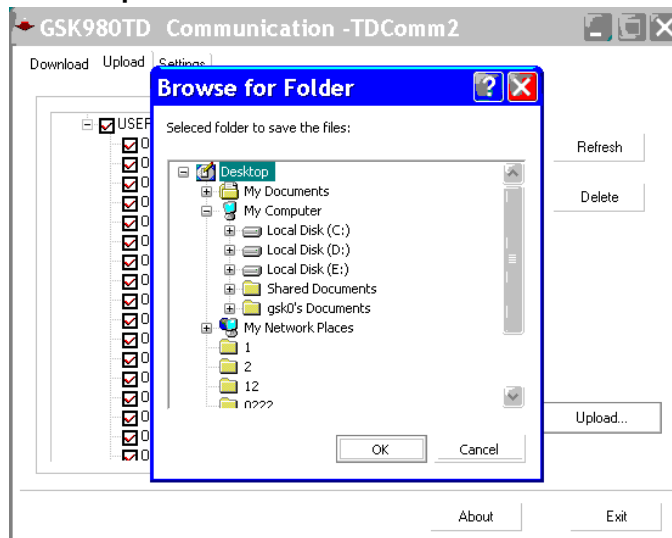
2. Run the communication software at the PC side, then switch to the “Upload” page;

3. Key in the address key , symbol key and the address keys , , by sequence at CNC side;

4. Press key to start the output, the characters “OUTPUT” will be shown and flickered at the right bottom of the CNC display page, and the page of PC side is shown as following after the transmission is over:

5. Save the part program one by one by the method of step 5 stated in Section 10.4.1 of this chapter.

Note: It may also operate at PC side by the Method 2 in Section 10.4.1 of this chapter, select all the part programs and then select a path to save them.

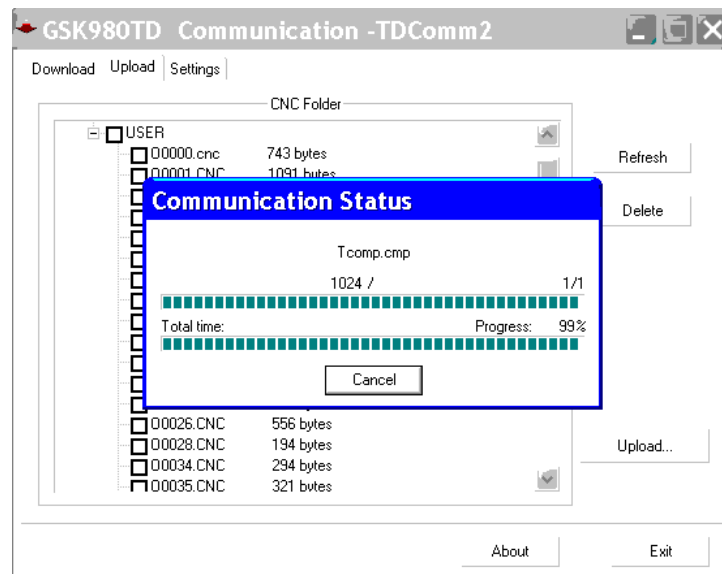


10.4.3 Output of the tool offset

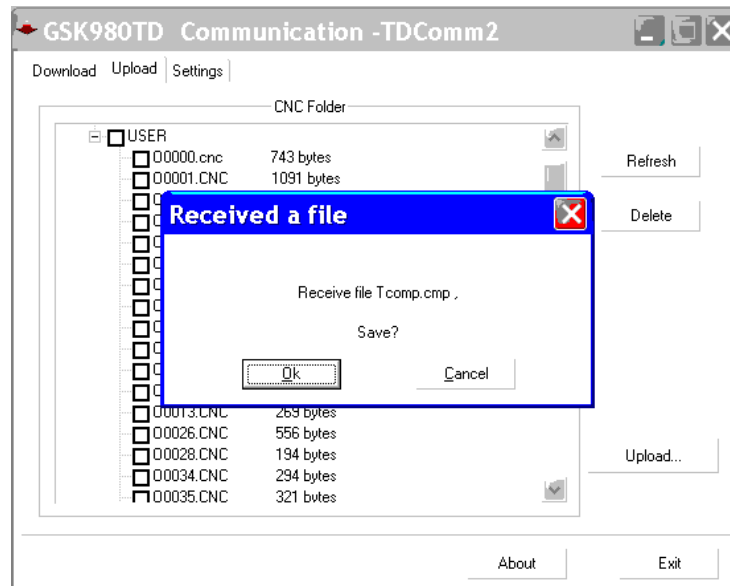
1. Select Edit mode and enter the TOOL OFFSET page;

2. Run the communication software at the PC side, then switch to the “Upload” page;

3. Press key to start the output at CNC side, the characters “OUTPUT” will be shown and flickered at the right bottom of the CNC display page, and the page of PC side is shown as following:



4. After the transmission, the page of PC side is shown as following (default name: Tcomp.cmp):



5. Select a path to save the file by the method of step 5 stated in Section 10.4.1 of this chapter.

10.4.4 Output of the parameter


1. Select Edit mode and enter the page under the Parameter interface;

If the bit parameter is to be transferred, enter the BIT PARAMETER page;

If the bit parameter is to be transferred, enter the DATA PARAMETER page;

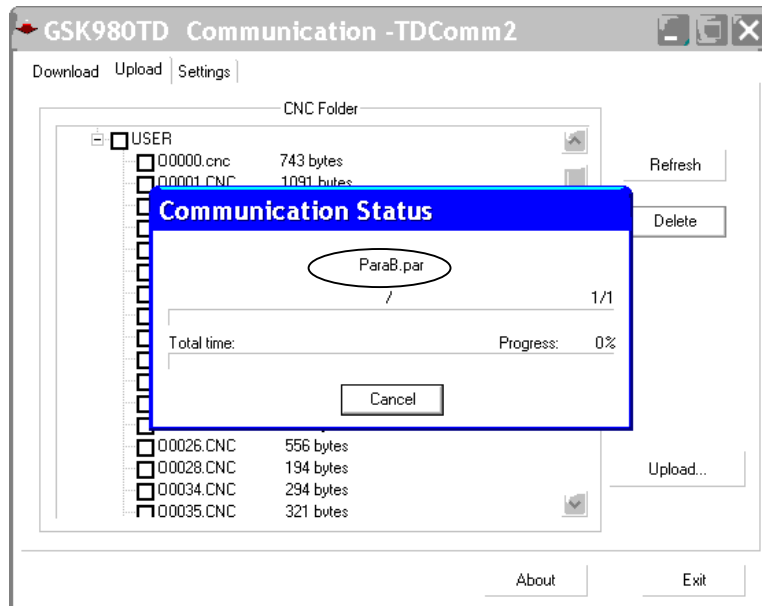
If the screw-pitch data is to be transferred, enter the SCREW-PITCH COMP page;

2. Run the communication software at the PC side, then switch to the "Upload" page;

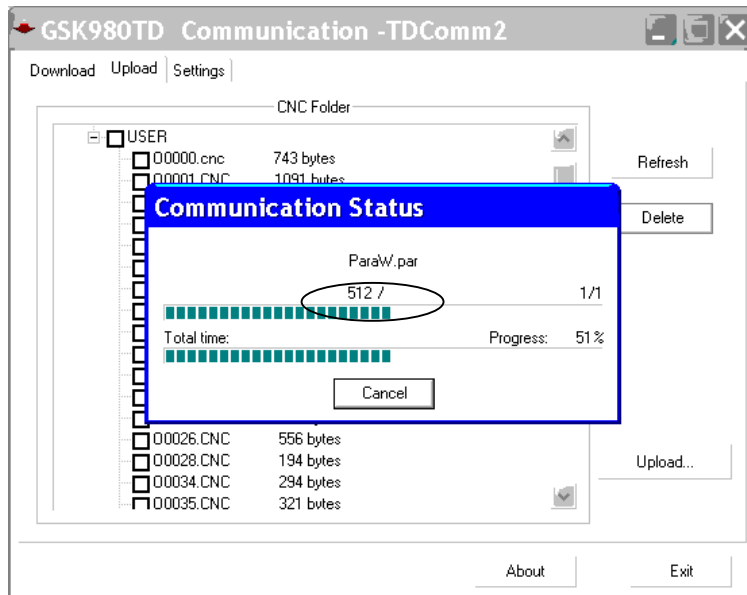
3. Press  key to start the transmission at CNC side, the characters "OUTPUT" will be shown and flickered at the right bottom of the CNC display page,

4. During the transmission, the file name shown at PC side is different depending on the bit parameter, the data parameter and the screw-pitch data, as the following figures show (the default name is marked by an ellipse):

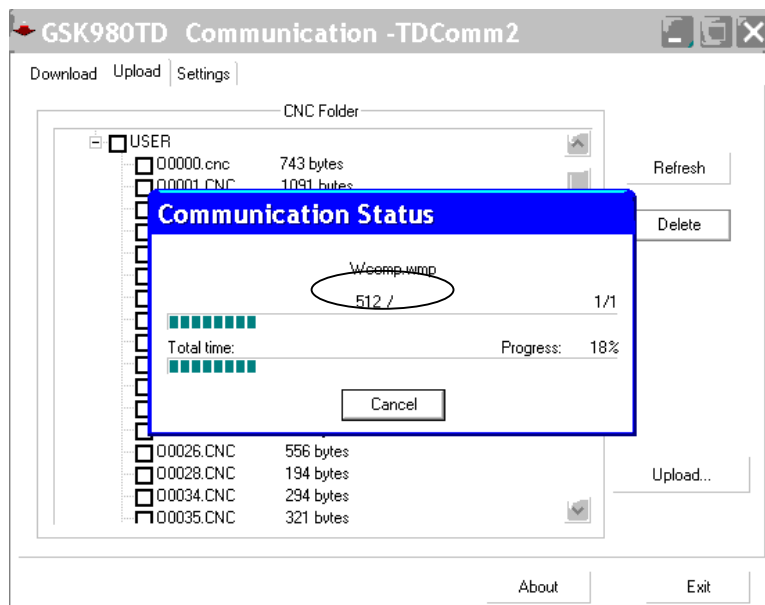
The bit parameter transmission is as following (the default name: **ParaB.par**):



The data parameter transmission is as following(the default name: **ParaW.par**):



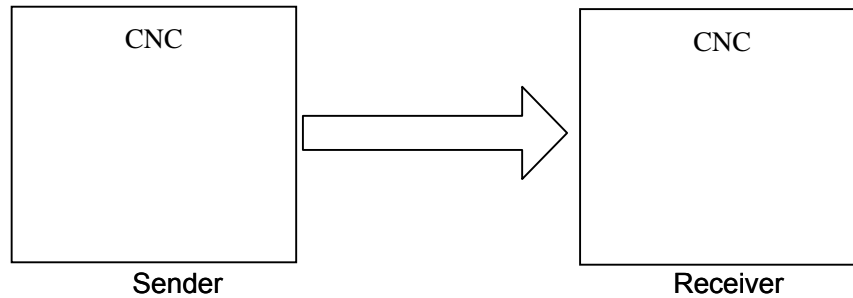
The screw-pitch data transmission is as following(the default name: **Wcomp.wmp**):



5. Select a path to save the file by the method of step 5 stated in Section 10.4.1 of this chapter.

10.5 Communication between CNC and CNC

To facilitate the GSK980MD turning machine using, the data transmission between two CNC is allowable. And the CNC which sends data is called sender, the CNC which receives the data is called receiver. Their sketch map is as following:



Cautions for mutual data transmission between two CNC systems:

1. The communication function of the sender and receiver are both valid, i.e. the BIT5 (RS232) of the bit parameter No.002 of both CNC systems are set to 1;
2. The communication baudrates of the sender and receiver are identical, i.e. the setting of the data parameter No.044 of both CNC systems are identical;
3. The sender and receiver are both in Edit mode;
4. The sender must enter the page where the data to be sent locate (e.g. It should enter the BIT PARAMETER page if the bit parameter is to be transferred);
5. The receiver should enter the corresponding authority level and turn on the relative (parameter or program) switch, as the following table shows:

Data received	Authority	Remark
Part program (Program No. less than 9000)	4 th , 3 rd or 2 nd level	Turn on program switch
Macro (Program No. more than or equal to 9000)	2 nd level	Turn on program switch
Tool offset	4 th , 3 rd or 2 nd level	
Bit parameter	3 rd or 2 nd level	Turn on parameter switch
Data parameter	3 rd or 2 nd level	Turn on parameter switch
Screw-pitch data	2 nd level	Turn on parameter switch

6. The operation steps are the same as that at CNC side in Section 10.4 "Data Output (CNC→PC)" of this chapter.

PART 3

INSTALLATION AND CONNECTION

CHAPTER 1 INSTALLATION LAYOUT	I -1
1.1 GSK980MD Connection.....	I -1
1.1.1 GSK980MD back cover interface layout.....	I -1
1.1.2 Interface explanation.....	I -1
1.2 GSK980MD Installation	I -2
1.2.1 GSK980MD External dimensions	I -2
1.2.2 GSK980MD-B External dimensions	I -2
1.2.3 Installation conditions of the cabinet	I -3
1.2.4 Protection methods against interference	I -3
CHAPTER 2 DEFINITION and CONNECTION of INTERFACE SIGNALS	II -1
2.1 Connection to Driver.....	II -1
2.1.1 Drive interface definition.....	II -1
2.1.2 Command pulse and direction signals	II -1
2.1.3 Driver alarm signal	II -1
2.1.4 Axis enable signal nEN.....	II -2
2.1.5 Pulse disable signal nSET	II -2
2.1.6 Zero signal nPC.....	II -2
2.1.7 Connection to driver	II -3
2.2 Connection to Spindle Encoder	II -4
2.2.1 Spindle encoder interface definition	II -4
2.2.2 Signal explanation.....	II -4
2.2.3 Connection of spindle encoder interface.....	II -4
2.3 Connection to Handwheel.....	II -5
2.3.1 Handwheel interface definition.....	II -5
2.3.2 Signal explanation.....	II -5
2.4 Connection to Transducer	II -6
2.4.1 Analog spindle interface definition.....	II -6
2.4.2 Signal explanation.....	II -6
2.4.3 Interface connection of transducer.....	II -6
2.5 Connection of GSK980MD to PC	II -6
2.5.1 Communication interface definition	II -6
2.5.2 Communication interface connection	II -7
2.6 Connection of Power Interface	II -7
2.7 I/O interface definition.....	II -8
2.7.1 Input signal.....	II -9
2.7.2 Output signal	II -10
2.8 Machine Zero	II -11
CHAPTER 3 PARAMETER	III-1
3.1 Parameter description (by sequence)	III-1
3.1.1 Bit parameter.....	III-1
3.1.2 Data parameter	III-6
3.2 Parameter description (by function sequence).....	III-11
3.2.1 Axis control logic.....	III-11
3.2.2 Acceleration&deceleration control	III-12
3.2.3 Machine protection	III-13
3.2.4 Thread function	III-13

3.2.5	Spindle control.....	III-14
3.2.6	Tool function.....	III-14
3.2.7	Edit and display	III-15
3.2.8	Precision compensation	III-16
3.2.9	Communication setting.....	III-16
3.2.10	Machine zero return.....	III-16
CHAPTER 4 MACHINE DEBUGGING METHODS AND STEPS		IV-1
4.1	Emergency Stop and Limit	IV-1
4.2	Driver configuration	IV-1
4.3	Gear Ratio Adjustment	IV-1
4.4	Acceleration&deceleration characteristic adjustment	IV-2
4.5	Machine Zero Adjustment	IV-3
4.6	Spindle Adjustment.....	IV-5
4.6.1	Spindle encoder	IV-5
4.6.2	Spindle brake	IV-5
4.6.3	Switch volume control for spindle speed.....	IV-5
4.6.4	Analog voltage control for spindle speed	IV-5
4.7	Backlash Offset.....	IV-5
4.8	Step/Handwheel adjustment	IV-7
4.9	Other Adjustment.....	IV-7
CHAPTER 5 DIAGNOSIS MESSAGE.....		V-1
5.1	CNC Diagnosis	V-1
5.1.1	Signal diagnosis from machine to CNC	V-1
5.1.2	Axes moving state and data diagnosis of CNC	V-1
5.1.3	Keys diagnosis	V-1
5.1.4	CNC internal state.....	V-3
5.2	PLC state.....	V-3
5.2.1	X address *(others are defined by PLC except the following fixed addresses)	V-3
5.2.2	Address Y(except the following address, the other Y addresses are defined by PLC)	V-5
5.3	PLC Data	V-5
CHAPTER 6 MEMORIZING SCREW-PITCH ERROR COMPENSATION FUNCTION		VI-1
6.1	Function Explanation.....	VI-1
6.2	Specification	VI-1
6.3	Parameter Setting.....	VI-1
6.3.1	Screw-pitch compensation.....	VI-1
6.3.2	Screw-pitch error origin.....	VI-1
6.3.3	Offset clearance	VI-1
6.3.4	Offset value.....	VI-2
6.4	Cautions of Offset Setting.....	VI-2
6.5	Setting Examples of Offset Parameters	VI-2

CHAPTER 1 INSTALLATION LAYOUT

1.1 GSK980MD Connection

1.1.1 GSK980MD back cover interface layout

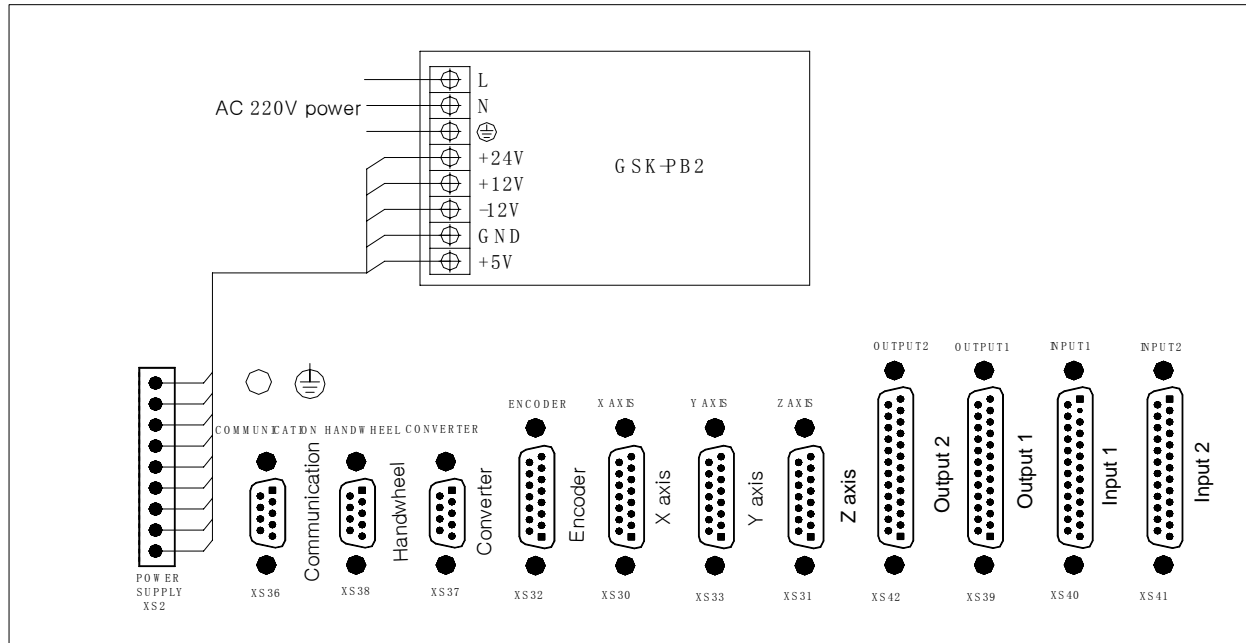


Fig. 1-1 GSK980MD back cover interface layout

1.1.2 Interface explanation

- Power box: GSK-PB2, for +5V, +24V, +12V, -12V, GND power supply
- XS30: X axis, 15-core DB female socket, for connecting X axis driver
- XS31: Z axis, 15-core DB female socket, for connecting Z axis driver
- XS33: Y axis, 15-core DB female socket, for connecting Y axis driver
- XS32: encoder, 15-core DB female socket, for connecting spindle encoder
- XS36: communication, 9-core DB female socket, for connecting PC RS232 interface
- XS37: transducer, 9-core DB male socket, for connecting transducer
- XS38: handwheel, 9-core DB male socket, for connecting handwheel
- XS39: output 1, 25-core DB female socket, interface for CNC signal outputting to machine
- XS40: output 1, 25-core DB male socket, interface for CNC receiving machine signal
- XS41: output 2, 25-core DB male socket, interface for CNC receiving machine signal
- XS42: output 2, 25-core DB female socket, interface for CNC signal outputting to machine signal

1.2 GSK980MD Installation

1.2.1 GSK980MD External dimensions

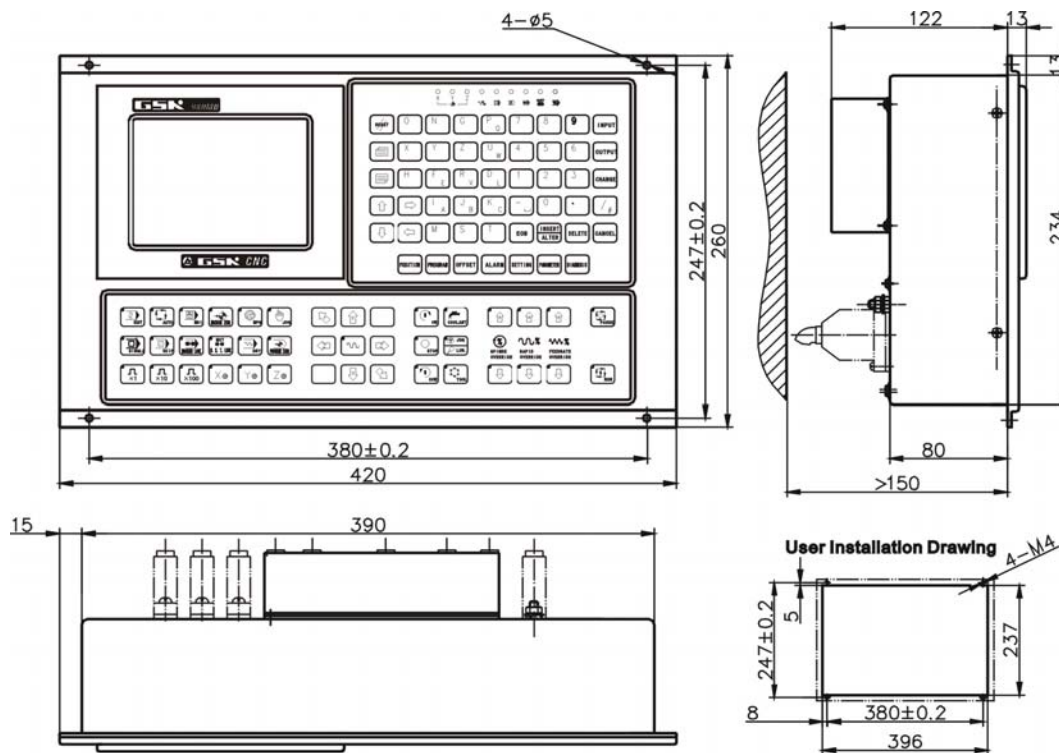


Fig. 1-2 GSK980MD External dimensions

1.2.2 GSK980MD-B External dimensions

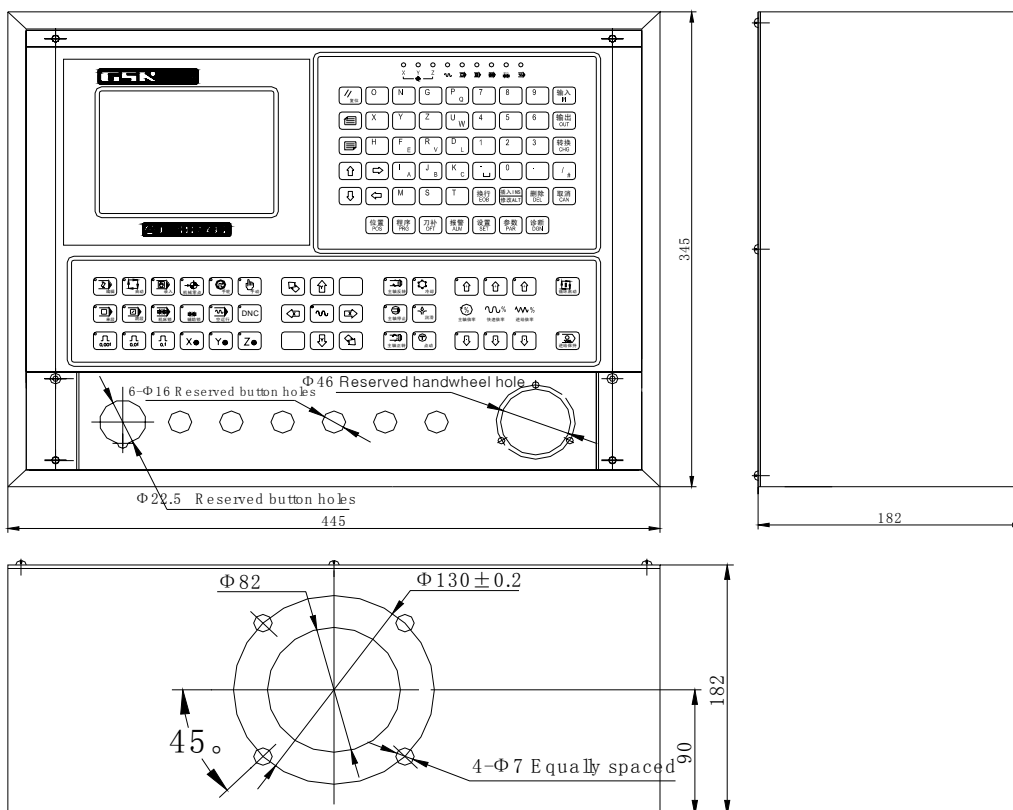


Fig. 1-3 GSK980MD-B external dimensions

1.2.3 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 less than 10°C when the cabinet inside temperature rises;
- Fans can 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.

1.2.4 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 (transducer, AC contactor, static generator, high-pressure generator and powered sectional devices etc.);
2. To supply the CNC via an isolation transformer with the machine grounded, the CNC and driver should connect to independent grounding wires from the grounding point;
3. To suppress 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);

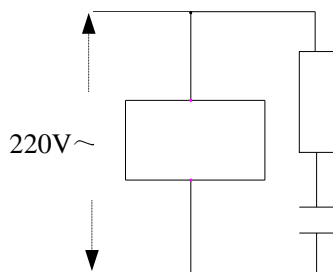


Fig. 1-4

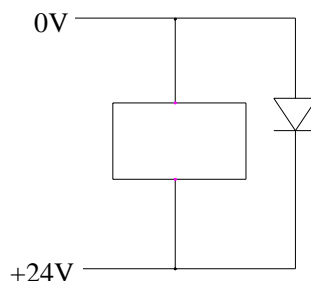


Fig. 1-5

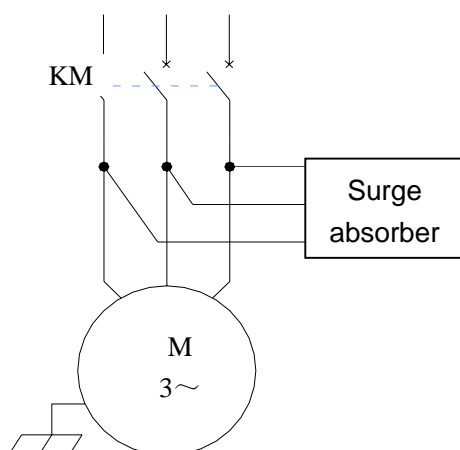


Fig. 1-6

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 driver	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 and CONNECTION of INTERFACE SIGNALS

2.1 Connection to Driver

2.1.1 Drive interface definition

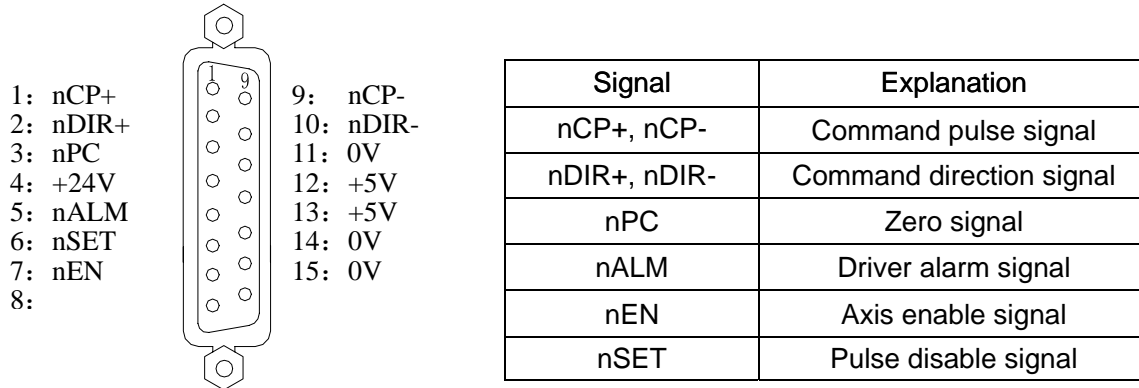


Fig. 2-1 XS30, XS31, XS33 interface
(15-core DB female socket)

Note: N stands for X, Z, or Y, the same is the following.

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 as Fig. 2-2:

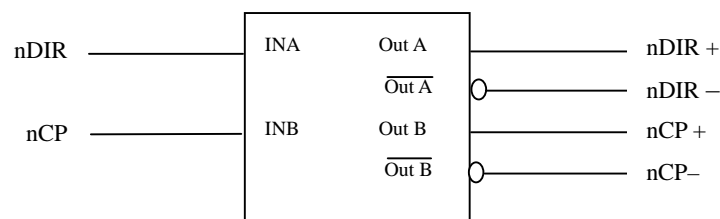


Fig. 2-2 Interior circuit of command pulse and direction signals

2.1.3 Driver alarm signal

The low or high level of the driver alarm level is set by the CNC bit parameter No.009 BIT0, BIT1 and BIT2, whose interior circuit is as Fig. 2-3:

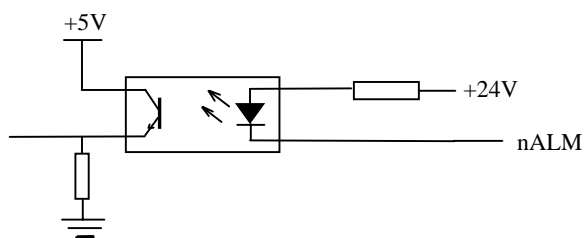


Fig. 2-3 Interior circuit of driver alarm signal

This input circuit requires that the driver transmits signal by the following types in Fig. 2-4:

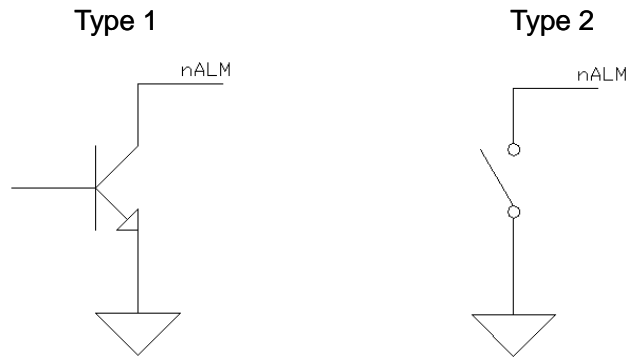


Fig. 2-4 Signal types of driver

2.1.4 Axis enable signal nEN

nEN signal output is valid as CNC works normally (nEN signal to 0V); when the driver alarm or emergency alarm occurs, CNC cuts off nEN signal output (nEN signal off 0V). the interior interface circuit is shown as Fig. 2-5:

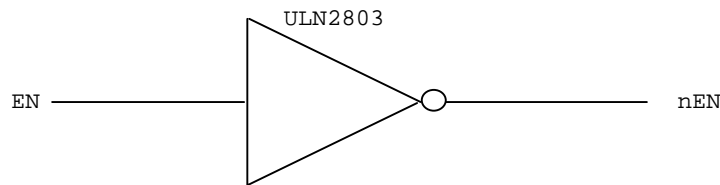


Fig. 2-5 Interior interface circuit for axis enable signal

2.1.5 Pulse disable signal nSET

nSET signal is used to control servo input disable which can enhance the anti-disturbance capability between CNC and driver. 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 as Fig. 2-6:

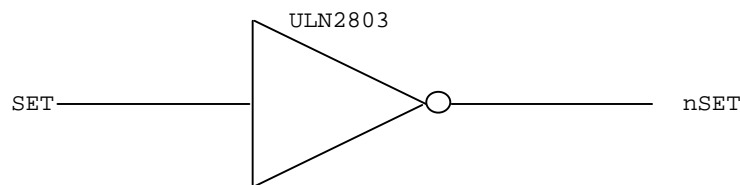


Fig. 2-6 Pulse disable signal circuit

2.1.6 Zero signal nPC

The one-turn or approach switch signal is taken as zero signal for machine zero return. It's interior connection circuit is as Fig.2-7.

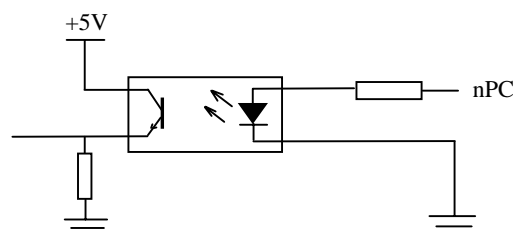


Fig. 2-7 Zero signal circuit

Note: nPC signal uses +24V level.

a) The connection for NPN Hall elements taken as both deceleration signal and zero signal is as Fig. 2-8:

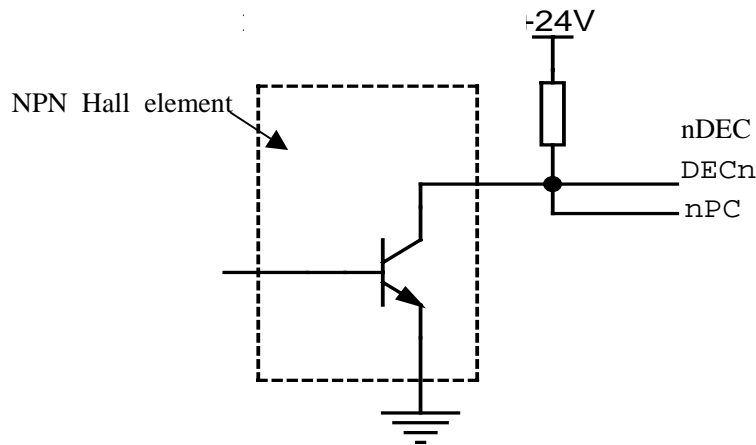


Fig. 2-8 Connection using NPN Hall elements

b) The connection for PNP Hall elements taken as both deceleration signal and zero signal is as Fig. 2-9:

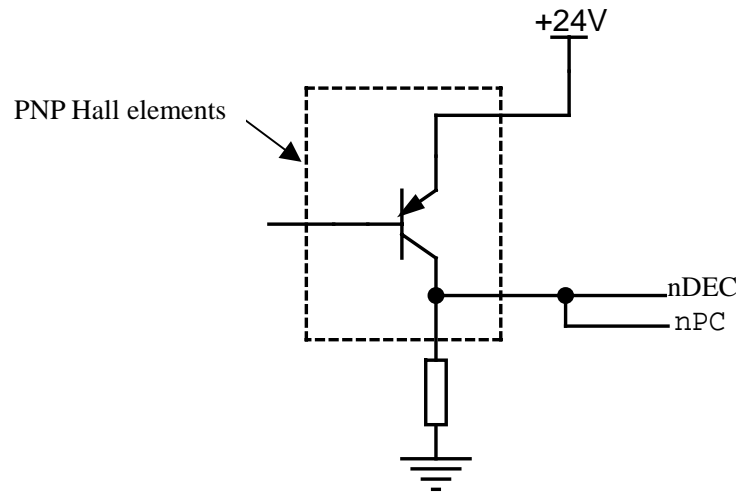
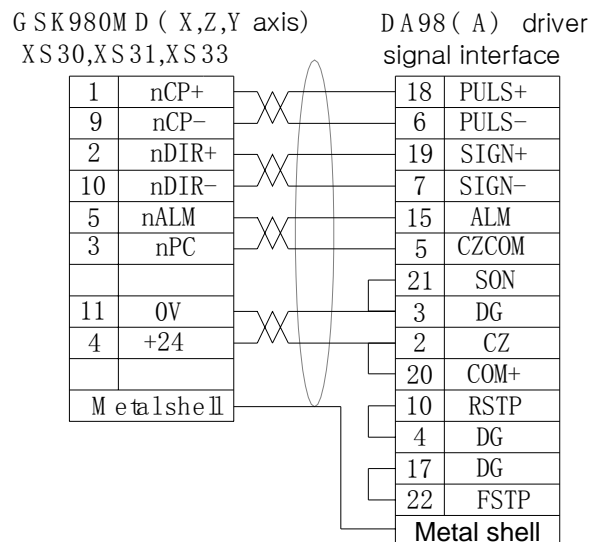


Fig. 2-9 Connection using PNP Hall elements

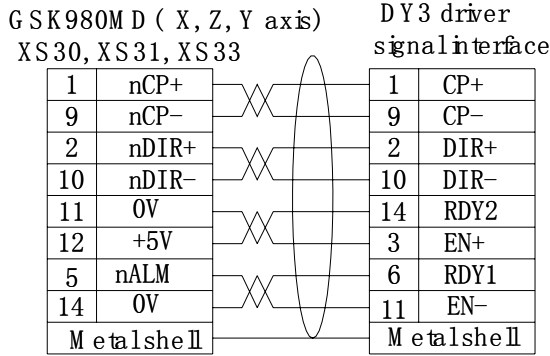
2.1.7 Connection to driver

The connection of GSK 980TD to GSK driver is shown as Fig. 2-10:

Connection of GSK 980M D to DA98 (A) driver



Connection of GSK980MD to DY3 driver



Connection of GSK980MD to DF3 driver

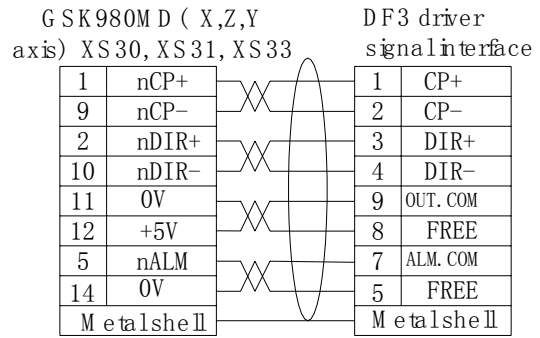


Fig. 2-10 Connection of GSK 980MD to a driver

2.2 Connection to Spindle Encoder

2.2.1 Spindle encoder interface definition

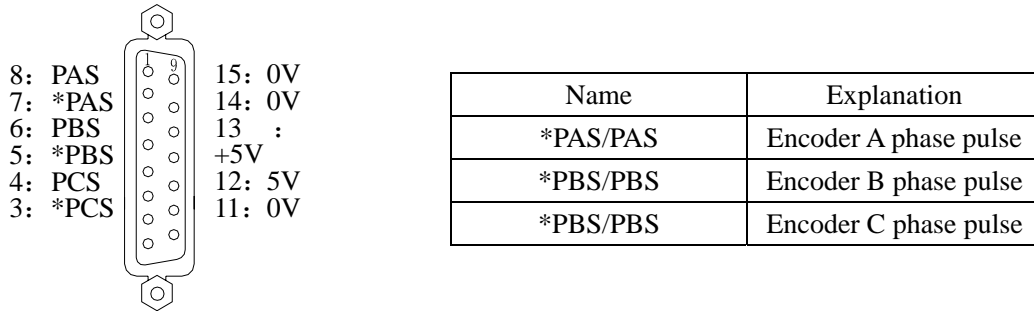


Fig. 2-11 XS32 encoder interface (15-core DB female socket)

2.2.2 Signal explanation

*PCS/PCS, *PBS/PBS, *PAS/PAS are the encoder C, B, A phase differential input signals respectively, which are received by 26LS32; *PAS/PAS, *PBS/PBS are normal square wave of phase shift 90° with the maximum signal frequency less than 1MHz; the encoder pulses for GSK980MD are set by data parameter No.070, whose range is from 100 to 5000.

Its interior connection circuit is as Fig. 2-12: (n=A, B, C)

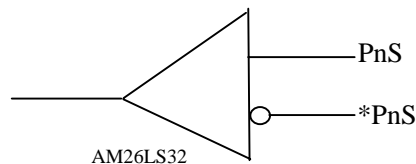


Fig. 2-12 Encoder signal circuit

2.2.3 Connection of spindle encoder interface

The connection of GSK980MD to spindle encoder is shown as Fig. 2-13, use twisted pair cables for connection.

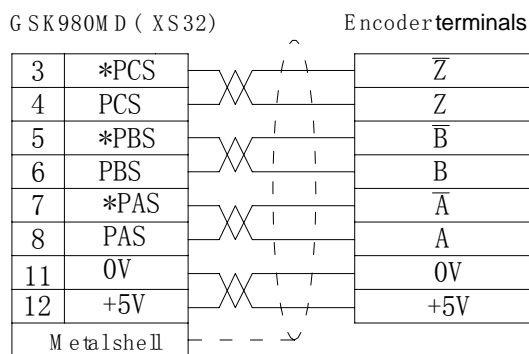
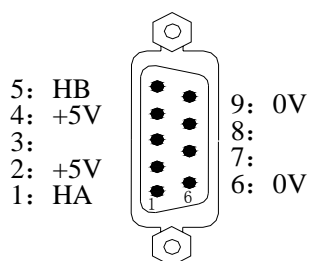


Fig. 2-13 Connection of GSK980MD to encoder

2.3 Connection to Handwheel

2.3.1 Handwheel interface definition



Signal	Explanation
HA	Handwheel A phase signal
HB	Handwheel B phase signal
+5V, 0V	DC power supply

2.3.2 Signal explanation

HA, HB are the handwheel A, B phase input signals respectively. Their interior circuit is shown as Fig. 2-15:

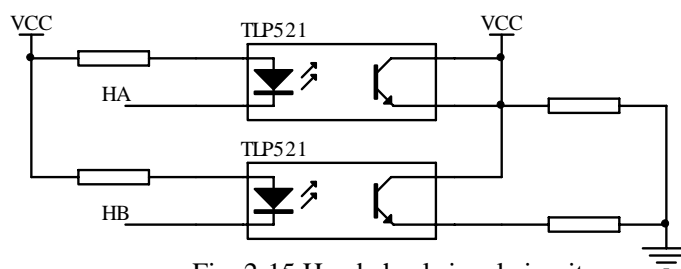


Fig. 2-15 Handwheel signal circuit

The connection of GSK980MD to handwheel is as Fig. 2-16:

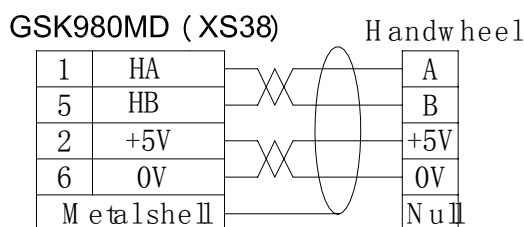
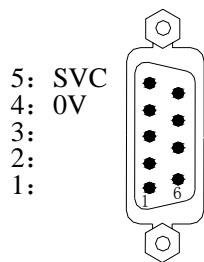


Fig. 2-16 Connection of GSK980MD to handwheel

2.4 Connection to Transducer

2.4.1 Analog spindle interface definition



Signal	Explanation
SVC	0~10V analog voltage
0V	Signal grounding

Fig. 2-17 XS37 analog spindle interface (9-core DB pin)

2.4.2 Signal explanation

The analog spindle interface SVC can output 0~10V voltage, its interior signal circuit is as Fig. 2-18:

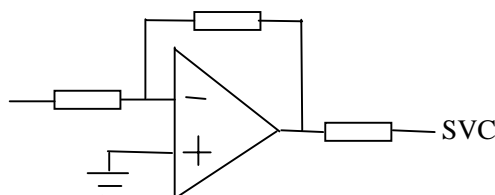


Fig. 2-18 SVC signal circuit

2.4.3 Interface connection of transducer

The connection of GSK980MD to transducer is as Fig. 2-19:

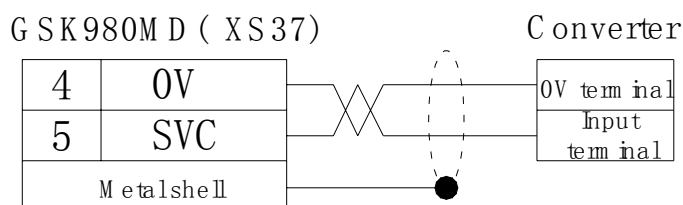
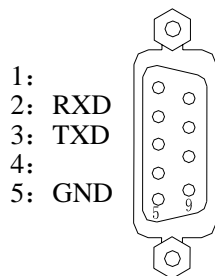


Fig. 2-19 Connection of GSK980MD to transducer

2.5 Connection of GSK980MD to PC

2.5.1 Communication interface definition



Signal	Explanation
RXD	For data reception
TXD	For data transmitting
GND	For signal grounding

Fig. 2-20 XS36 communication interface
(9-core DB female socket)

2.5.2 Communication interface connection

The communication between GSK980MD and PC can be done via RS232 interface (GSK980MD communication software needed). The connection of them is shown as Fig. 2-21:

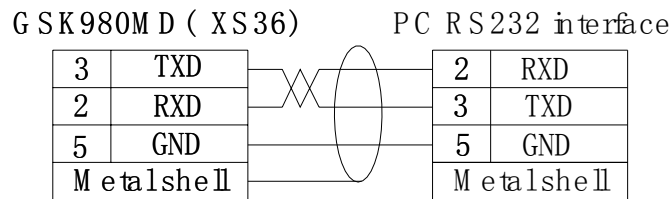


Fig. 2-21 Connection of GSK980MD to PC

The communication of a GSK980MD to another GSK980MD can be made via their XS36 interfaces, and the connection of them is shown as Fig. 2-22:

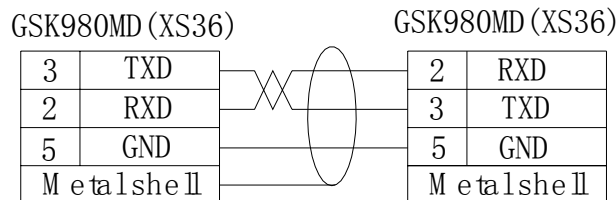


Fig. 2-22 Communication connection of GSK980MD to GSK980MD

2.6 Connection of Power Interface

GSK-PB2 power box is applied in this GSK980MD, which involves 4 groups of voltage: +5V(3A), +12V(1A), -12V (0.5A) , +24V (0.5A) , and its common terminal is COM (0V) . The connection of GSK-PB2 power box to GSK980MD XS2 interface has been done for its supply, and the user only need to connect it to a 220V AC power for use.

The XS2 interface definition of GSK980MD is as following Fig. 2-23:

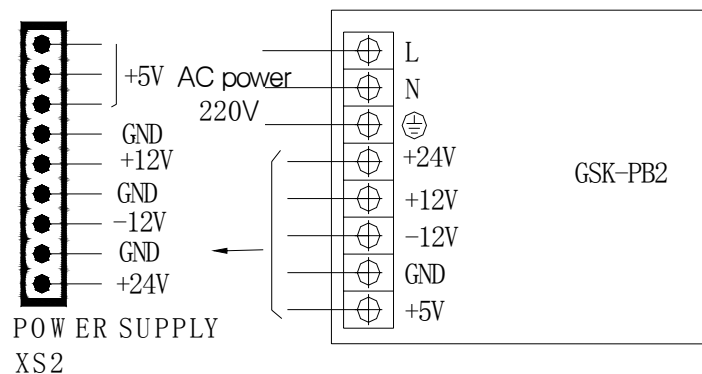
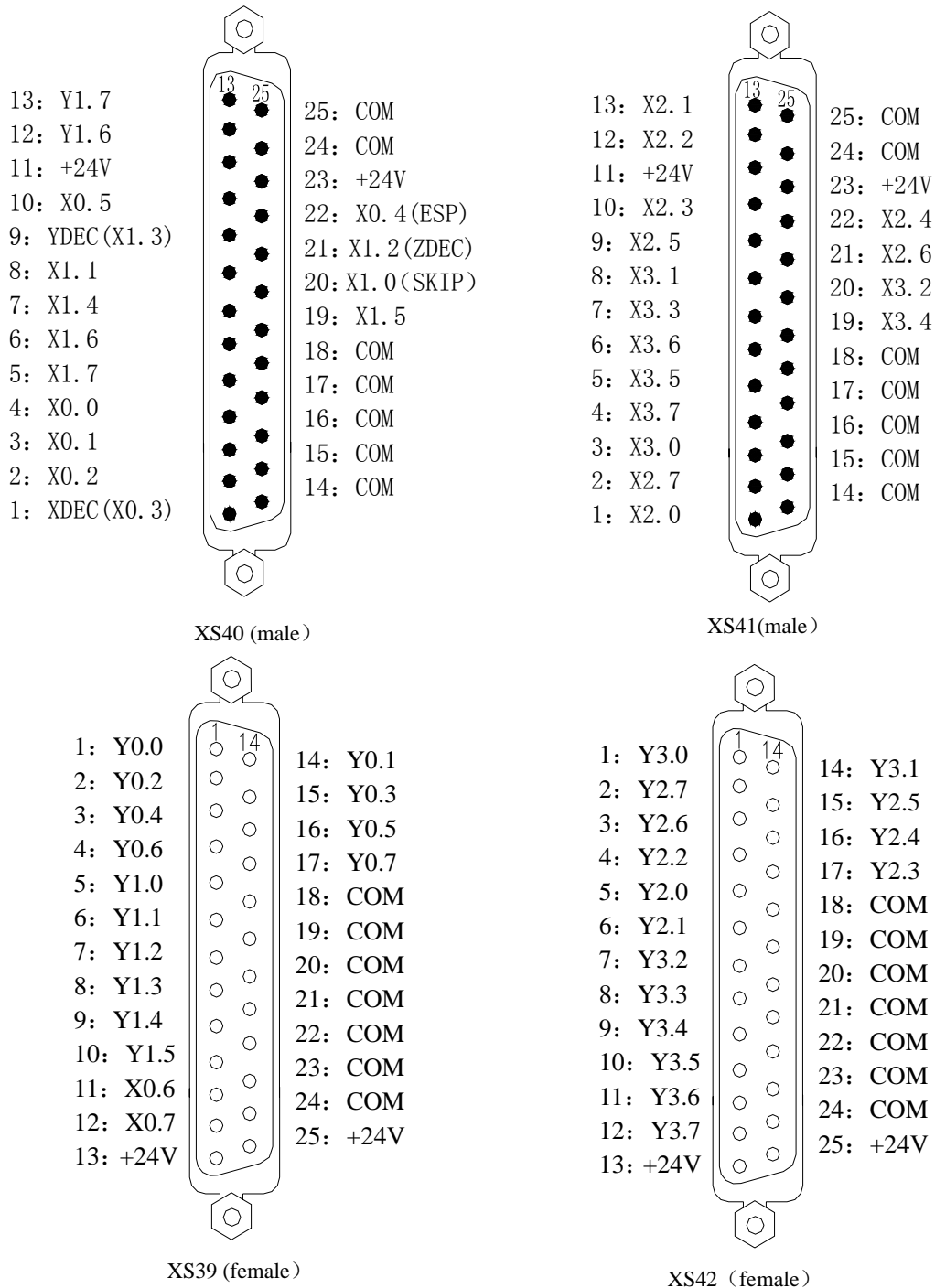


Fig. 2-23

2.7 I/O interface definition



Note 1: The I/O function of GSK980MD milling CNC is defined by ladder, see details about it in GSK980MD standard ladder configuration.

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 impedance.

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 names.

Note 5: XDEC, YDEC, ZDEC, ESP, SKIP are fixed signals that can't be altered.

2.7.1 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 trigger point of input signal on machine should meet the following conditions:

The capacity of the trigger point: DC30V, 16mA above

Leakage current between trigger points in open circuit: 1mA below

Voltage drop between trigger points in close 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 which is used by the keys on machine, stroke switch and trigger point of relay, as is shown in Fig. 2-24:

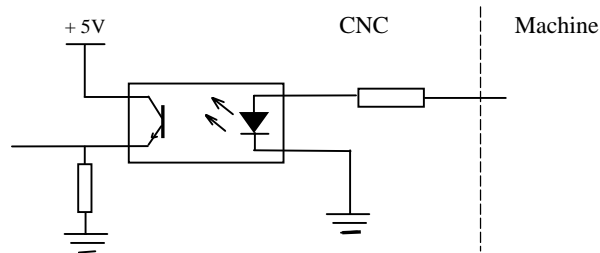


Fig. 2-24

The other type is input by switch with no trigger point (transistor), as is shown in Fig. 2-25, 2-26:

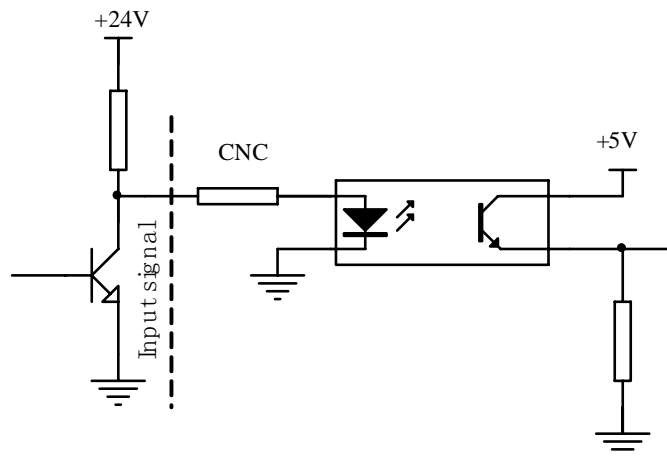


Fig. 2-25 NPN connection

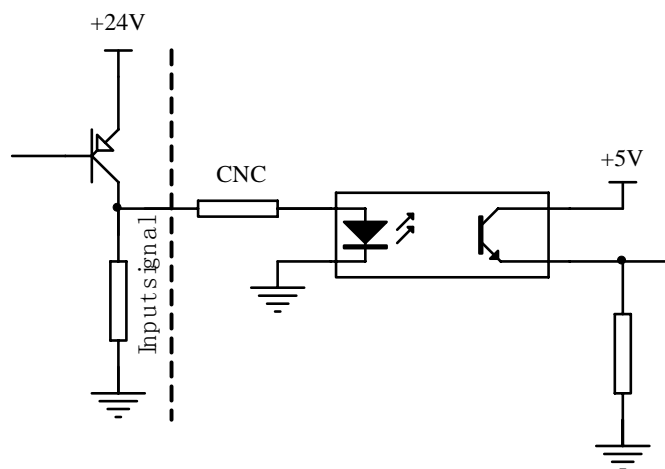


Fig. 2-26 PNP connection

2.7.2 Output signal

The output signal is used for the machine 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. 2-27:

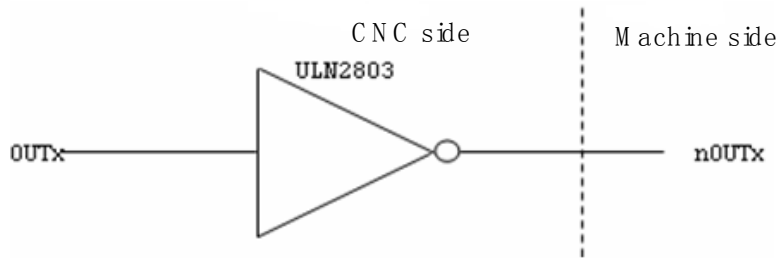


Fig. 2-27 Circuit for digital volume output module

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 as follows:

- To drive LED

A series 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. 2-28:

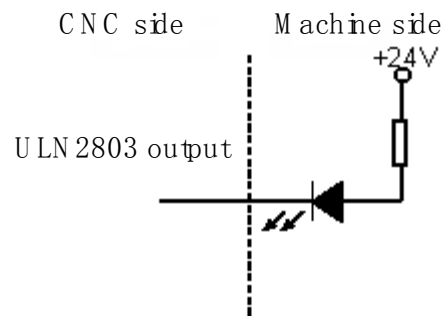


Fig. 2-28

- 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 can't light up. It is shown in Fig. 2-29:

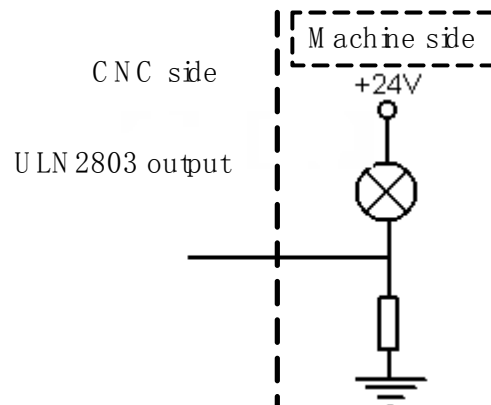


Fig. 2-29

● 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. 2-30:

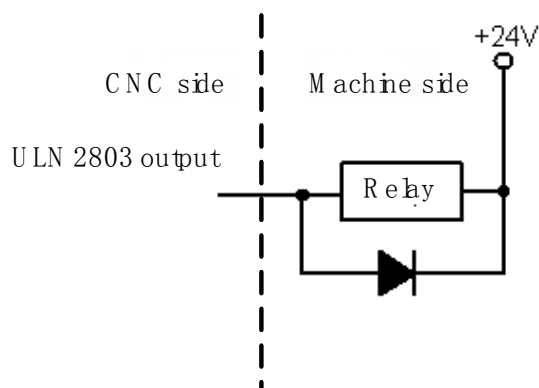


Fig. 2-30

2.8 Machine Zero

● Relative signal

XDEC	X axis deceleration signal		XPC	X axis zero signal
YDEC	Y axis deceleration signal		YPC	Y axis zero signal
ZDEC	Z axis deceleration signal		ZPC	Z axis zero signal

● CNC diagnosis

000	XDEC	YDEC	ZDEC					
Corresponding pin-out	XS40.1	XS40.9	XS40.2					
PLC address	X0.3	X1.3	X1.2					

008						ZPC	YPC	XPC
Corresponding pin-out						XS31.3	XS33.3	XS30.3

● Control parameter

004			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

005							PPD	
-----	--	--	--	--	--	--	-----	--

PPD =1: Relative coordinate set by G92

=0: Relative coordinate not set by G92

006						ZMZ	ZMY	ZMX
-----	--	--	--	--	--	-----	-----	-----

ZMZ =1: Z axis machine zero return type C

=0: Z axis machine zero return type B

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

007						ZCZ	ZCY	ZCX
-----	--	--	--	--	--	-----	-----	-----

ZCZ =1: The deceleration signal (ZDEC) and one-turn signal (PCZ) of Z axis are parallel for machine zero return (an approach switch acting as both the deceleration signal and zero signal);

=0: The deceleration signal (ZDEC) and one-turn signal (PCZ) of Z axis are separated for machine zero return (the deceleration signal and zero signal are separated);


ZCX =1: The deceleration signal (XDEC) and one-turn signal (PCX) of X axis are parallel for machine zero return (an approach switch acting as both the deceleration signal and zero signal);

=0: The deceleration signal (XDEC) and one-turn signal (PCX) of X axis are separated for machine zero return (the deceleration signal and zero signal are separated);

ZCY =1: The deceleration signal (YDEC) and one-turn signal (PCY) of Y axis are parallel for machine zero return (an approach switch acting as both the deceleration signal and zero signal);

=0: The deceleration signal (YDEC) and one-turn signal (PCY) of Y axis are separated for machine zero return (the deceleration signal and zero signal are separated);

011						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.

012								ISOT
-----	--	--	--	--	--	--	--	------

ISOT =1: Manual rapid traverse valid prior to machine zero return after power on

=0: Manual rapid traverse invalid prior to machine zero return after power on

014						ZRSZ	ZRSY	ZRSX
-----	--	--	--	--	--	------	------	------

ZRSZ, ZRSX, ZRSY =1: To select machine zero return type B, C, which have machine zero, it needs to detect deceleration and zero signals as machine zero return is performed;

=0: To select machine zero return type A, which has no machine zero, it directly returns to machine zero without detecting deceleration and zero signals as machine zero return is performed;

183						MZRZ	MZRY	MZRX
-----	--	--	--	--	--	------	------	------

MZRZ, MZRZ, MZRY =1: To select negative zero return of X, Z, Y axes;

=0: To select positive zero return of X, Z, Y axes.

Data parameter

033	Low speed of reference point return of axes
-----	---

075	High speed of reference point return of axes
076	X axis machine zero offset (0.001)
077	Y axis machine zero offset (0.001)
078	Z axis machine zero offset (0.001)
080	X machine coordinate of 1 st reference point (0.001mm)
081	Y machine coordinate of 1 st reference point (0.001mm)
082	Z machine coordinate of 1 st reference point (0.001mm)
084	X machine coordinate of 2 nd reference point (0.001mm)
085	Y machine coordinate of 2 nd reference point (0.001mm)
086	Z machine coordinate of 2 nd reference point (0.001mm)
088	X machine coordinate of 3rd reference point (0.001mm)
089	Y machine coordinate of 3rd reference point (0.001mm)
090	Z machine coordinate of 3rd reference point (0.001mm)
092	X machine coordinate of 4th reference point (0.001mm)
093	Y machine coordinate of 4th reference point (0.001mm)
094	Z machine coordinate of 4th reference point (0.001mm)

● **Signal connection**

The interior wiring circuit of deceleration signal is as Fig. 2-31:

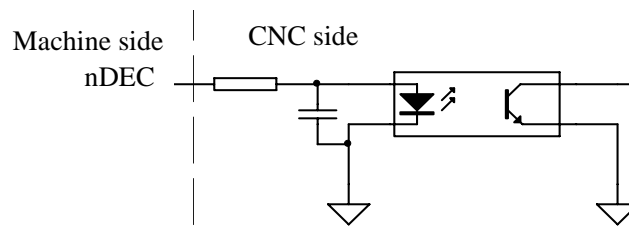


Fig. 2-31

● **Machine zero return type B by regarding servo motor one-turn signal as zero signal**

① Its sketch map is as follows:

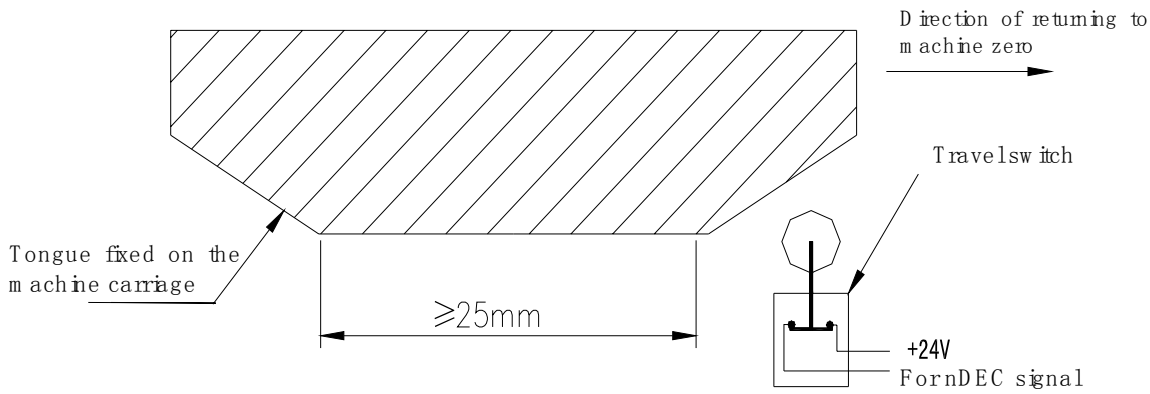


Fig. 2-32

② The circuit of deceleration signal

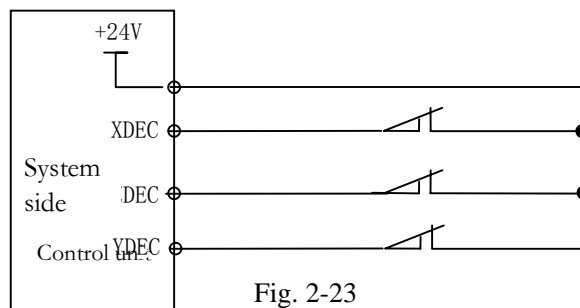


Fig. 2-23

③ Action time sequence of machine zero return

When the BIT0(ZMX), BIT1(ZMY) and BIT2(ZMZ) of the bit parameter No.006 are all set for 0, and the BIT5 (DECI) of the bit parameter No.004 is set for 0, the initial backlash direction of the machine zero return is positive, and the deceleration signal low level is valid. The action time sequence of machine zero return is shown as follows:

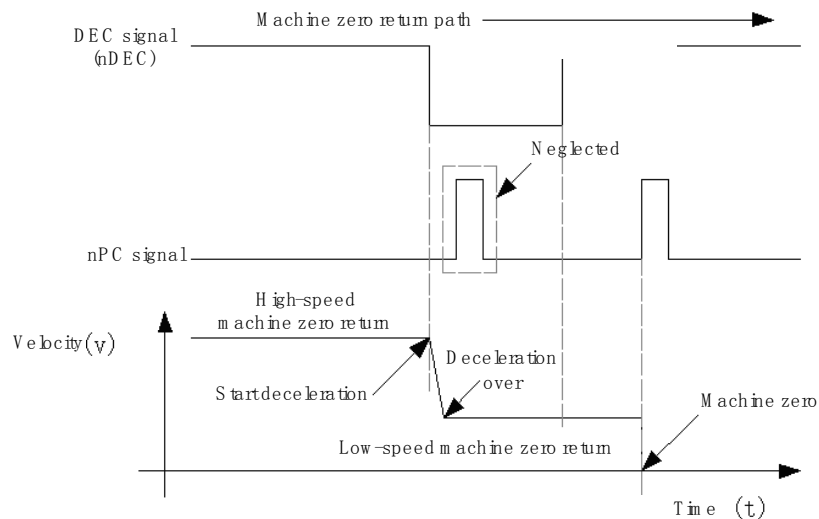


Fig. 2-34

④ Machine zero return process

A: For machine zero return mode, press the manual positive or negative feed key(machine zero return direction set by bit parameter No.183), 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 trigger point is closed again. And CNC begins to detect the encoder one-turn signal (PC), if this signal level changes, the motion will be stopped. And the corresponding zero indicator on the operator panel lights up for machine zero return completion.

● **machine zero return type B as an approach switch is taken as both deceleration and zero signals**

① Its sketch map is as following:

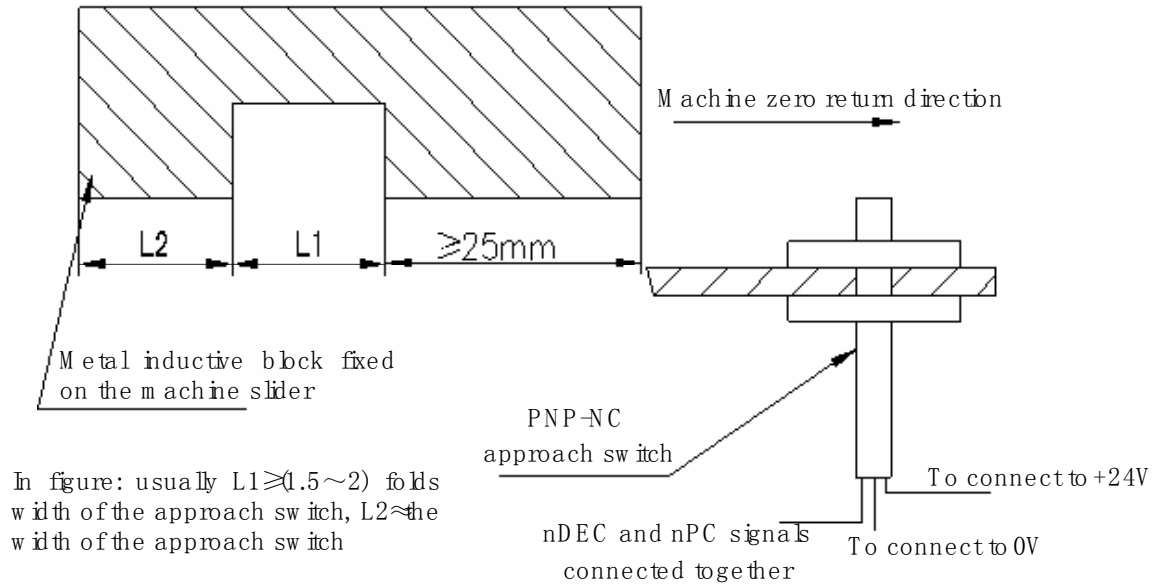


Fig.2-35

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 the BIT0 (ZMX), BIT1 (ZMY) and BIT2 (ZMZ) of the bit parameter No.006 are all set for 0, and the BIT5 (DECI) of the bit parameter No.004 is 0, the action time sequence of zero return is shown as following figure:

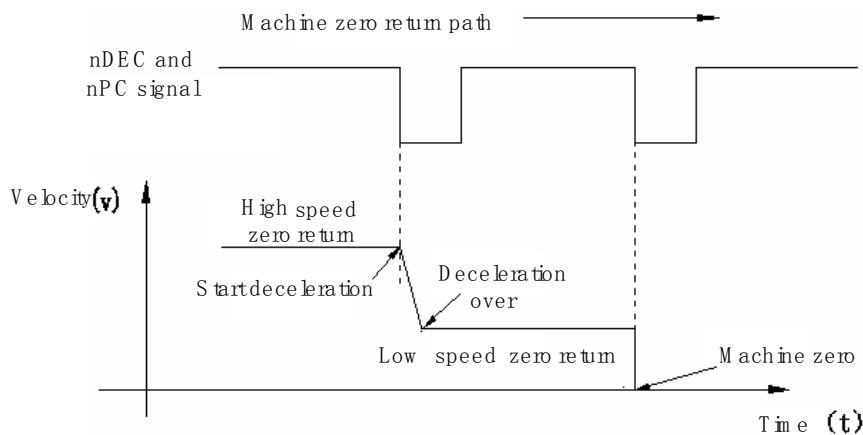


Fig.2-36 Zero return time sequence

④ Machine zero return 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-turn signal taken as zero signal

① Its sketch map is as following:

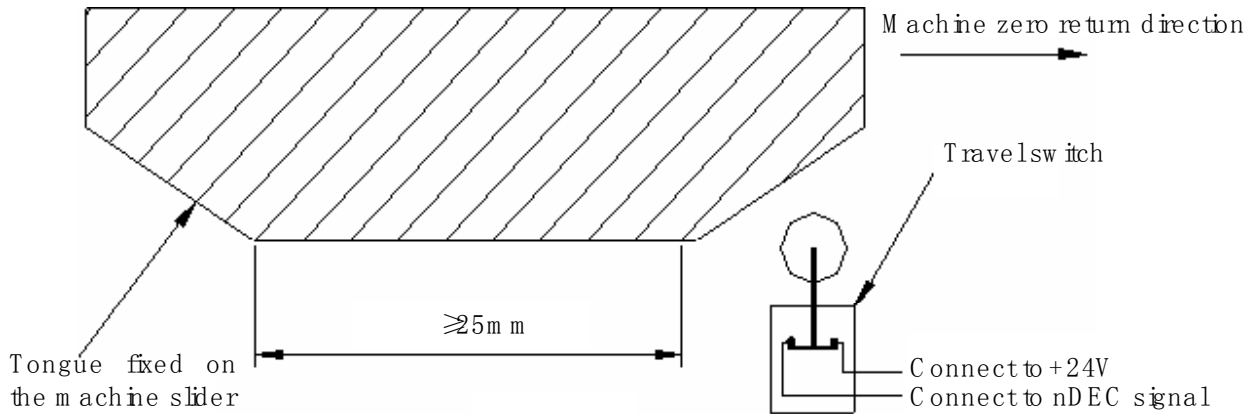


Fig.2-37

② Circuit of the deceleration signal

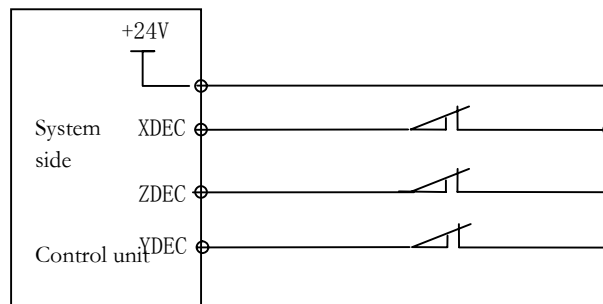


Fig. 2-38

③ Action time sequence of machine zero return

When the BIT0 (ZMX), BIT1 (ZMY) and BIT2 (ZMZ) of the bit parameter No.006 are all set for 1, and the BIT5 (DECI) of the bit parameter No.004 is set for 0, the initial backlash direction of the machine zero return is positive, and the deceleration signal low level is valid. The action time sequence of machine zero return is shown as follows:

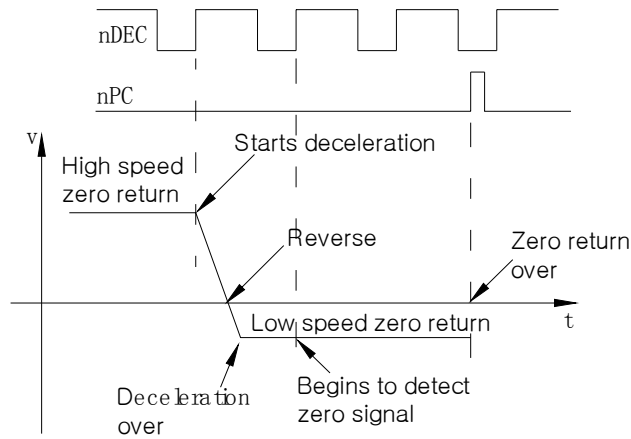


Fig. 2-39

④ Machine zero return 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 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 signal. If the zero signal level changes, the movement stops. Then zero return indicator for the corresponding axis on the panel lights up and machine zero operation finishes.

● Machine zero return type C as an approach switch is taken as both deceleration and zero signals

① Its sketch map is as following:

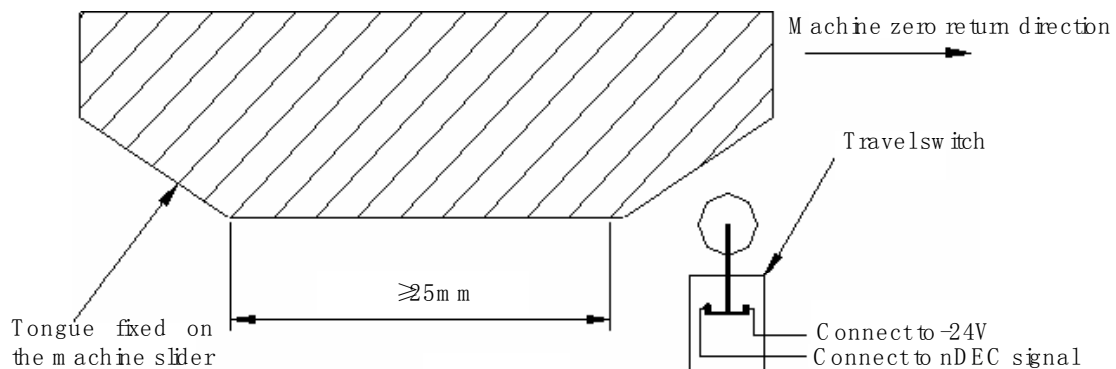


Fig. 2-40

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 the BIT0 (ZMX) , BIT1 (ZMY) and BIT2 (ZMZ) of the bit parameter No.006 are all set for 1, and the BIT5 (DECI) of the bit parameter No.004 is set for 0, the action time sequence of machine zero return is shown as follows:

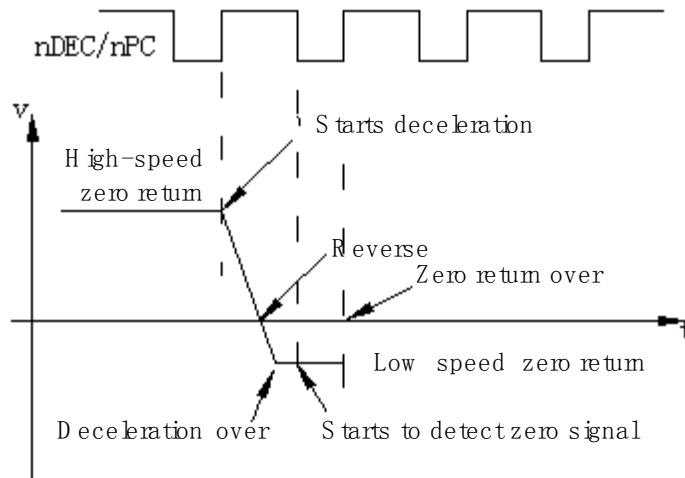


Fig. 2-41

④ Machine zero return 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 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 for the corresponding axis on the panel lights up and machine zero return operation finishes.

CHAPTER 3 PARAMETER

In this chapter the CNC bit and data parameters are introduced, by these parameters various functions can be set.

3.1 Parameter description (by sequence)

3.1.1 Bit parameter

The bit parameter is expressed as following:

Parameter No.	BIT7	BIT6	BIT5	BIT4	BIT3	BIT2	BIT1	BIT0	

0	0	1	***	***	***	Analog spindle	handw heel	***	***	***
---	---	---	-----	-----	-----	-------------------	---------------	-----	-----	-----

Bit3 1: MPG mode

0: Step mode

Bit4 1: Analog voltage control of spindle speed

0: Switching volume control of spindle speed

0	0	2	***	***	RS232	LIFJ	MDITL	LIFC	Offset C	TLIF
---	---	---	-----	-----	-------	------	-------	------	----------	------

Bit0 1: Tool life management valid

0: Tool life management invalid

Bit1 1: Tool nose radius compensation valid

0: Tool nose radius compensation invalid

Bit2 1: Tool life counting type 2 by times

0: Tool life counting type 1 by times

Bit3 1: Tool life management valid in MDI mode

0: Tool life management invalid in MDI mode

Bit4 1: Tool life management group skip valid

0: Tool life management group skip invalid

Bit5 1: RS232 communication valid

0: RS232 communication invalid

0	0	3	***	***	Screw-pitch	***	***	***	D_R	***
---	---	---	-----	-----	-------------	-----	-----	-----	-----	-----

Bit1 1: Tool offset D diameter input;

0: Tool offset D radius input;

Bit5 1: Screw-pitch error compensation valid;

0: Screw-pitch error compensation invalid;

0	0	4	***	RDRN	DECI	***	PROD	DCS	INI	SCW
---	---	---	-----	------	------	-----	------	-----	-----	-----

Bit6 1: G0 rapid traverse speed in dry run mode



0: G0 manual feedrate in dry run mode

Bit5 1: Deceleration signal high level for machine zero return

0: Deceleration signal low level for machine zero return

Bit3 1: Relative programming position display in POSITION page

0: Relative position display involving tool offset in POSITION page

- Bit2 1: Program start by  key valid in MDI mode
 0: Program start by  key invalid in MDI mode
- Bit1 1: Inch input
 0: Metric input
- Bit0 1: Inch output(inch system)valid after repower
 0: Metric output(metric system)valid after repower

0	0	5	***	***	SMAL	M30	***	***	PPD	PCMD
---	---	---	-----	-----	------	-----	-----	-----	-----	------

- Bit5 1: Spindle manual gear shift for S command
 0: Spindle auto gear shift for S command
- Bit4 1: Cursor to beginning after M30 execution
 0: Cursor not to beginning after M30 execution
- Bit1 1: Relative coordinate set by G92
 0: Relative coordinate not set by G92
- Bit0 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	***	***	***	OVRI	***	ZMZ	ZMY	ZMX
---	---	---	-----	-----	-----	------	-----	-----	-----	-----

- Bit4 1: feed override reverse on machine panel
 0: feed override not reverse on machine panel
- Bit2 1: Z zero return type C
 0: Z zero return type B
- Bit1 1: Y zero return type C
 0: Y zero return type B
- Bit0 1: X zero return type C
 0: X zero return type B

0	0	7	DISP	***	***	***	SMZ	ZCZ	ZCY	ZCX
---	---	---	------	-----	-----	-----	-----	-----	-----	-----

- Bit7 1: Enter absolute page after power on
 0: Enter relative page after power on
- Bit3 1: To execute next block till all moving blocks executed
 0: For smooth transition between blocks
- Bit2 1: Deceleration signal (DECZ) and one-turn signal (PCZ) of Z axis parallel (DECZ and zero signals together by an approach switch) during machine zero return
 0: Deceleration signal (DECZ) and one-turn signal (PCZ) of Z axis separate (separate DECZ and zero signal) during machine zero return
- Bit1 1: Deceleration signal (DECY) and one-turn signal (PCY) of Y axis parallel (DECY and zero signals together by an approach switch) during machine zero return
 0: Deceleration signal (DECY) and one-turn signal (PCY) of Y axis separate (separate DECY and zero signal) during machine zero return

- Bit0 1: Deceleration signal (DECX) and one-turn signal (PCX) of X axis parallel (DECZ and zero signals together by an approach switch) during machine zero return
 0: Deceleration signal (DECX) and one-turn signal (PCX) of X axis separate (separate DECX and zero signal) during machine zero return

0	0	8	***	***	***	***	AVGL	DIRY	DIRZ	DIRX
---	---	---	-----	-----	-----	-----	------	------	------	------

Bit3 0: Linear smoothing invalid

1: Linear smoothing valid when 7#3(SMZ)=0, i.e. smoothing transition valid between blocks.

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 differs 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.

Bit2 1: Direction signal (DIR) is high level as Z axis moves positively

0: Direction signal (DIR) is low level as Z axis moves negatively

Bit1 1: Direction signal (DIR) is high level as Y axis moves positively


0: Direction signal (DIR) is low level as Y axis moves negatively

Bit0 1: Direction signal (DIR) is high level as X axis moves positively

0: Direction signal (DIR) is low level as X axis moves negatively

Standard setting: 00000111

0	0	9	***	***	***	***	RSJG	ZALM	YALM	XALM
---	---	---	-----	-----	-----	-----	------	------	------	------

Bit3 0: CNC turns off spindle, lubrication, cooling output when pressing  key.

1: CNC holds on spindle, lubrication, cooling output when pressing  key.

Bit2 1: Z axis low level alarm signal (ZALM)

0: Z axis high level alarm signal (ZALM)

Bit1 1: Y axis low level alarm signal (YALM)

0: Y axis high level alarm signal (YALM)

Bit0 1: X axis low level alarm signal (XALM)

0: X axis high level alarm signal (XALM)

0	1	0	CPF8	CPF7	CPF6	CPF5	CPF4	CPF3	CPF2	CPF1
---	---	---	------	------	------	------	------	------	------	------

Bit0~ Bit7: Setting values of backlash compensation pulse frequency

The set frequency = $(2^7 \times \text{CPF8} + 2^6 \times \text{CPF7} + 2^5 \times \text{CPF6} + 2^4 \times \text{CPF5} + 2^3 \times \text{CPF4} + 2^2 \times \text{CPF3} + 2^1 \times \text{CPF2} + \text{CPF1} + 1)$ Kpps

0	1	1	BDEC	BD8	***	***	***	ZNIK	***	JSPD
---	---	---	------	-----	-----	-----	-----	------	-----	------

Bit7 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 (by bit parameter No.010) or 1/8 of it.

Bit6 1: Backlash compensation is done by the 1/8 of the set frequency

0: Backlash compensation is done by the set frequency

- Bit2 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
- Bit0 1: Spindle JOG valid in any mode
0: Spindle JOG only valid in Manual mode

0	1	2	***	***	***	TMANL	EAL	***	EBCL	ISOT
---	---	---	-----	-----	-----	-------	-----	-----	------	------

- Bit4 1: Manual tool change for T code
0: Auto tool change for T code
- Bit3 1: Program editing allowed during CNC alarming
0: Program editing not allowed during CNC alarming
- Bit1 1: Program end sign EOB displays “;”(semicolon)
0: Program end sign EOB displays “*” (asterisk)
- Bit0 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	SRCD	***	***	***	SKPI	G31P
---	---	---	------	-----	------	-----	-----	-----	------	------

- Bit7 1: Coordinate system holding on at power down
0: Coordinate system power down not holding, power on for G54 coordinate system
- Bit6 1: G01 status when power on
0: G00 status when power on
- Bit5 1: G54 coordinate system when reset
0: Coordinate system not changed when reset
- Bit1 1: High level valid for skip signal
0: Low level valid for skip signal
- Bit0 1: G31 immediately stops when skip signal is valid
0: G31 slows down to stop when skip signal is valid

0	1	4	LPTK	***	***	***	***	ZRSZ	ZRSY	ZRSX
---	---	---	------	-----	-----	-----	-----	------	------	------

- Bit7 1: Serial holes positioning is performed by cutting path(G01~03)
0: Serial holes positioning is performed by rapid traverse path (G00)
- Bit2, Bit1, Bit0 =1: There are machine zeroes in Z, Y, X axes, it detects deceleration signal and zero signal when performing machine zero return
=0: There are no machine zeroes in Z, Y, X axes, it returns to machine zero without detecting deceleration signal and zero signal when performing machine zero return

1	7	2	***	MST	MSP	MOT	ESP	***	***	SOVI
---	---	---	-----	-----	-----	-----	-----	-----	-----	------

- Bit6 1: External cycle start signal (ST) invalid, it is not the cycle start switch and can be defined by macro command.(#1014)
0: External cycle start signal (ST) valid
- Bit5 1: External stop signal (SP) invalid, it is not the stop switch and can be defined by macro command.(#1015)
0: External stop signal (SP) valid with external stop switch connected, otherwise CNC shows “stop” .
- Bit4 1: Not detect software stroke limit
0: Detect software stroke limit
- Bit3 1: Emergency stop invalid
0: Emergency stop valid
- Bit0 1: The respective interface is defined as external override switch signal, 0V1,0V2,0V4,0V8 input signal
0: External override switch signal 0V1,0V2,0V4,0V8 invalid

1	7	3
---	---	---

***	***	***	ESCD	***	***	SINC	SOUS
-----	-----	-----	------	-----	-----	------	------

- Bit4 1: S code off at emergency stop
0: S code not off at emergency stop
- Bit1 1: 0.1 invalid, 0.001,0.01 valid in Step(MPG) mode
0: 0.1, 0.01, 0.001mm valid in Step (MPG) mode
- Bit0 1: S1, S2 valid, S3,S4 invalid as spindle analog voltage is invalid, and the corresponding output interfaces are U02, U03
0: S1~S4 valid as spindle analog voltage is invalid

1	7	4
---	---	---

***	***	***	***	KEY1	***	***	***
-----	-----	-----	-----	------	-----	-----	-----

- Bit3 1: Program switch is "ON"as power on
0: Program switch is "OFF"as power on

1	7	5
---	---	---

***	SAR	***	THDA	SPFD	ZVAL	YVAL	XVAL
-----	-----	-----	------	------	------	------	------

- Bit6 1: Detect spindle SAR signal prior to cutting
0: Not detect spindle SAR signal prior to cutting
- Bit4 1: Threading is exponential acceleration and deceleration
0: Threading is linear acceleration and deceleration
- Bit3 1: Spindle stop is unallowed during cutting, feeding stops if spindle stops, and No. 404 alarm occurs, spindle stops and feeding stops either.
0: Spindle stop is allowed during cutting, and feeding not stop after spindle stop
- Bit2 1: For Z axis move key,↑ is positive, ↓is negative
0: For Z axis move key, ↓is positive, ↑is negative
- Bit1 1: For Y axis move key,↑ is positive, ↓is negative
0: For Y axis move key, ↓is positive, ↑is negative
- Bit0 1: For X axis move key, →is positive, ←is negative
0: For X axis move key, ←is positive, →is negative

1	8	0
---	---	---

***	***	***	***	***	***	***	SPOS
-----	-----	-----	-----	-----	-----	-----	------

- Bit0 1: For DIS TO GO display in POS&PRG
0: For RELATIVE display in POS&WIN

1	8	3
---	---	---

CALH	SOT	***	***	***	MZRZ	MZRY	MZRZ
------	-----	-----	-----	-----	------	------	------

- Bit7 1: Length offset not cancel in reference point return
0: Length offset cancel in reference point return
- Bit6 1: Software limit valid after zero return at power on
0: Software limit valid after power on
- Bit2 1: Machine zero return in negative Z axis
0: Machine zero return in positive Z axis
- Bit1 1: Machine zero return in negative Y axis
0: Machine zero return in positive Y axis
- Bit0 1: Machine zero return in positive X axis
0: Machine zero return in negative X axis

1	8	4	***	PTEST	***	***	***	L2	L1	L0
---	---	---	-----	-------	-----	-----	-----	----	----	----

Bit6 1: Interface auto detection valid (CNC repower needed)

0: Interface auto detection invalid

Bit0, Bit1, Bit2: Interface language selection

Language	Bit2	Bit1	Bit0
Chinese	0	0	0
English	0	0	1
French	0	1	0
Spanish	0	1	1
German	1	0	0
Italian	1	0	1
Russian	1	1	0
Korean	1	1	1

3.1.2 Data parameter

0	1	5	CMRX: X axis multiplier coefficient
0	1	6	CMRY: Y axis multiplier coefficient
0	1	7	CMRZ: Z axis multiplier coefficient

Setting range: 1~32767

0	1	8	CMDX: X axis frequency division coefficient
0	1	9	CMDY: Y axis frequency division coefficient
0	2	0	CMDZ: Z 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

0	2	1	Voltage offset value when spindle max. speed analog voltage 10V output
---	---	---	--

Setting range: -2000~2000 (unit: mV)

0	2	2	RPDFX: X axis max. rapid traverse speed
0	2	3	RPDFY: Y axis max. rapid traverse speed
0	2	4	RPDFZ: Z axis max. rapid traverse speed

Setting range: 10~99999999 (unit: mm/min)

0	2	5	LINTX: Acceleration&deceleration time constant of X axis rapid traverse (ms)
0	2	6	LINTY: Acceleration&deceleration time constant of Y axis rapid traverse (ms)
0	2	7	LINTZ: Acceleration&deceleration time constant of Z axis rapid traverse (ms)

Setting range: 0~4000 (unit: ms)

Chapter 3 Parameter

0 2 8	THDFL: Threading axes start speed(mm/min)
Setting range:6~8000 (unit: mm/min)	
0 2 9	FEEDT: Exponential acceleration&deceleration time constant of cutting and manual feed
Setting range:0~4000 (unit: ms)	
0 3 0	FEDFL: Exponential acceleration start speed and deceleration end speed in cutting feed
Setting range:0~8000 (unit: mm/min)	
0 3 1	FEDM: Axes top feedrate of cutting
Setting range:10~8000 (unit: mm/min)	
0 3 2	RPDFL: Rapid traverse speed when rapid override is F0
Setting range:6~4000 (unit: mm/min)	
0 3 3	ZRNFL: Low speed of axes machine zero return(mm/min)
Setting range:6~4000 (unit: mm/min)	
0 3 4	BKLX: X axis backlash offset (0.001mm)
0 3 5	BKLY: Y axis backlash offset (0.001mm)
0 3 6	BKLZ: Z axis backlash offset (0.001mm)
Setting range:0~2000 (unit: 0.001mm)	
0 3 7	GRMAX1: Max.spindle speed of 1 st gear when analog voltage output is 10V (rpm)
0 3 8	GRMAX2: Max.spindle speed of 2 nd gear when analog voltage output is 10V (rpm)
0 3 9	GRMAX3: Max.spindle speed of 3 rd gear when analog voltage output is 10V (rpm)
0 4 0	GRMAX4: Max.spindle speed of 4 th gear when analog voltage output is 10V (rpm)
Setting range:10~9999 (unit: r/min)	
0 4 1	JOGFL: Exponential ac-deceleration start speed and deceleration end speed in manual feed
Setting range:0~8000 (unit: mm/min)	
0 4 2	SEQINC:Block No. increment for block No.auto insertion
Setting range:1~100	
0 4 3	SPDLC: Voltage compensation for 0V analog voltage output (mV)
Setting range:-1000~1000 (unit: mV)	

0 4 4

BRATE0: Serial communication baudrate

Setting range:1200, 2400, 4800, 9600, 19200, 38400 57600 115200 (unit: bit/s)

0 4 5

LT1X1: Max. X coordinate value of software limit

0 4 6

LT1Y1: Max. Y coordinate value of software limit

0 4 7

LT1Z1: Max. Z coordinate value of software limit

0 4 8

LT1X2: Min. X coordinate value of software limit

0 4 9

LT1Y2: Min. Y coordinate value of software limit

0 5 0

LT1Z2: Min. Z coordinate value of software limit

Setting range:-9999999~+9999999 (unit: 0.001mm)

0 5 1

Retraction amount of G73 high speed peck drilling cycle

Setting range:0~1000 (mm)

0 5 2

Start point of G73 high speed peck drilling cycle

Setting range:0~1000 (mm)

0 5 3

Initial value of cutting feedrate when power on

Setting range:0~100 (mm)

0 5 5

TMAX: Total tool number selection

Setting range:1~32

0 5 6

RESET_TIME: Reset output time

Setting range:16~4080 (ms)

0 5 7

SAR_DELEY: Delay of spindle speed in-position signal detection

Setting range:0~4080 (ms)

0 6 0

PECORGX: Screw-pitch error compensation number of X axis machine zero

0 6 1

PECORGY: Screw-pitch error compensation number of Y axis machine zero

0 6 2

PECORGZ: Screw-pitch error compensation number of Z axis machine zero

Setting range:0~255

0 6 4

PECINTX: Interval of X axis screw-pitch error compensation

0 6 5

PECINTY: Interval of Y axis screw-pitch error compensation

0 6 6

PECINTZ: Interval of Z axis screw-pitch error compensation

Setting range:10000~999999 (0.001mm)

0	6	9
---	---	---

SPD_ERROR_VAR: Max. spindle speed fluctuation allowed by system

Setting range:0~50 (r/min)

0	7	0
---	---	---

ENCODER_CNT: spindle encoder pulses/rev

Setting range:0~5000 (unit: p/r)

0: Not detect spindle encoder in G74, G84 tapping.

0	7	1
0	7	2

ENCODER_MAIN_GEAR: Transmission ratio of encoder and spindle- spindle gear teeth

ENCODER_SLAVE_GEAR: Transmission ratio of encoder and spindle- encoder gear teeth
--

Setting range:1~255

0	7	5
---	---	---

REF_SPEED: High speed of axes machine zero return

Setting range:0~7600(mm/min)

0	7	6
0	7	7
0	7	8

REF_OFFSETX: X axis machine zero offset

REF_OFFSETY: Y axis machine zero offset

REF_OFFSETZ: Z axis machine zero offset

Setting range:-99999~99999(0.001mm)

0	8	0
0	8	1
0	8	2
0	8	4
0	8	5
0	8	6
0	8	8
0	8	9
0	9	0
0	9	2
0	9	3
0	9	4

REF1_COORDX: X machine coordinate of 1 st reference point
--

REF1_COORDY: Y machine coordinate of 1 st reference point
--

REF1_COORDZ: Z machine coordinate of 1 st reference point
--

REF2_COORDX: X machine coordinate of 2nd reference point
--

REF2_COORDY: Y machine coordinate of 2nd reference point
--

REF2_COORDZ: Z machine coordinate of 2nd reference point
--

REF3_COORDX: X machine coordinate of 3rd reference point
--

REF3_COORDY: Y machine coordinate of 3rd reference point
--

REF3_COORDZ: Z machine coordinate of 3rd reference point
--

REF4_COORDX: X machine coordinate of 4th reference point
--

REF4_COORDY: Y machine coordinate of 4th reference point
--

REF4_COORDZ: Z machine coordinate of 4th reference point
--

Setting range:-99999999~99999999 (0.001mm)

0	9	7
---	---	---

G110,G111,G134,G135 Lead of helical cutting (0.001mm)

Setting value 0, 99999999

Standard setting: 500

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.

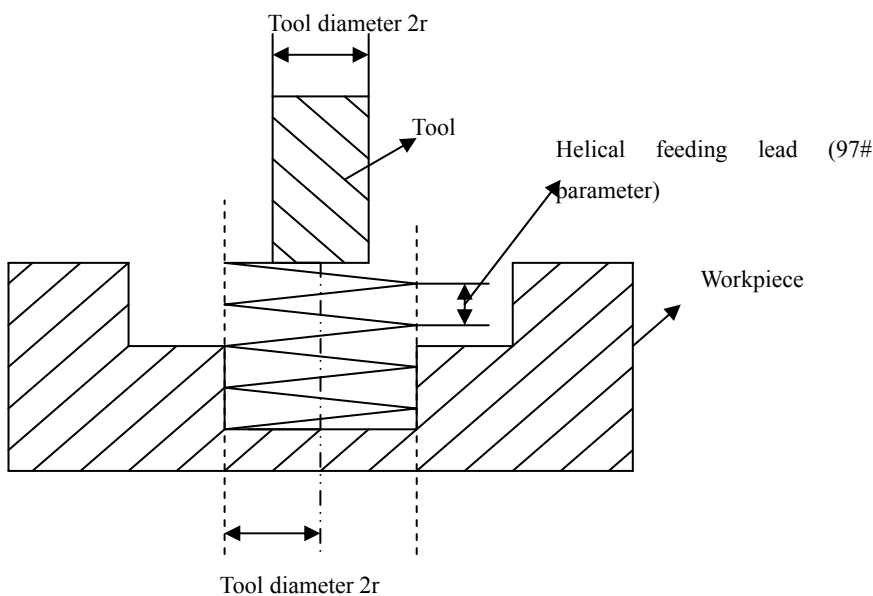
Rough milling command (G110,G111,134,G135) helical feed function:

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 directly.

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 as following:



1	0	0
---	---	---

Setting value 0,100

Standard setting: 20

In the circular command (G02,G03), it is the travel limit of the difference of the start point radius and end point radius.

If error exceeds this value, alarm is issued.

Arc radius error limit (0.001mm)

1	3	9
---	---	---

KEY_P_NUM: Key number valid at the same time
--

3.2 Parameter description (by function sequence)

3.2.1 Axis control logic

0	0	4	***	RDRN	***	***	***	***	***	***	SCW
----------	----------	----------	-----	-------------	-----	-----	-----	-----	-----	-----	------------

- Bit6 1: Rapid traverse speed for G0 in Dry run mode
 0: Manual feedrate for G0 in Dry run mode
- Bit0 1: Inch system for min. command unit, valid after repower
 0: Metric system for min. command unit, valid after repower

0	0	5	***	***	***	***	***	***	***	***	PCMD
----------	----------	----------	-----	-----	-----	-----	-----	-----	-----	-----	-------------

- Bit0 1: Axis output wave form is pulse.
 0: Axis output wave form is square.



Square output, max. output frequency 266KPPS



Pulse output, max. output frequency 266KPPS,
 Pulse width 1 μs.

0	0	6	***	***	***	OVRI	***	***	***	***
----------	----------	----------	-----	-----	-----	-------------	-----	-----	-----	-----

- Bit4 1: Machine panel feedrate override reversed
 0: Machine panel feedrate override not reversed

0	0	7	***	***	***	***	SMZ	***	***	***
----------	----------	----------	-----	-----	-----	-----	------------	-----	-----	-----

- Bit3 1: Execute next block till all moving blocks are executed precisely
 0: For smooth transition between blocks

0	0	8	***	***	***	***	AVGL	DIRY	DIRZ	DIRX
----------	----------	----------	-----	-----	-----	-----	-------------	-------------	-------------	-------------

- Bit3 1: Linear smoothing is valid
 0: Linear smoothing is invalid
- Bit2 1: High level for direction signal(DIR) in Y axis positive moving
 0: Low level for direction signal(DIR) in Y axis negative moving
- Bit1 1: High level for direction signal(DIR) in Z axis positive moving
 0: Low level for direction signal(DIR) in Z axis negative moving
- Bit0 1: High level for direction signal(DIR) in X axis positive moving
 0: Low level for direction signal(DIR) in X axis negative moving

0	0	9	***	***	***	***	***	ZALM	YALM	XALM
----------	----------	----------	-----	-----	-----	-----	-----	-------------	-------------	-------------

- Bit2 1: Low level for Z axis alarm signal (ZALM)
 0: High level for Z axis alarm signal (ZALM)
- Bit1 1: Low level for Y axis alarm signal (YALM)
 0: High level for Y axis alarm signal (YALM)
- Bit0 1: Low level for X axis alarm signal (XALM)
 0: High level for X axis alarm signal (XALM)

1	7	5	***	SAR	***	***	SPFD	ZVAL	YVAL	XVAL
---	---	---	-----	-----	-----	-----	------	------	------	------

- Bit6 1: Detect spindle SAR signal before cutting
0: Not detect spindle SAR signal before cutting
- Bit3 1: Spindle stop is unallowed during cutting, feeding stops if spindle stops, and No. 404 alarm occurs, spindle in cutting stops and feeding stops either.
0: Spindle stop is allowed during cutting, and feeding not stop after spindle stop
- Bit2 1: For Z axis move key, ↑ is positive, ↓ is negative
0: For Z axis move key, ↓ is positive, ↑ is negative
- Bit1 1: For Y axis move key, ↑ is positive, ↓ is negative
0: For Y axis move key, ↓ is positive, ↑ is negative
- Bit0 1: For X axis move key, → is positive, ← is negative
0: For X axis move key, ← is positive, → is negative

0	1	5	CMRX: X axis multiplier coefficient
0	1	6	CMRY: Y axis multiplier coefficient
0	1	7	CMRZ: Z axis multiplier coefficient

Setting range: 1~32767

0	1	8	CMDX: X axis frequency division coefficient
0	1	9	CMDY: Y axis frequency division coefficient
0	2	0	CMDZ: Z axis frequency division coefficient

Setting range: 1~32767

0	2	2	RPDFX: X axis max. rapid traverse speed
0	2	3	RPDFY: Y axis max. rapid traverse speed
0	2	4	RPDFZ: Z axis max. rapid traverse speed

Setting range: 10~99999999 (unit: mm/min)

3.2.2 Acceleration&deceleration control

0	2	9	FEEDT: Exponential acceleration&deceleration time constant of cutting and manual feed
---	---	---	---

Setting range: 0~4000 (unit: ms)

0	3	0	FEDFL: Exponential acceleration start speed and deceleration end speed in cutting feed
---	---	---	--

Setting range: 0~8000 (unit: mm/min)

0	2	5	LINTX: Acceleration&deceleration time constant of X axis rapid traverse (ms)
0	2	6	LINTY: Acceleration&deceleration time constant of Y axis rapid traverse (ms)
0	2	7	LINTZ: Acceleration&deceleration time constant of Z axis rapid traverse (ms)

Setting range: 0~4000 (unit: ms)

0	2	9
---	---	---

FEEDT: Exponential acceleration&deceleration time constant of cutting and manual feed

Setting range:0~4000 (unit: ms)

0	3	0
---	---	---

FEDFL: Exponential acceleration start speed and deceleration end speed in cutting feed

Setting range:0~8000 (unit: mm/min)

3.2.3 Machine protection

1	7	2
---	---	---

***	MST	MSP	MOT	ESP	***	***	***
-----	-----	-----	-----	-----	-----	-----	-----

Bit6 1: External cycle start signal (ST) invalid, it is not the cycle start switch and can be defined by macro command.(#1014)

0: External cycle start signal (ST) valid

Bit5 1: External stop signal (SP) invalid, it is not the stop switch and can be defined by macro command.(#1015)

0: External stop signal (SP) valid with external stop switch connected, otherwise CNC shows “stop” .

Bit4 1: Not detect software stroke limit

0: Detect software stroke limit

Bit3 1: Emergency stop invalid

0: Emergency stop valid

1	7	3
---	---	---

***	***	***	ESCD	***	***	***	***
-----	-----	-----	------	-----	-----	-----	-----

Bit4 1: S code off at emergency stop

0: S code not off at emergency stop

1	8	3
---	---	---

***	SOT	***	***	***	***	***	***
-----	-----	-----	-----	-----	-----	-----	-----

Bit6 1: Software limit valid after power on zero return

0: Software limit valid after power on

0	4	5
0	4	6
0	4	7
0	4	8
0	4	9
0	5	0

LT1X1: Max. X coordinate value of software limit
LT1Y1: Max. Y coordinate value of software limit
LT1Z1: Max. Z coordinate value of software limit
LT1X2: Min. X coordinate value of software limit
LT1Y2: Min. Y coordinate value of software limit
LT1Z2: Min. Z coordinate value of software limit

Setting range:-9999999~+9999999 (unit: 0.001mm)

3.2.4 Thread function

1	7	5
---	---	---

***	***	***	THDA	***	***	***	***
-----	-----	-----	------	-----	-----	-----	-----

Bit4 1: Exponential acceleration&deceleration for threading

0: Linear acceleration&deceleration for threading

0	2	8	THDFL: Threading axes start speed(mm/min)
---	---	---	---

Setting range:6~8000 (unit: mm/min)

3.2.5 Spindle control

0	0	1	***	***	***	Analog spindle	***	***	***	***
---	---	---	-----	-----	-----	----------------	-----	-----	-----	-----

Bit4 1: Analog voltage control of spindle speed
0: Switching volume control of spindle speed

0	0	5	***	***	SMAL	***	***	***	***	***
---	---	---	-----	-----	------	-----	-----	-----	-----	-----

Bit5 1: Spindle manual gear shift for S command
0: Spindle auto gear shift for S command

0	1	1	***	***	***	***	***	***	***	JSPD
---	---	---	-----	-----	-----	-----	-----	-----	-----	------

Bit0 1: Spindle JOG valid in any mode
0: Spindle JOG only valid in Manual mode

0	2	1	Voltage offset value when spindle max. speed analog voltage 10V output
---	---	---	--

Setting range:-2000~2000 (unit: mV)

0	3	7	GRMAX1: Max.spindle speed of 1 st gear when analog voltage output is 10V(rpm)
0	3	8	GRMAX2: Max.spindle speed of 2 nd gear when analog voltage output is 10V
0	3	9	GRMAX3: Max.spindle speed of 3 rd gear when analog voltage output is 10V
0	4	0	GRMAX4: Max.spindle speed of 4 th gear when analog voltage output is 10V

Setting range:10~9999 (unit: r/min)

0	4	3	SPDLC: Voltage compensation for 0V analog voltage output(mV)
---	---	---	--

Setting range:-1000~1000 (unit: mV)

3.2.6 Tool function

0	0	2	***	***	***	LIFJ	MDITL	LIFC	Tool offset C	TLIF
---	---	---	-----	-----	-----	------	-------	------	---------------	------

Bit0 1: Tool life management valid
0: Tool life management invalid
Bit1 1: Tool nose radius compensation valid
0: Tool nose radius compensation invalid
Bit2 1: Tool life counting type 2, by times
0: Tool life counting type 1, by times
Bit3 1: Tool life management valid in MDI mode
0: Tool life management invalid in MDI mode
Bit4 1: Tool life management skip group valid
0: Tool life management skip group invalid

0	1	2	***	***	***	TMANL	***	***	***	***
---	---	---	-----	-----	-----	-------	-----	-----	-----	-----

Bit4 1: Manual tool change for T code
0: Auto tool change for T code

1	8	3	CALH	SOT	***	***	***	MZRZ	MZRY	MARX
---	---	---	------	-----	-----	-----	-----	------	------	------

Bit7 1: Length offset not cancel in reference point return
0: Length offset cancel in reference point return

3.2.7 Edit and display

0	0	4	***	***	***	***	***	***	PROD	***
---	---	---	-----	-----	-----	-----	-----	-----	------	-----

Bit1 1: Relative programming position display in POSITION page
0: Relative position display involving tool offset in POSITION page

0	0	5	***	***	***	M30	***	***	PPD	***
---	---	---	-----	-----	-----	-----	-----	-----	-----	-----

Bit4 1: Cursor to beginning after M30 execution
0: Cursor not to beginning after M30 execution

Bit1 1: Relative coordinate set by G92
0: Relative coordinate not set by G92

0	0	7	DISP	***	***	***	***	***	***	***
---	---	---	------	-----	-----	-----	-----	-----	-----	-----

Bit7 1: Enter absolute page after power on
0: Enter relative page after power on

0	1	2	***	***	***	***	EAL	***	EBCL	***
---	---	---	-----	-----	-----	-----	-----	-----	------	-----

Bit3 1: Program editing allowed during CNC alarming
0: Program editing unallowed during CNC alarming
Bit1 1: Program end sign EOB displays “;”(semicolon)
0: Program end sign EOB displays “*” (asterisk)

1	7	4	***	***	***	***	KEY1	***	***	***
---	---	---	-----	-----	-----	-----	------	-----	-----	-----

Bit3 1: Program switch is “ON” as power on
0: Program switch is “OFF” as power on

1	8	0	***	***	***	***	***	***	***	SPOS
---	---	---	-----	-----	-----	-----	-----	-----	-----	------

Bit0 1: DIST TO GO display in POS&PRG
0: RELATIVE display in POS@WIN

1	8	4	***	***	***	***	***	L2	L1	L0
---	---	---	-----	-----	-----	-----	-----	----	----	----

Bit0, Bit1, Bit2: Interface language selection

Language	Bit2	Bit1	Bit0
Chinese	0	0	0
English	0	0	1
French	0	1	0
Spanish	0	1	1

German	1	0	0
Italian	1	0	1
Russian	1	1	0
Korean	1	1	1

3.2.8 Precision compensation

0	1	0	CPF8	CPF7	CPF6	CPF5	CPF4	CPF3	CPF2	CPF1
---	---	---	------	------	------	------	------	------	------	------

Bit0~ Bit7: Setting values of backlash compensation pulse frequency

The set frequency = $(2^7 \times \text{CPF8} + 2^6 \times \text{CPF7} + 2^5 \times \text{CPF6} + 2^4 \times \text{CPF5} + 2^3 \times \text{CPF4} + 2^2 \times \text{CPF3} + 2^1 \times \text{CPF2} + \text{CPF1} + 1)$ Kpps

0	1	1	BDEC	BD8	***	***	***	***	***	***
---	---	---	------	-----	-----	-----	-----	-----	-----	-----

Bit7 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(by bit parameter No.010) or 1/8 of it.

Bit6 1: Backlash compensation is done by the 1/8 of the set frequency

0: Backlash compensation is done by the set frequency

0	0	3	***	***	Screw-pitch	***	***	***	***	***
---	---	---	-----	-----	-------------	-----	-----	-----	-----	-----

Bit5 1: Screw-pitch error compensation valid

0: Screw-pitch error compensation invalid

0	3	4	BKLX: X axis backlash compensation(0.001mm)							
0	3	5	BKLY: Y axis backlash compensation(0.001mm)							
0	3	6	BKLZ: Z axis backlash compensation(0.001mm)							

Setting range:0~2000 (unit: 0.001mm)

3.2.9 Communication setting

0	0	2	***	***	RS232	***	***	***	***	***
---	---	---	-----	-----	-------	-----	-----	-----	-----	-----

Bit5 1: RS232 communication valid

0: RS232 communication invalid

0	4	4	BRATE0: Serial communication baudrate							
---	---	---	---------------------------------------	--	--	--	--	--	--	--

Setting range: 1200, 2400, 4800, 9600, 19200, 38400 57600 115200 (unit: bit/s)

3.2.10 Machine zero return

0	0	6	***	***	***	***	***	ZMZ	ZMY	ZMX
---	---	---	-----	-----	-----	-----	-----	-----	-----	-----

Bit2 1: Z zero return type C

0: Z zero return type B

Bit1 1: Y zero return type C

0: Y zero return type B

Bit0 1: X zero return type C
0: X zero return type B

0	0	7	***	***	***	***	***	ZCZ	ZCY	ZCX
---	---	---	-----	-----	-----	-----	-----	-----	-----	-----

Bit2 1: Deceleration signal (DECZ) and one-turn signal (PCZ) of Z axis parallel (DECZ and zero signals together by an approach switch) during machine zero return
0: Deceleration signal (DECZ) and one-turn signal (PCZ) of Z axis separate (separate DECZ and zero signal) during machine zero return

Bit1 1: Deceleration signal (DECY) and one-turn signal (PCY) of Y axis parallel (DECY and zero signals together by an approach switch) during machine zero return
0: Deceleration signal (DECY) and one-turn signal (PCY) of Y axis separate (separate DECY and zero signal) during machine zero return

Bit0 1: Deceleration signal (DECX) and one-turn signal (PCX) of X axis parallel (DECZ and zero signals together by an approach switch) during machine zero return
0: Deceleration signal (DECX) and one-turn signal (PCX) of X axis separate (separate DECX and zero signal) during machine zero return

0	1	1	***	***	***	***	***	ZNIK	***	***
---	---	---	-----	-----	-----	-----	-----	------	-----	-----

Bit2 1: Direction keys locked during zero return, zero return continues to end by pressing direction key once
0: Direction keys unlocked but should be held on during zero return

0	1	2	***	***	***	***	***	***	***	ISOT
---	---	---	-----	-----	-----	-----	-----	-----	-----	------

Bit0 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	4	***	***	***	***	***	ZRSZ	ZRSY	ZRSX
---	---	---	-----	-----	-----	-----	-----	------	------	------

Bit2, Bit1, Bit0 =1: There are machine zeroes in Z, Y, X axes, it detects deceleration signal and zero signal when performing machine zero return
=0: There are no machine zeroes in Z, Y, X axes, it returns to machine zero without detecting deceleration signal and zero signal when performing machine zero return.

1	8	3	***	***	***	***	***	MZRZ	MZRY	MARX
---	---	---	-----	-----	-----	-----	-----	------	------	------

Bit2 1: Machine zero return in negative Z axis
0: Machine zero return in positive Z axis

Bit1 1: Machine zero return in negative Y axis
0: Machine zero return in positive Y axis

Bit0 1: Machine zero return in positive X axis
0: Machine zero return in negative X axis

0	3	3	ZRNFL							
---	---	---	-------	--	--	--	--	--	--	--

ZRNFL Low speed of reference point return,FL speed(for all axes)

0	7	5	High speed of machine zero return							
---	---	---	-----------------------------------	--	--	--	--	--	--	--

CHAPTER 4 MACHINE DEBUGGING METHODS AND STEPS

The trial run methods and steps at initial power on for this GSK980MD are described in this chapter. The corresponding operation can be performed after the debugging by the following steps.

4.1 Emergency Stop and Limit

This GSK980MD system has software limit function, it is suggested that the stroke limit switches are fixed in the positive or negative axes for hardware limit. Their connection is shown as following:

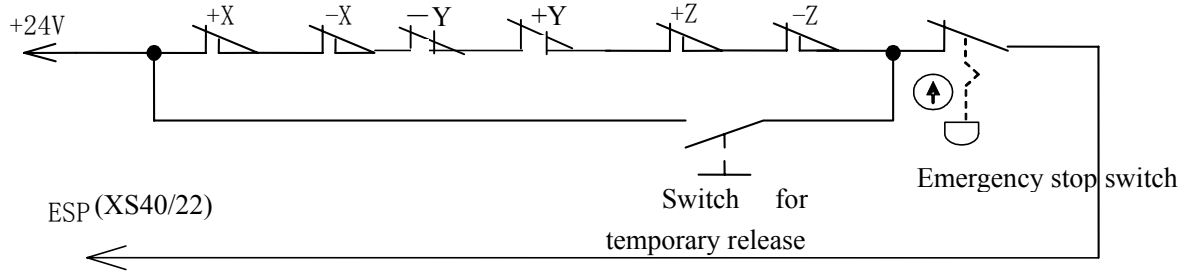


Fig.4-1

So the BIT3 (ESP) of bit parameter No.172 should be set to 0.

And the diagnostic message DGN.000 can monitor the emergency stop input signal.

In Manual or MPG mode, slowly move the axes to testify 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 for reverse moving.

4.2 Driver configuration

The BIT2, BIT1, BIT0 of bit parameter No.009 for our driver are all set for 1 according to the BIT2, BIT1, BIT0 (ZALM, YALM, XALM to Z, Y, X axis respectively) of alarm logic level bit parameter No.009 for driver.

If the machine moving direction is not consistent with the moving command, modify the BIT2, BIT1, BIT0 (DIRZ, DIRY, DIRX to Z, Y, X axis respectively) of bit parameter No.008.

4.3 Gear Ratio Adjustment

The data parameter No.015~No.020 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.

Calculation formula:

$$\frac{CMR}{CMD} = \frac{\delta \times 360}{\alpha \times L} \times \frac{Z_M}{Z_D}$$

CMR: command multiplier coefficient(data parameter No.015, No.016, No.017)

CMD: command frequency division coefficient(data parameter No.018, No.019, No.020)

α : pulse volume, motor rotation angle for a pulse

L: lead

Δ : min. input command unit of CNC(0.001 for all axes of GSK980MD)

Z_M : gear teeth of lead screw

Z_D : gear teeth of motor

Example: If gear teeth of lead is 50, gear teeth of motor is 30, pulse volume $\alpha=0.075^\circ$, screw lead is 4mm, the electronic gear ratio of X axis:

$$\frac{CMR}{CMD} = \frac{\delta \times 360}{\alpha \times L} \times \frac{Z_M}{Z_D} = \frac{0.001 \times 360}{0.075 \times 4} \times \frac{50}{30} = \frac{1}{1}$$

Then data parameter No.015 (CMRX) =1, No.018 (CMDX) =1.

If the electronic gear ratio numerator is more than the denominator, the allowed CNC max. speed will decrease. For example: the data parameter No.017 (CMRZ) =2, No.020 (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: the data parameter No.017 (CMRZ) =1, No.020 (CMDZ) =5, so the pulse is not output as the input increment is 0.004, but a pulse is output if the input increment is 0.005.

In order to ensure the CNC positioning precision and speed index, 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 matched with the CNC system.

When matching with the step drive, choose the driver 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.

4.4 Acceleration&deceleration characteristic adjustment

Adjust the relative CNC parameters according to the factors such as the driver, motor characteristic and machining load:

Data parameter: No.022, No.023, No.024: X, Y, Z axis rapid traverse speed;

Data parameter No.025, No.026, No.027: linear acceleration&deceleration time constant of X, Y, Z axis rapid traverse speed;

Data parameter No.028: axes start/end speed in threading;

Data parameter No.029: Exponential acceleration&deceleration time constant of cutting and manual feeding;

Data parameter No.030: Exponential acceleration&deceleration start/end speed in cutting feeding;

Data parameter No.031: upper limit of axes cutting feedrate;

Data parameter No.032: rapid traverse speed when rapid override is F0;

Data parameter No.041: Exponential acceleration&deceleration start/end speed in manual feeding;

Data parameter No.053: cutting feedrate when power on

Data parameter No.054: axes manual rapid traverse speed

BIT3(SMZ) of bit parameter No.007: for validity of smoothing transition between blocks

The larger the acceleration&deceleration time constant is, the slower the acceleration&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. If the acceleration&deceleration time constant is set too small, and the start/end speed is set too large, it is easily to cause driver alarm, motor out-of-step or machine vibration.

When the bit parameter No.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 next block to obtain an accurate positioning at the path intersection, but this will reduce the machining efficiency. When BIT3=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 driver is applied, the BIT3 of the bit parameter №007 should be set to 1 to avoid the out-of-step.

When the stepper motor driver is applied, 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 as follows (the electronic gear ratio 1:1):

Data parameter №022≤5000	Data parameter №023≤5000
Data parameter №024≤5000	Data parameter №025≥350
Data parameter №026≥350	Data parameter №027≥350
Data parameter №029≥150	Data parameter №028≤100
Data parameter №030≤50	

If AC servo driver is applied, the machining efficiency can be improved by a larger start speed and a smaller acceleration&deceleration time constant setting.

Data parameter №022=10000	Data parameter №023=10000
Data parameter №024=10000	Data parameter №025≤60
Data parameter №026≤60	Data parameter №027≤60
Data parameter №029≤50	Data parameter №028≤500
Data parameter №030≤50	

The parameter setting above is recommended for use, refer to the actual conditions of the driver, motor characteristic and motor load for its proper setting.

4.5 Machine Zero Adjustment

Adjust the relevant parameters based on the valid level of the connection signal, zero return type or direction applied:

BIT5 (DECI) of the bit parameter №004: valid level of deceleration signal as machine zero return

BIT0, BIT1, BIT2 (ZMX, ZMY, ZMZ) of the bit parameter №006: return and initial backlash direction of X, Y, Z axes machine zeroes at deceleration

BIT0, BIT1, BIT2 (ZCX, ZCY, ZCZ) of the bit parameter №007: for an approach switch taken as both deceleration and zero signals

BIT2 (ZNLK) of the bit parameter №011: for direction keys lock when performing zero return

BIT0, BIT1, BIT2 (ZRSCX, ZRSCY, ZRSCZ) of the bit parameter №014: for deceleration and zero signals detection of X, Y, Z axes

Data parameter No.033: low speeds of X, Y, Z axes machine zero return

Data parameter No.075: high speeds of X, Y, Z axes machine zero return

BIT0, BIT1, BIT2 (MZRX, MZRY, MZRZ) of the bit parameter №183: for positive or negative zero turn of X, Y, Z axes

Only the stroke limit switch validity is confirmed, can the machine zero return be performed.

The machine zero is usually fixed at the max. travel point, and the effective stroke 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 suited AC servo motor connection : connection schematic diagram using a travel switch or a servo motor revolution signal repectively

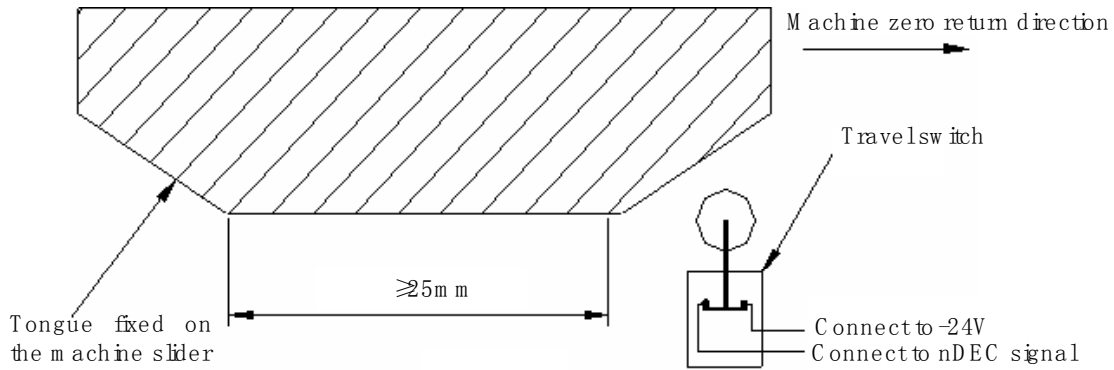


Fig. 4-2

By this connection type, when the deceleration switch is released in machine zero return, the one-turn 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-turn 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)

The parameter setting is as following:

Bit parameter №004 BIT5 (DECI) =0

Bit parameter №006 BIT0 (ZMX) , BIT1 (ZMY) , BIT2 (ZMZ) =0

Bit parameter №007 BIT0 (ZCX) , BIT1 (ZCY) , BIT2 (ZCZ) =0

Bit parameter №011 BIT2 (ZNLK) =1

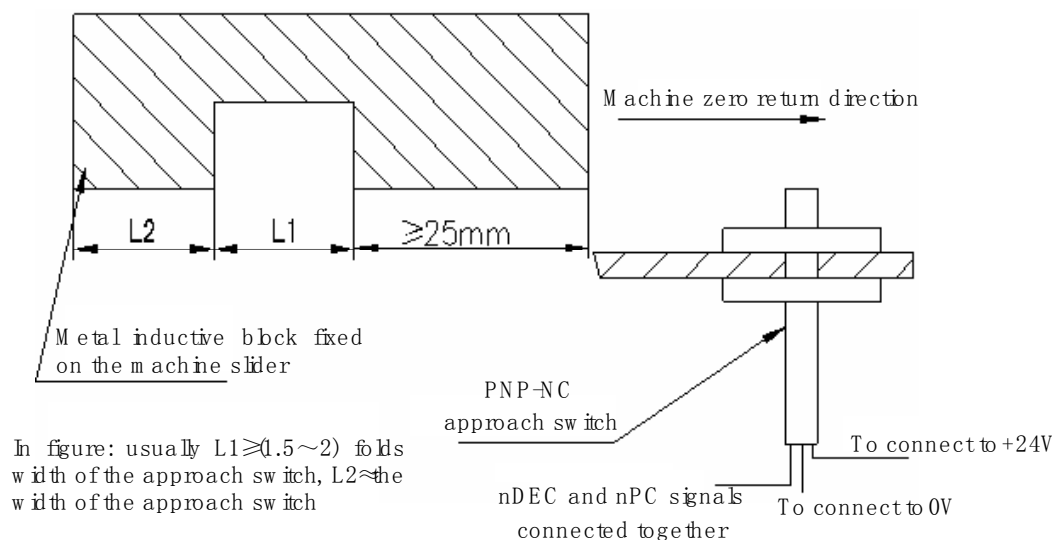
Bit parameter №014 BIT0 (ZRSCX) , BIT1 (ZRSCY) , BIT2 (ZRSCZ) =1

Data parameter №033=200

Data parameter №075=7600

Data parameter №183 BIT0 (MZRX) , BIT1 (MZRY) , BIT2 (MZRZ) =0

2 The connection for suited stepper motor: the schematic diagram using an approach switch taken as both deceleration signal and zero signal



For avoidance of out-of-step at high speed zero return for suited stepper motor, it is better to set the rapid override switch to 50%, and the parameter setting is as following:

Bit parameter №004 BIT5 (DECI) =0

Bit parameter №006 BIT0 (ZMX) , BIT1 (ZMY) , BIT2 (ZMZ) =0

Bit parameter №007 BIT0 (ZCX) , BIT1 (ZCY) , BIT2 (ZCZ) =1

Bit parameter №011 BIT2 (ZNLK) =0

Bit parameter №014 BIT0 (ZRSCX) , BIT1 (ZRSCY) , BIT2 (ZRSCZ) =1

Data parameter №033=200

Data parameter №075=5000

Data parameter №183 BIT0 (MZRX) , BIT1 (MZRY) , BIT2 (MZRZ) =0

The BIT5~BIT7 of diagnosis message are used to check the validity of deceleration zero signal.

The BIT0~BIT2 of diagnosis message are used to check the validity of PC signal.

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.70. The transmission ratio(spindle gears/encoder gears) between encoder and spindle is 1/255~255. The spindle gears are set by CNC data parameter No.071, and the encoder gears by data parameter No.072.

Synchronous belt transmission should be applied for it (no sliding transmission).

The DGN.011 and DNG.012 of diagnosis message are used to check the validity of threading signal from the spindle encoder.

4.6.2 Spindle brake

After M05 code 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.

4.6.3 Switch volume control for spindle speed

When the machine is controlled by a multi-speed motor, the motor speed command is S01~S08. Its relevant parameter is as following:

Bit parameter №001 Bit4=0: for switch volume control of spindle speed

4.6.4 Analog voltage control for spindle speed

This function can be obtained by the parameter setting of CNC. By interface outputting 0V~10V analog voltage to control transducer, the stepless shift can be obtained. And the relative parameters are needed to be adjusted:

Bit parameter №001 Bit4=1: for spindle speed analog voltage control;

Data parameter №021: offset value as spindle speed command voltage is 10V;

Data parameter №043: offset value as spindle speed command voltage is 0V;

Data parameter №037~ №040: for the limit of spindle speed gear 1~4; it defaults the spindle gear 1 when CNC power on.

Basic parameters needed to be adjusted for transducer (refer to the relevant transducer manual for the adjustment)

CCW or CW command mode selection: by common terminal VF;

Frequency setting mode selection: by common terminal FR.

If the speed by programming is not consistent with that detected by the encoder, it can be adjusted to be consistent with the actual one by adjusting the data parameter №037~№040.

Speed adjustment method: select the spindle first gear, input S9999 code in MDI mode to run the spindle, view the spindle speed shown on the right bottom of the screen, then reinput the speed value displayed into the parameter

№037. The other spinle gear adjustment is identical with this.

When entering S9999 code, the voltage should be 10V, S0 for 0V. If there is an voltage error, adjust bit parameter №021 and №043 to correct the voltage offset value(corrected by manufacturer, usually not needed).

When the current gear is the max. speed, if the analog voltage output by CNC is higher than 10V, set a smaller value for data parameter №021; when the S00 code is entered, if there is still slow rotation in the spindle, it means the analog voltage output by CNC is higher than 0V, so set a smaller value for data parameter №043.

If the machine is not fixed with an encoder, the spindle speed can be detected by a speed sensor, input S9999 in MDI mode to set the speed value displayed by sensor to the data parameter №037.

4.7 Backlash Offset

The backlash offset is input by diameter value with the unit 0.001mm, which is irrelevant to the programming by diameter or by radius. It can be measured by a dial indicator, a micrometer or a laser detector. Because the backlash offset can improve the machining precision only by accurate compensation, the backlash offset is not recommended to be measured in the MPG or Step mode, but the following method is suggested:

- Program editing

```
O0001;
N10 G01 Z10 F800 G91 ;
N20 Z15 ;
N30 Z1 ;
N40 Z-1 ;
N50 M30 .
```

- Set the backlash error offset for 0 before measuring:
- Run the program by single blocks, search the measuring benchmark after 2 positioning operations, record the current data, move 1mm in the same direction, then move 1mm reversely to point B, read the current data.

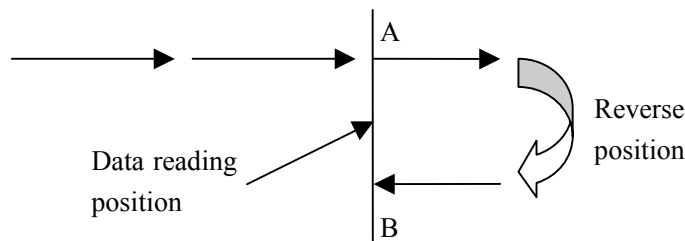


Fig. 4-4 Schematic map of backlash measuring methods

- Backlash error offset value= | data of point A –data of point B |; then input the result to the data parameter №034 (BK LX) ,№035 (BK LY) or №036 (BK LZ) .

Data A: dial meter data at point A

Data B: dial meter data at point B

Note 1: The backlash offset mode and frequency can be set by BIT7 and BIT6 of CNC parameter №011;

Note 2: Check the machine backlash every 3 months' using.

4.8 Step/Handwheel adjustment



The key on the panel can be used to select the Step mode or MPG mode, which is set by the BIT3 of bit parameter №001.

Bit3 =1: MPG mode valid, Step mode invalid;

=0: Step mode valid, MPG mode invalid;

When matching with the step drive, it is better to shield the Step(MPG) 0.1mm increment, which can be selected by BIT1 of bit parameter №173.

1	7	3							SINC	
---	---	---	--	--	--	--	--	--	------	--

SINC =0: Step 0.1, 0.01, 0.001 valid in Step (MPG) mode;

=1: Step 0.1mm invalid, 0.001,0.01mm valid in Step (MPG) mode.

4.9 Other Adjustment

1	7	2		MST	MSP	MOT	ESP			SOVI
---	---	---	--	-----	-----	-----	-----	--	--	------

MST =0: External Cycle Start(ST) signal valid;

=1: External Cycle Start(ST) signal invalid. It is not cycle start switch, and it can be defined by macro(#1014).

MSP =0: External Stop(SP) signal valid. It is must connected with an external stop switch, or “HALT” will be shown by CNC.

=1: External Dwell(SP) signal invalid. It is not dwell switch, and it can be defined by macro(#1015).

MOT =0: Check software limit;

=1: Not check software limit.

SOVI =1: Interfaces defined for the corresponding external override switch 0V1, 0V2, 0V4, 0V8 input signal;

=0: External override switch 0V1, 0V2, 0V4, 0V8 signal invalid.

ESP =1: Not check external ESP signal;

=0: Check external ESP signal.

1	7	3				ESCD			SINC	SOUS
---	---	---	--	--	--	------	--	--	------	------

ESCD =0: S code not off in emergency stop;

=1: S code off in emergency stop.

SINC =0: Step 0.1, 0.01, 0.001mm valid in Step (MPG) mode;

=1: Step 0.1mm invalid, 0.001, 0.01mm valid in Step (MPG) mode.

SOUS =0: S1~S4 valid as spindle analog voltage invalid;

=1: S1, S2 valid, S3, S4 not valid with the corresponding output interfaces U02, U03 as spindle analog voltage invalid.

CHAPTER 5 DIAGNOSIS MESSAGE

5.1 CNC Diagnosis

This diagnosis section is used to check the CNC interface and internal running state that it can't be modified.

5.1.1 Signal diagnosis from machine to CNC

0	0	0	XDEC	YDEC	ZDEC	***	***	SKIP	***	ESP
Pin-out			XS40.1	XS40.21	XS40.9			XS40.20		XS40.10
PLC fixed address			X0.3	X1.3	X1.2			X1.0		X0.5

ESP: Emergency signal

XDEC, YDEC, ZDEC: Deceleration signal of X, Y, Z axes reference return

SKIP: Skip signal

5.1.2 Axes moving state and data diagnosis of CNC

0	0	7	ZTDR	YTDR	XTDR			ZDRO	YDRO	XDRO
----------	----------	----------	-------------	-------------	-------------	--	--	-------------	-------------	-------------

ZTDR, YTDR, XTDR: Moving path direction of Z, Y, X axis, 1 for positive, 0 for negative;

ZDRO, YDRO, XDRO: Z,Y,X axis moving direction output.

0	0	8						ZPC	YPC	XPC
Pin-out								XS31.3	XS33.3	XS30.3

RFZ, RFY, RFX: Z,Y,X axis reference counter

ZPC, YPC, XPC: Z,Y,X axis zero signal(machine→CNC)

0	0	9						ZALM	YALM	XALM
Pin-out								XS31.5	XS33.5	XS30.5

ZALM, YALM, XALM: Z,Y,X axis alarm signal(machine→CNC)

0	1	0								
----------	----------	----------	--	--	--	--	--	--	--	--

MPG speed data: the corresponding bit displayed changes if signal is valid.

0	1	1								
0	1	2								

Spindle feedback data: the corresponding bit displayed changes if spindle encoder signal input is valid.

0	1	3								
0	1	4								

Spindle analog output value: the corresponding bit displayed changes if spindle analog voltage is output.

5.1.3 Keys diagnosis

DGN.016~DGN.022 are the diagnosis messages of edit keypad keys; DGN.024~DGN.029 are the diagnosis messages of machine panel keys. When pressing a key in the operation panel, the corresponding bit displays "1", and "0" after releasing this key. If it displays reversely, it means there is a fault in the keypad circuit.

0	1	6
Corresponding key		

RST	O	N	G	P/Q	7	8	9

0	1	7
Corresponding key		

PGU	X	Y	Z	U/W	4	5	6

0	1	8
Corresponding key		

PGD	H	F/E	R/V	D/L	1	2	3

0	1	9
Corresponding key		

CRU	RIGHT	I/A	J/B	K/C	-	0	.

0	2	0
Corresponding key		

CRD	LEFT	M	S	T	EOB	INS/ALT	DEL

0	2	1
Corresponding key		

***	POS	RPG	OFT	ALM	SET	PAR	DGN

0	2	2
Corresponding key		

IN	OUT	CHG	/, #	CAN	***	***	***

0	2	4
Corresponding key		

EDT	AUT	MDI	HOME	HNDL	JOG	SBK	BDT

0	2	5
Corresponding key		

MLK	AFL	DRN	DNC	0.001	0.01	0.1	HX

0	2	6
Corresponding key		

HY	HZ	Z	Y	***	X	RT	X

0	2	7
Corresponding key		

***	Y	Z	SPM	COOL	SPS	RHST	SPP

0	2	8
Corresponding key		

JSP	SP0+	TRV+	OV+	SP0-	TRV-	OV-	ST
JOG	SPINDLE OVERRIDE	RAPID OVERRIDE	FEEDRATE OVERRIDE	SPINDLE OVERRIDE	RAPID OVERRIDE	FEEDRATE OVERRIDE	CYCLE START

0	2	9
Corresponding key		

SP	***	***	***	***	***	***	***
FEED HOLD							

5.1.4 CNC internal state

During the CNC auto run, the current CNC running state can be viewed by DGN.096 and DGN.097 diagnosis messages if there is no alarm and moving.

1	0	3	PLC execution time (ms)
1	0	4	Hardware check error times
1	0	5	Spindle encoder counter
1	0	6	MPG counting value
1	0	7	Setting time of timing stop (h)
1	0	8	System total running time(h)
1	0	9	Start time of timing stop (h)
1	1	0	Error times of spindle pulse sampling

5.2 PLC state

This part of diagnosis is used to detect the signal state of machine→PLC (X), PLC→machine (Y), CNC→PLC (F), PLC→CNC (G) and alarm address A, which can't be modified. See the relative PLC manual for address F, G significance, and the signal significance of address A is defined by user himself.

5.2.1 X address *(others are defined by PLC except the following fixed addresses)

X0000			ESP		XDEC			
-------	--	--	-----	--	------	--	--	--

ESP: Emergency stop signal

XDEC: Deceleration signal of X axis

X0001					ZDEC	YDEC		SKIP
-------	--	--	--	--	------	------	--	------

ZDEC, YDEC: Deceleration signal of Z,Y axis

X0020	BLOCK SKIP	SINGLE	MANUAL	MPG	MACHINE ZERO	MDI	AUTO	EDIT
-------	------------	--------	--------	-----	--------------	-----	------	------

BLOCK SKIP: block skip key in machine panel

SINGLE: single block key in machine panel

MANUAL: Jog key in machine panel

MPG: MPG/step key in machine panel

MACHINE ZERO: reference return key in machine panel

MDI: manual data input key in machine panel

AUTO: auto run key in machine panel

EDIT: edit key in machine panel

X0021	MPG X	0.1	0.01	0.001	DNC	DRY	MST LOCK	MACHINE LOCK
--------------	-------	-----	------	-------	-----	-----	----------	--------------

MPG X: X axis handwheel key in machine panel

0.1: 0.1 incremental key in machine panel

0.01: 0.01 incremental key in machine panel

0.001: 0.001 incremental key in machine panel

DNC: DNC run key in machine panel

DRY: dry run key in machine panel

MST LOCK: miscellaneous function lock key in machine panel

MACHINE LOCK: machine lock key in machine panel

X0022	X right	RAPID	X left	***	Y up	Z up left	MPG Z	MPG Y
--------------	---------	-------	--------	-----	------	-----------	-------	-------

X right: X axis Jog right key in machine panel

RAPID: manual rapid key in machine panel

X left: X axis Jog left key in machine panel

Y up: Y axis Jog up key in machine panel

Z up left: Z axis up left key in machine panel

MPG Z: Z axis handwheel in machine panel

MPG Y: Y axis handwheel in machine panel

X0023	CCW	LUB	STOP	COOLING	CW	Z down right	Y down	***
--------------	-----	-----	------	---------	----	--------------	--------	-----

CCW: spindle CCW key in machine panel

LUB: lubrication key in machine panel

STOP: spindle stop key in machine panel

COOLING: cooling key in machine panel

CW: spindle CW key in machine panel

Z down right: Jog Z down right key in machine panel

Y down: Jog Y down key in machine panel

X0024	CYCLE START	FEED-	RAPID-	S-	FEED+	RAPID+	S+	JOG
--------------	-------------	-------	--------	----	-------	--------	----	-----

CYCLE START: cycle start key in machine panel

FEED-: feed override - key in machine panel

RAPID-: rapid override - key in machine panel

S-: analog spindle override - key in machine panel

FEED+: feed override + key in machine panel

RAPID+: rapid override + key in machine panel

JOG: Jog key in machine panel

X0025							FEED HOLD
-------	--	--	--	--	--	--	-----------

FEED HOLD: feed holding key in machine panel

X0026							RESET
-------	--	--	--	--	--	--	-------

RESET: reset key of MDI panel

5.2.2 Address Y(except the following address, the other Y addresses are defined by PLC)

Y0004							
-------	--	--	--	--	--	--	--

Bit7: X axis zero return in-position indicator

Bit6: Y axis zero return in-position indicator

Bit5: Dry run indicator

Bit4: Single block indicator

Bit3: Single block indicator

Bit2: Block skip indicator

Bit1: Machine lock indicator

Bit0: MST lock indicator

Y0010	***	***	***	***	***	***	***	XEN1
--------------	-----	-----	-----	-----	-----	-----	-----	------

XEN1: XEN1 X drive enable signal 1

Y0011	***	***	***	***	***	***	XSET	XEN2
-------	-----	-----	-----	-----	-----	-----	------	------

XSET: XSET X drive move signal

XEN2: XEN2 X drive enable signal 2

[illegible]

ZEN1:	ZEN1	Z drive enable signal 1
-------	------	-------------------------

Y0015	***	***	***	***	***	***	ZSET	ZEN2
--------------	-----	-----	-----	-----	-----	-----	------	------

ZSET: ZSET Z drive move signal

ZEN2: ZEN2 Z drive enable signal 2

5.3 PLC Data

The PLC data includes T, C, DT, DC, D, their significance is defined by user requirement.

CHAPTER 6 MEMORIZING SCREW-PITCH ERROR COMPENSATION FUNCTION

6.1 Function Explanation

There are more or less precision errors in the screw-pitch of machine axes lead screw, it will definitely affect the parts machining precision. This GSK980MD has the screw-pitch error compensation memorizing function that it can accurately compensate the screw-pitch error.

6.2 Specification

1 The offset is concerned with the offset origin, offset clearances, offset point, mechanical moving direction etc.;

2 after performing the machine zero return, take this reference point as the offset origin, and set the offset value to be compensated in the parameters on the basis of the axes offset clearances;

3 Points to be compensated: 256 points for each axis

4 Axis compensated: X, Y, Z axis

5 Offset range: $-255 \sim +255 \mu\text{m}$ for each offset point

6 Offset clearance: $1000 \sim 9999999 \mu\text{m}$;

7 Offset of point N(N=0,1,2,3,...255) is determined by the N, N-1 mechanical error;

8 Actual offset clearance: set an appropriate value in the range above according to the max. offset range and mechanical travel;

9 The setting is identical with the CNC parameters input, see the explanation in the relative operation.

6.3 Parameter Setting

6.3.1 Screw-pitch compensation

Bit parameter

0	0	3			Screw-pitch						
---	---	---	--	--	-------------	--	--	--	--	--	--

Bit5 =1: Screw-pitch error compensation valid;

=0: Screw-pitch error compensation invalid.

6.3.2 Screw-pitch error origin

That the screw-pitch error compensation start from a position No. in the offset list, which is determined by the machine zero, is called screw-pitch error offset origin (reference point). Each axis may be set in any position from 0 to 255, which is set by data parameter N₀₆₀, N₀₆₁, N₀₆₂ depending on the mechanical requirement.

Data parameter

0	6	0		Screw-pitch error offset No. of X machine zero
0	6	1		Screw-pitch error offset No. of Y machine zero
0	6	2		Screw-pitch error offset No. of Z machine zero

6.3.3 Offset clearance

Screw-pitch offset clearance: N₀₆₄, N₀₆₅, N₀₆₆;

Input unit: 0.001mm;

Setting range: $1000 \sim 9999999$.

Bit parameter

0	6	4	Clearance of X axis screw-pitch offset
---	---	---	--

0	6	5	Clearance of Y axis screw-pitch offset
---	---	---	--

0	6	6	Clearance of Z axis screw-pitch offset
---	---	---	--

6.3.4 Offset value

The axes screw-pitch offset values are set according to the parameter No. in the following table. The offset value is input by diameter with the unit 0.001mm, which is irrelevant to the programming by diameter or by radius.

Offset No.	X	Y	Z
000
001	5	-2	3
002	-3	4	-1
...
255

6.4 Cautions of Offset Setting

- ①The setting and modification of screw-pitch offset can only be done on the condition that the operation authority is 2nd level password.
- ②Offset is not allowed if the offset clearance entered is 0.
- ③After the parameter of screw-pitch offset is set, only the machine zero is returned could the compensation be done.

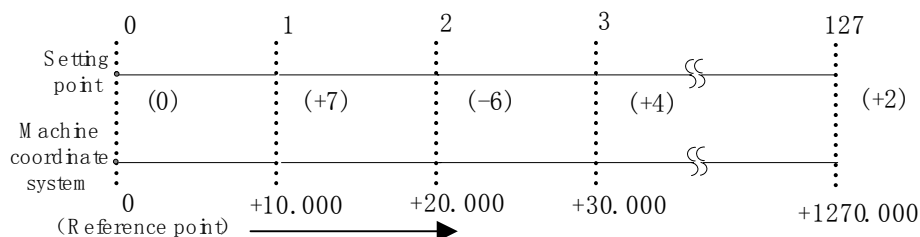
6.5 Setting Examples of Offset Parameters

- ①Data parameter №060 (screw-pitch error origin) =0, Data parameter №064 (screw-pitch offset clearance) =0,

When the screw-pitch error origin is set to 0:

The offset value for the 1st section is set by the position №001 in the offset table, the offset value for the 2nd section is set by the position №002 in the offset table, and the offset value for the Nth section is set by the position №000+N in the offset table.

The machine zero is regarded as the reference point of screw-pitch error origin, it begins to compensate the position №001 in the offset table from the machine zero. So the screw-pitch error compensation can only be performed in the positive moving of the machine zero coordinate system.



The position No.000 in the offset table corresponds to the reference point(i.e screw-pitch error origin 0),the offset point 1 corresponds to a point 10.000 positive moving from this reference point, and an following offset point from this point every 10.000 distance. The 127th offset point is the offset at 1270.000 position. Therefore, at offset

Chapter 6 Memorizing Screw-Pitch Error Compensation Function

point 1, set an offset value moving from 0 to 10.000, at offset point 2, set an offset value moving from 10.000 to 20.000. At offset point N, set an offset value moving from $(N-1) \times (\text{offset clearance})$ to $N \times (\text{offset clearance})$.

Above is the example of following offset clearance error

Offset clearance	Offset value
0~10.000	+7
10.000~20.000	-6
20.000~30.000	+4

Machine coordinate system	Offset parameter No.	Offset value	Driver current command pulses before offsetting	Driver current command pulses after offsetting
Reference point 0	000	000	00000	00000
10.000	001	7	10000	10007
20.000	002	-6	20000	20001
30.000	003	4	30000	30005
.....	004	...		

Actually the machine moves from the reference point to the point of +30.000, the screw-pitch offset is: $(+7)+(-6)+(+4)=+5$

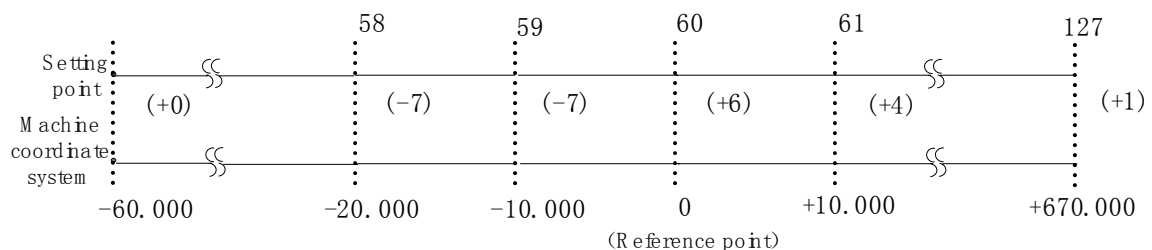
②Data parameter No.060 (screw-pitch error origin) =60, No.064 (Offset clearance) =10.000

When the screw-pitch error origin is set to 60:

For the positive moving, the 1st section error offset is set by position No.061 in the offset table, the 2nd section by position No.062. The Nth section error offset is set by position No.060+N in the offset table.

For the negative moving, the 1st section error offset is set by position No.060 in the offset table, the 2nd section by position No.059. The Nth section error offset is set by position No.060+N in the offset table.

By taking the machine zero as the reference point, the screw-pitch error origin moves from the positive coordinate system of machine zero to compensate the corresponding position No.061 in the offset table, and from the negative coordinate system to compensate the position No.060. Therefore the screw-pitch offset can be performed during the moving in both the positive and the negative coordinate system of machine zero.



The position No.060 in the offset table corresponds to the reference point (60), offset point 61 to a point positive 10.000 from origin. So there is an offset point every 10.000 from point 61, the 127th offset point corresponds to the offset at +670.000 position. While the offset point 59 corresponds to the negative 10.000 point from reference point. Also there is an offset point every 10.000 from point 59, the offset point 0 corresponds to the offset at -600.000 position. Therefore the point N is set by the offset from $(N-61) \times (\text{offset clearance})$ to $(N-60) \times (\text{offset clearance})$.

Above is the example of following offset clearance error

Offset clearance	Offset value
0~10.000	+4
-10.000~0	+6
-20.000~-10.000	-7
-30.000~-20.000	-7

Machine coordinate system	Offset parameter No.	Offset value	Driver current command pulses before offsetting	Driver current command pulses after offsetting
-30.000	058	-7	-30000	-29992
-20.000	059	-7	-20000	-19999
-10.000	060	+6	-10000	-10006
Reference point 0			0	0
10.000	061	+4	10000	10004
.....	062	...		

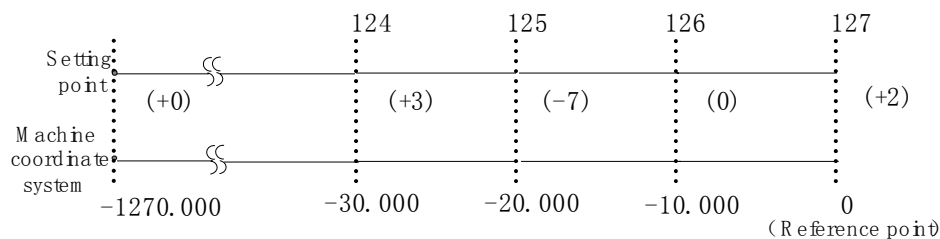
Actually the machine moves from -30.000 point to the point of +10.000, the screw-pitch offset is:
 $(-7)+(-7)+(+6)+(+4)=(-4)$

③Data parameter №060 (screw-pitch error origin) =127, №064 (offset clearance) =10000

When the screw-pitch error origin is set to 127:

The offset value for the 1st section is set by the position №127 in the offset table, the offset value for the 2nd section is set by the position №126 in the offset table, and the offset value for the Nth section is set by the position №128-N in the offset table.

The machine zero is regarded as the reference point of screw-pitch error origin, it begins to compensate the position №127 in the offset table from the machine zero. So the screw-pitch error compensation can only be performed in the negative moving of the machine zero coordinate system.



The position No.128 in the offset table corresponds to the reference point(127),the offset point 126 corresponds to a point 10.000 negative moving from this reference point, and an following offset point from this point every -10.000 distance. The 1st offset point is the offset at -1260.000 position. Therefore, at offset point 127, set an offset value moving from 0 to -10.000, at offset point 126, set an offset value moving from -10.000 to -20.000. At offset point N, set an offset value moving from $(N-128) \times (\text{offset clearance})$ to $(N-127) \times (\text{offset clearance})$.

Above is the example of following offset clearance error

Offset clearance	Offset value
0~-10.000	+2
-20.000~-10.000	0
-30.000~-20.000	-7
-40.000~-30.000	+3

Machine coordinate system	Offset parameter No.	Offset value	Driver current command pulses before offsetting	Driver current command pulses after offsetting
Reference point 0			0	0
-10.000	127	2	10000	10002
-20.000	126	0	20000	20002
-30.000	125	-7	30000	29995
-40.000	124	3	40000	39998

Actually the machine moves from the point of -40.000 to the reference point, the screw-pitch offset is:
 $(+3)+(-7)+(0)+(+2)=(-2)$

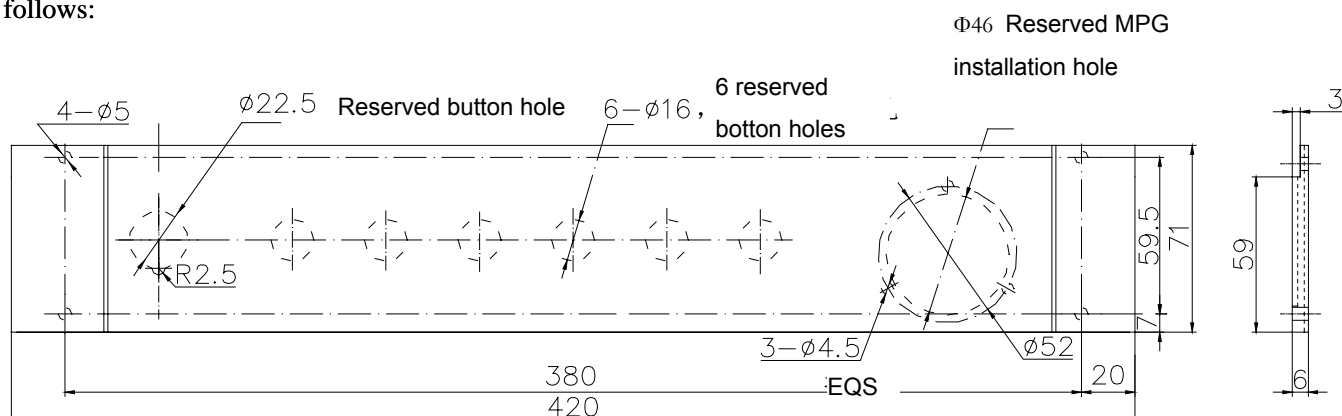
APPENDIX

Contents

Appendix 1	Figure Size for Additional Panel AP01	Appendix -1
Appendix 2	Figure Size for Additional Panel AP02	Appendix -1
Appendix 3	Function Configuration of GSK980MD Standard Ladder Diagram	Appendix -2
1	INFORMATION OF LADDER DIAGRAM	Appendix -2
1.1	Usage	Appendix -2
1.2	Information of Current Version	Appendix -2
2	ADDRESS DEFINITION	Appendix -2
2.1	Fixed Input Address	Appendix -2
2.2	Drawing for Address Interface	Appendix -3
3	FUNCTION CONFIGURATION	Appendix -4
3.1	Spindle CCW and CW Control	Appendix -4
3.2	Spindle JOG	Appendix -5
3.3	Switch Value Control for Spindle Speed	Appendix -6
3.4	Cycle Start and Feed Hold	Appendix -7
3.5	Cooling Control	Appendix -7
3.6	Lubricating control	Appendix -8
3.7	Optional Block Skip	Appendix -9
3.8	Machine Lock	Appendix -9
3.9	MST Lock	Appendix -9
3.10	Single Block	Appendix -10
3.11	Dry Run	Appendix -10
3.12	Travel Limit and Emergency Stop	Appendix -10
3.13	Machine Zero	Appendix -11
Appendix 4	List of CNC Alarm	Appendix -12
Appendix 5	GSK980MD Standard Ladder Diagram	Appendix -15

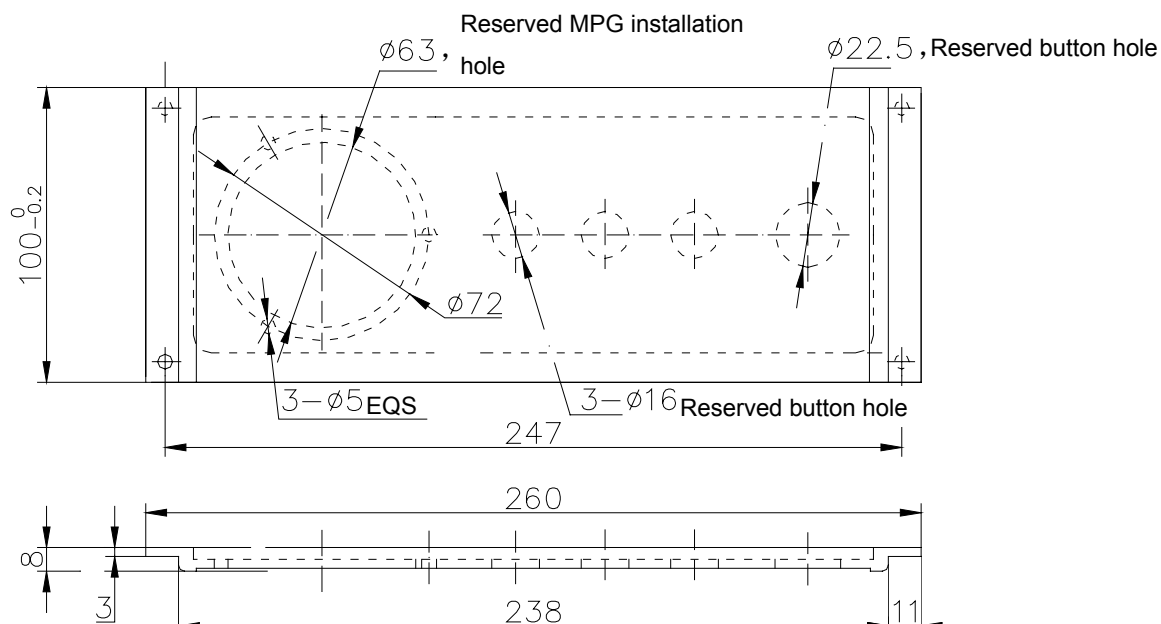
Appendix 1 Figure Size for Additional Panel AP01

AP01: Aluminum alloy 420×71 (mm), it can be spliced below the panel, its figure and installation size are as follows:



Appendix 2 Figure Size for Additional Panel AP02

AP02: Aluminum alloy 100×260 (mm), it can be spliced to the side of panel, its figure and installation size are as follows:



Appendix 3 Function Configuration of GSK980MD Standard Ladder Diagram

1 INFORMATION OF LADDER DIAGRAM

1.1 Usage

Usage range: Configuration for standard ladder diagram (Compatible for GSK980MA/MB/MC)

I/O requirement: To choose based on the special function requirement

Software version: Standard

1.2 Information of Current Version

Designer of ladder diagram: GSK

Version number for ladder diagram: V2.0-07.05.10

Verification of ladder diagram: 8847

Remark for ladder diagram: GSK980MD standard ladder diagram (980MC interface)

2 ADDRESS DEFINITION

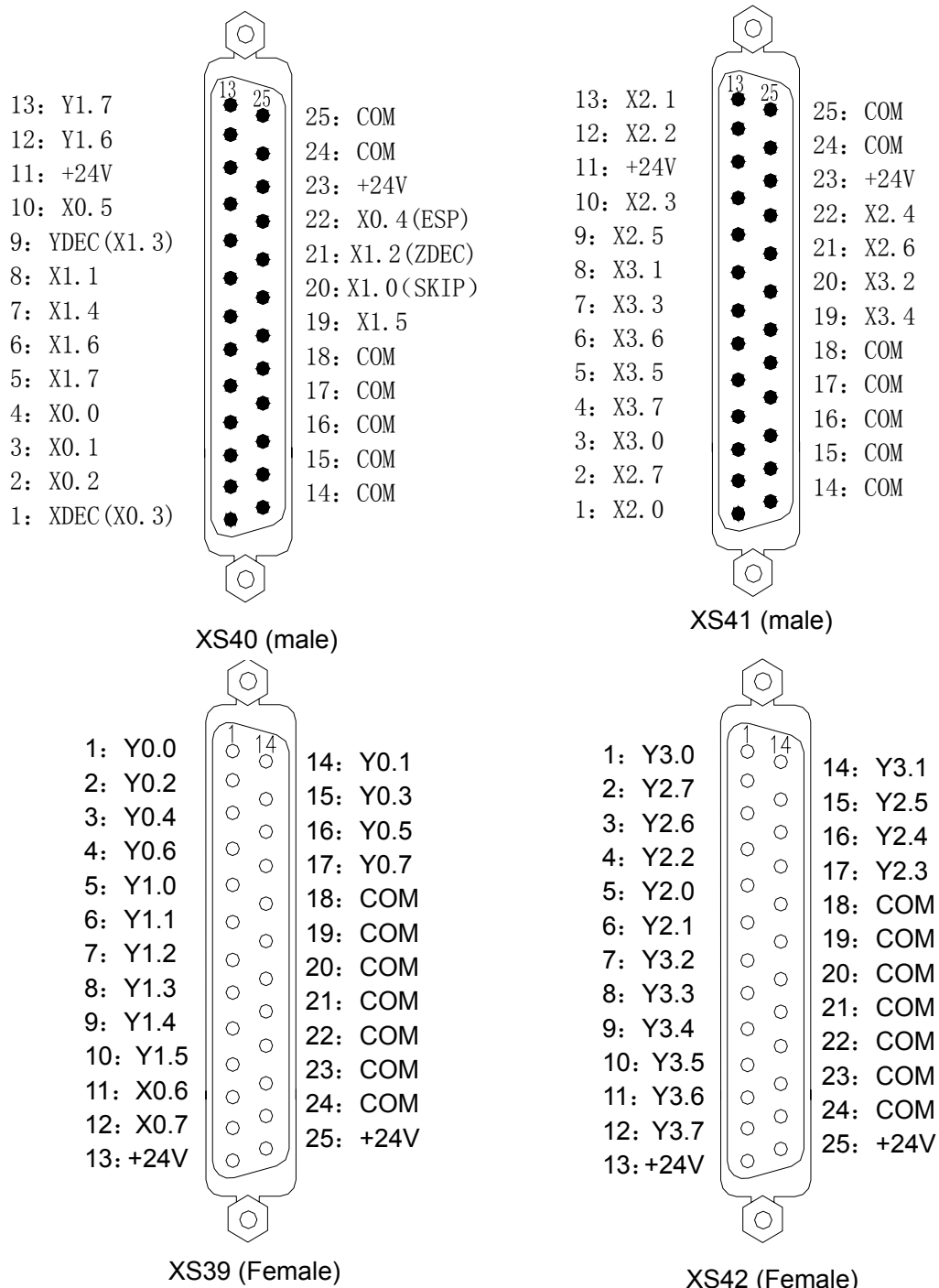
2.1 Fixed Input Address

Corresponding DB Pin-out	PLC address	Standard PLC address definition	Function for standard PLC address definition	Remark
XS40.4	X0.0	OV2	External override signal 2	
XS40.3	X0.1	OV4	External override signal 4	
XS40.2	X0.2	BDT	External skip signal	
XS40.1	X0.3	XDEC	X axis deceleration signal	Fixed address
XS40.22	X0.4	ESP	External urgent stop signal	Fixed address
XS40.10	X0.5			
XS39.11	X0.6			
XS39.12	X0.7			
XS40.20	X1.0	SKIP	Skip signal	Fixed address
XS40.8	X1.1	ST	External cycle start signal	
XS40.21	X1.2	ZDEC	Z axis deceleration signal	Fixed address
XS40.9	X1.3	YDEC	Y axis deceleration signal	Fixed address
XS40.7	X1.4	SP	External dwell signal	
XS40.19	X1.5	SPAL	Spindle alarm signal	
XS40.6	X1.6	OV0	External override signal 0	
XS40.5	X1.7	OV1	External override signal 1	
XS39.1	Y0.0	S02	Spindle mechanical gear signal 2	
XS39.14	Y0.1	S03	Spindle mechanical gear	
XS39.2	Y0.2	M10	Standby signal	
XS39.15	Y0.3	M08	Cooling signal	
XS39.3	Y0.4	M04	Spindle CW signal	
XS39.16	Y0.5	M05	Spindle stop signal	
XS39.4	Y0.6			

Appendix

XS39.17	Y0.7	SPZD	Spindle braking signal	
XS39.5	Y1.0	S01	Spindle mechanical gear signal 1	
XS39.6	Y1.1	M32	Lubricating output signal	
XS39.7	Y1.2	M03	Spindle CCW signal	
XS39.8	Y1.3	S04/#1103	Spindle mechanical gear signal 4	
XS39.9	Y1.4			
XS39.10	Y1.5			
XS40.12	Y1.6	ENB/#1107	Spindle enable signal	
XS40.13	Y1.7	FNI/#1106	M30 finish signal	

2.2 Drawing for Address Interface



3 FUNCTION CONFIGURATION

3.1 Spindle CCW and CW Control

- Related signals

Signal type	Signal sign	Signal signification	Corresponding pin-out	PLC state	CNC diagnosis
Input signal		The CCW key on the machine panel		X23.7	No027.0
		The CW key on the machine panel		X23.3	No027.4
		The Stop key on the machine panel		X23.5	No027.2
		Spindle alarm signal	XS40.19	X1.5	
Output signal	M03	Spindle CCW signal	XS39.7	Y1.2	
	M04	Spindle CW signal	XS39.3	Y0.4	
	M05	Spindle stop signal	XS39.16	Y0.5	
	SPZD	Spindle brake signal	XS39.17	Y0.7	
	ENB	Spindle enable signal	XS40.12	Y1.6	
		Spindle CCW indicator on the machine panel		Y7.1	
		Spindle CW indicator on the machine panel		Y5.1	
Command input	M03	Command signal for spindle CCW			
	M04	Command signal for spindle CW			
	M05	Command signal for spindle stop			

- Control parameter

Bit parameter

0	0	9					RSJG			
Corresponding F signal							F204.3			

RSJG =1: CNC not close M03, M04, M08 and M32 output signals when resetting.

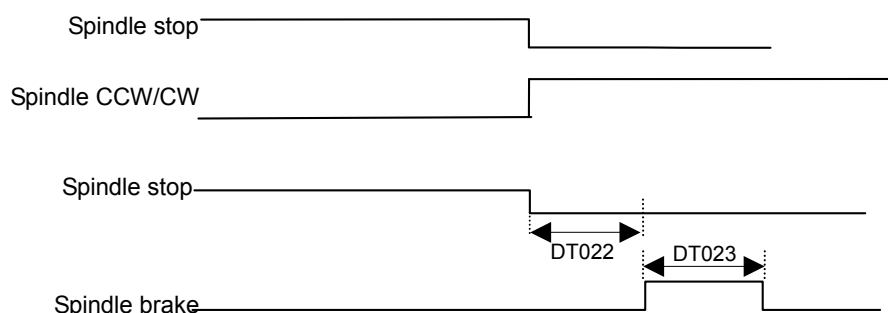
=0: CNC close M03, M04, M08 and M32 output signals when resetting.

PLC data

DT021	M code execution time
DT022	Delay time of spindle stop to braking output
DT023	Spindle braking output time

- Time sequence for operation

Spindle operation time sequence is as follows:



Note: DT022 is the delay time from the spindle stop signal issued to spindle brake signal issued; DT023 is spindle brake hold time.

- **Logic control**

M05 output is enabled after the CNC is power on. As M05 output is enabled, the M03 or M04 output is valid and held on when they are executed, and the M05 output is closed in the meantime; the M05 is executed when M03 or M04 output is enabled, and the M03 or M04 output is then closed, M05 output is enabled and held. Spindle braking signal SPZD output delay is set by data DT022 of PLC (The delay time between the spindle stop command output and braking signal SPZD output). The time for braking signal hold-on is set by data DT023 of PLC (Spindle brake output time).

If the spindle is in CCW or CW, , the alarm A4.2: M03 or M04 code error is generated if the M04 or M03 is performed.

Note 1: When the CNC external emergency stop or spindle alarm is issued, it closes the spindle rotation output signal, and the M05 signal is output in the meantime;

Note 2: When CNC is reset, if M03 or M04 output is cancelled is determined upon the BIT3 of bit parameter No.009:

When Bit3 is set to 0, M03 or M04 output is closed while the CNC is reset.

When Bit3 is set to 1, M03 or M04 output state is invariable while the CNC is reset.

Note 3: The alarm A0.1 (spindle alarm) is generated after the spindle alarm signal X1.5 is detected by PLC;

Note 4: In the spindle analog volume control, if the output voltage is more than 0; the spindle enable signal is enabled.

3.2 Spindle JOG

- **Related signals**

Signal type	Signal sign	Signal signification	Corresponding pin-out	PLC state	CNC diagnosis
Input signal		Signal for spindle JOG key on machine panel		X24.0	No028.7
Output signal		Indicator for spindle JOG start-up on machine panel		Y7.0	

- **Control parameter**

Bit parameter

0	1	1						JSPD
Corresponding F signal								F205.0

JSPD =0: Spindle JOG is enabled only by manual.

=1: Spindle JOG is enabled in any mode.

- **Function description**

The spindle rotates positively (CCW) when pressing and holding the Spindle JOG key on the machine panel, and the spindle is stopped immediately if releasing it.

3.3 Switch Value Control for Spindle Speed

● Related signals

Signal type	Signal sign	Signal signification	Pin-out	PLC state	CNC diagnosis
Input signal	S01	Output signal for spindle gear signal 1	XS39.5	Y1.0	
	S02	Output signal for spindle gear signal 2	XS39.1	Y0.0	
	S03	Output signal for spindle gear signal 3	XS39.14	Y0.1	
	S04	Output signal for spindle gear signal 4	XS39.8	Y1.3	
Command input	S01	Command signal for spindle gear signal 1			
	S02	Command signal for spindle gear signal 2			
	S03	Command signal for spindle gear signal 3			
	S04	Command signal for spindle gear signal 4			
	S00	Command signal for spindle gear signal cancellation			

● Control parameter

Bit parameter

0	0	1				Analog spindle				
Corresponding F signal						F204.0				

Bit4 =1: Analog voltage control for spindle speed;

=0: Switch value control for spindle speed.

1	7	3				ESCD				SOUS
Corresponding F signal						F211.4				F211.0

SOUS =0: S1, S2, S3 and S4 enabled in the switch value control for spindle speed.

=1: S3 and S4 disabled; S1 and S2 enabled in the switch value control for spindle speed.

ESCD =0: The S code not closed when stopping urgently;

=1: The S code closed when stopping urgently.

PLC data

DT019	S code performance time	
DT024	Delay time for spindle gear shift	

● Logic control

1. The S1~S4 output are disabled when CNC is power on. Any command is performed among S01, S02, S03 and S04, the corresponding S signal is output and held on, and the rest 3 S signals output is cancelled at the same time. When the S00 command is executed, the S1~S4 output are cancelled, and only one of them is enabled at the same time.

2. When SOUS is equal to 1, the alarm A1.2 (S3/S4 output is disabled when the bit parameter 173.0 is 1) is generated when S3 and S4 are being performed.

3.4 Cycle Start and Feed Hold

● Related signals

Signal type	Signal sign	Signal signification	Pin-out	PLC state	CNC diagnosis
Input signal	ST	External cycle start signal	XS40.8	X1.1	
	SP	Feed hold signal in external	XS40.7	X1.4	
		Cycle Start key signal on the machine panel		X24.7	№028.0
		Feed Hold key signal on the machine panel		X25.0	№029.7
		OUT cycle start signal on MDI panel		F197.1	№022.6
Output signal		Cycle start indicator on machine panel		Y9.0	
		Feed hold indicator on machine panel		Y8.0	
Command input	M00	Feed hold signal		F9.7	

● Control Parameter

Bit parameter

1	7	2		MST	MSP						
Corresponding F signal				F210.6	F210.5						

MST =1: External cycle start signal disabled;

=0: External cycle start signal enabled;

MSP =1: External feed hold signal disabled;

=0: External feed hold signal enabled, and external dwell switch needed, or CNC “dwell” alarm occurs;

0	0	4						DCS		
Corresponding F signal								F201.2		

DCS =1: Program run by OUT key on the MDI panel in MDI mode.

=0: Program run by OUT key on the MDI panel disabled in MDI mode

3.5 Cooling Control

● Related signals

Signal type	Signal sign	Signal signification	Pin-out	PLC state	CNC diagnosis
Input signal		Cooling key signal on machine panel		X23.4	№027.3
Output signal		Cooling on indicator on machine panel		Y5.0	
	M08	Cooling output signal	XS39.15	Y0.3	
Command input	M08	Command signal for cooling on			
	M09	Command signal for cooling off			

● Control parameter

Bit parameter

0	0	9					RSJG			
Corresponding F signal							F204.3			

RSJG =1: CNC not close M03, M04, M08 and M32 output signal when reset.

=0: CNC close M03, M04, M08 and M32 output signal when reset.

● **Function description**

M09 is effective (i.e. M08 ineffective) after CNC power on, M08 output is effective for cooling pump on when it is executed; M08 output is cancelled if M09 is executed, and the cooling pump is off.

Note 1: Cooling output is switched off when CNC is at emergency stop externally;

Note 2: Cooling output off or not is defined by the Bit3 of the bit parameter No.009 when CNC is reset:

If Bit 3 is equal to 0, the cooling output is off when CNC is reset.

If Bit 3 is equal to 1, the cooling status is unchanged when CNC is reset.

Note 3: If M09 has no corresponding output signal, the output of M08 is cancelled as M09 is executed.

Note 4: The cooling output is off when M30 is executed.

3.6 Lubricating control

● **Related signal**

Signal type	Signal sign	Signal signification	Pin-out	PLC state	CNC diagnosis
Input signal		Lubricating key signal on the machine panel		X23.6	№027.1
Output signal		Lubricating on indicator on machine panel		Y8.7	
	M32	Lubricating output signal	XS39.6	Y1.1	
Command input	M32	Lubricating on command signal			
	M33	Lubricating off command signal			

● **Control parameter**

PLC data

DT016	Interval for automatic lubricating
DT017	Lubricating time for automatic lubricating
DT018	Lubricating reverse or timing choice for non-automatic lubricating

● **Function description**

There are two lubricating functions defined by GSK980MD standard PLC program: non-automatic lubricating and automatic lubricating, they are set by PLC data.

DT017 =0: For non-automatic lubricating

>0: For automatic lubricating, lubricating time DT017 and lubricating interval time DT016 can be set.

DT018 =0: For non-automatic lubricating, lubricating reverse.

>1: For non-automatic, timing lubricating.

1. Non-automatic lubricating function

When PLC data DT018 is equal to 0, it is lubricating reverse output. Lubricating output is performed when pressing the Lubricating key on the machine panel; lubricating is cancelled if pressing the key again. When M32 is performed and the lubricating is output, then performing the M33, the lubricating output is cancelled.

When the PLC data DT018 is equal to 0, it is lubricating timing output, the lubricating output is performed when the Lubricating key on the machine panel is pressed. The lubricating output is cancelled after the time set by PLC data DT018. So, M32 is performed for the lubrication output, it is cancelled after the time set by PLC data DT018. If the time set by DT018 has not be reached, M33 is performed then the lubricating output is cancelled.

2. Automatic lubrication

Lubricating function is applied for the time set by DT017 after the system is power on, then the output is stopped. After the time set by DT016, lubrication is output again, and then it cycles in turn. The M32, M33 commands and the Lubricating key on the machine panel are all ineffective when the automatic lubrication is applied.

3.7 Optional Block Skip

The optional block skip function can be applied when one block is neither performed nor deleted in a program. When the “/” is at the head of a block and the optional block skip switch is on (the Block Skip key on machine panel or external output for optional block skip is enabled), this block is not run in automatic operation.

● Related Signals

Signal type	Signal sign	Signal signification	Pin-out	PLC state	CNC diagnosis
Input signal		optional block skip key signal on machine panel		X23.6	Nº024.0
		External optional block skip signal	XS40.2	X0.2	
Output signal		Optional block skip indicator on machine panel		Y6.6	
		Optional block skip indicator by state indication		Y4.2	

● Function description

1. When BDT signal is effective, a block headed with “/” is not performed.
2. The optional block skip function is enabled only in the modes of Auto, MDI and DNC.

3.8 Machine Lock

● Related signals

Signal type	Signal sign	Signal signification	Pin-out	PLC state	CNC diagnosis
Input signal		Machine lock key signal on the machine panel		X21.0	Nº025.7
Output signal		Machine lock indicator on the machine panel		Y6.5	
		Machine lock indicator of state indication		Y4.1	

● Function description

1. Machine locks are enabled in any mode.
2. The machine lock state can not be shifted when program is being run.

3.9 MST Lock

● Related signals

Signal type	Signal sign	Signal signification	Pin-out	PLC state	CNC diagnosis
Input signal		MST lock key signal on the machine panel		X21.1	Nº025.6
Output signal		MST lock indicator on machine panel		Y6.4	
		MST lock indicator of state indication		Y4.0	

● Function description

MST lock is enabled in the mode of Auto, MDI or DNC;

3.10 Single Block

● Related signal

Signal type	Signal sign	Signal signification	Pin-out	PLC state	CNC diagnosis
Input signal		Single key signal on machine panel		X20.6	No024.1
Signal output		Single indicator on machine panel		Y6.7	
		Single indicator of state indication		Y4.3	

● Function description

Single block is enabled in the mode of Auto, MDI or DNC.

3.11 Dry Run

● Related signals

Signal type	Signal sign	Signal signification	Pin-out	PLC state	CNC diagnosis
Input signal		Dry run key signal on machine panel		X21.2	No025.5
Output signal		Dry run indicator on machine panel		Y6.3	
		Dry run indicator of state indication		Y9.1	

● Function description

1. Dry run for program is enabled in the mode of Auto, MDI or DNC.
2. Dry run state can not be switched during the program execution.

3.12 Travel Limit and Emergency Stop

● Related Signal

Signal type	Signal sign	Signal signification	Pin-out	PLC state	CNC diagnosis
Input signal		Dry run key signal on machine panel	XS40.22	X1.4	

● Control parameter

Bit parameter

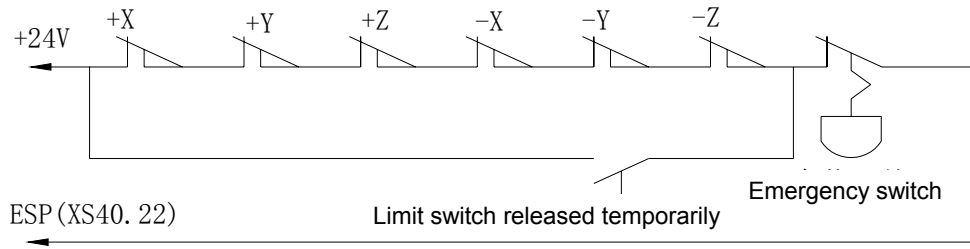
1	7	2					ESP			
---	---	---	--	--	--	--	-----	--	--	--

The external emergency stop function effective.

=1: The external emergency stop function is ineffective.

● External connection for machine

External emergency stop and travel switch connection is as follows:



● **Logic control**

When the contact of emergency stop switch is off, the ESP signal to +24V are cut off, and the CNC emergency stop alarm is generated. Now the CNC EN signal is disabled and the pulse output is stopped. In the emergency stop alarm, other functions can be defined by PLC other than above functions by NC. The function defined by standard PLC program is: when emergency stop alarm is issued, M03 or M04, M08 output signal is off and M05 signal is output at the same time.

3.13 Machine Zero

● **Related signal**

Signal type	Signal sign	Signal signification	Pin-out	PLC state	CNC diagnosis
Input signal	XDEC	External zero return deceleration signal along X axis	XS40.1	X0.3	
	YDEC	External zero return deceleration signal along Y axis	XS40.9	X1.3	
	ZDEC	External zero return deceleration signal along Z axis	XS40.21	X1.2	
	XPC	External machine zero signal along X axis	XS30.3		
	YPC	External machine zero signal along Y axis	XS33.3		
	ZPC	External machine zero signal along Z axis	XS31.3		
Output signal		X machine zero end indicator for state indication		Y4.7	
		Y machine zero end indicator for state indication		Y4.6	
		Z machine zero end indicator for state indication		Y4.5	

● **Control parameter**

Bit parameter

0	1	1						ZNIK		
Corresponding F signal								F205.2		

ZNIK =1: Axis movement key lock during zero return.

=0: Axis movement key not lock during zero return.

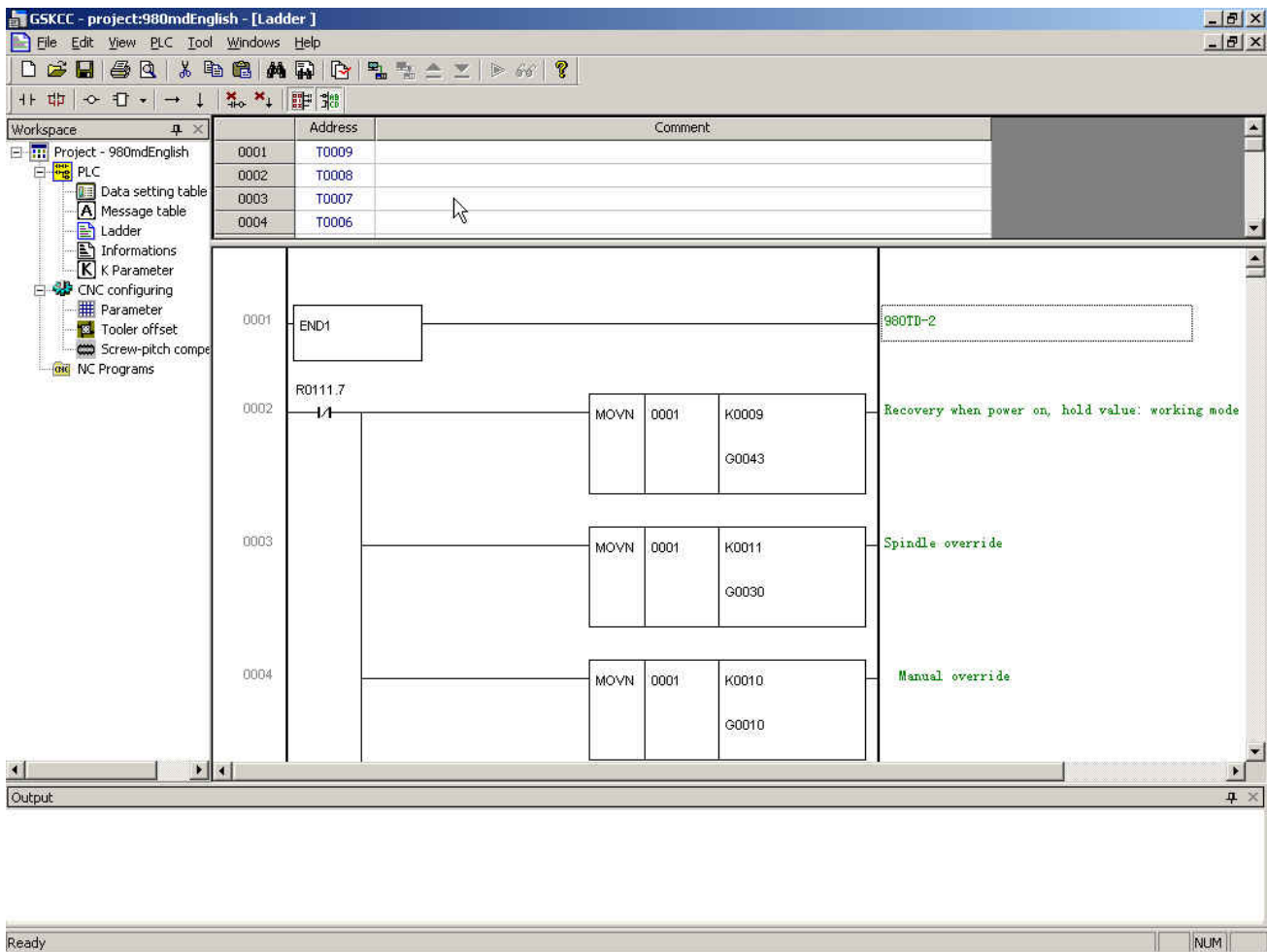
Appendix 4 List of CNC Alarm

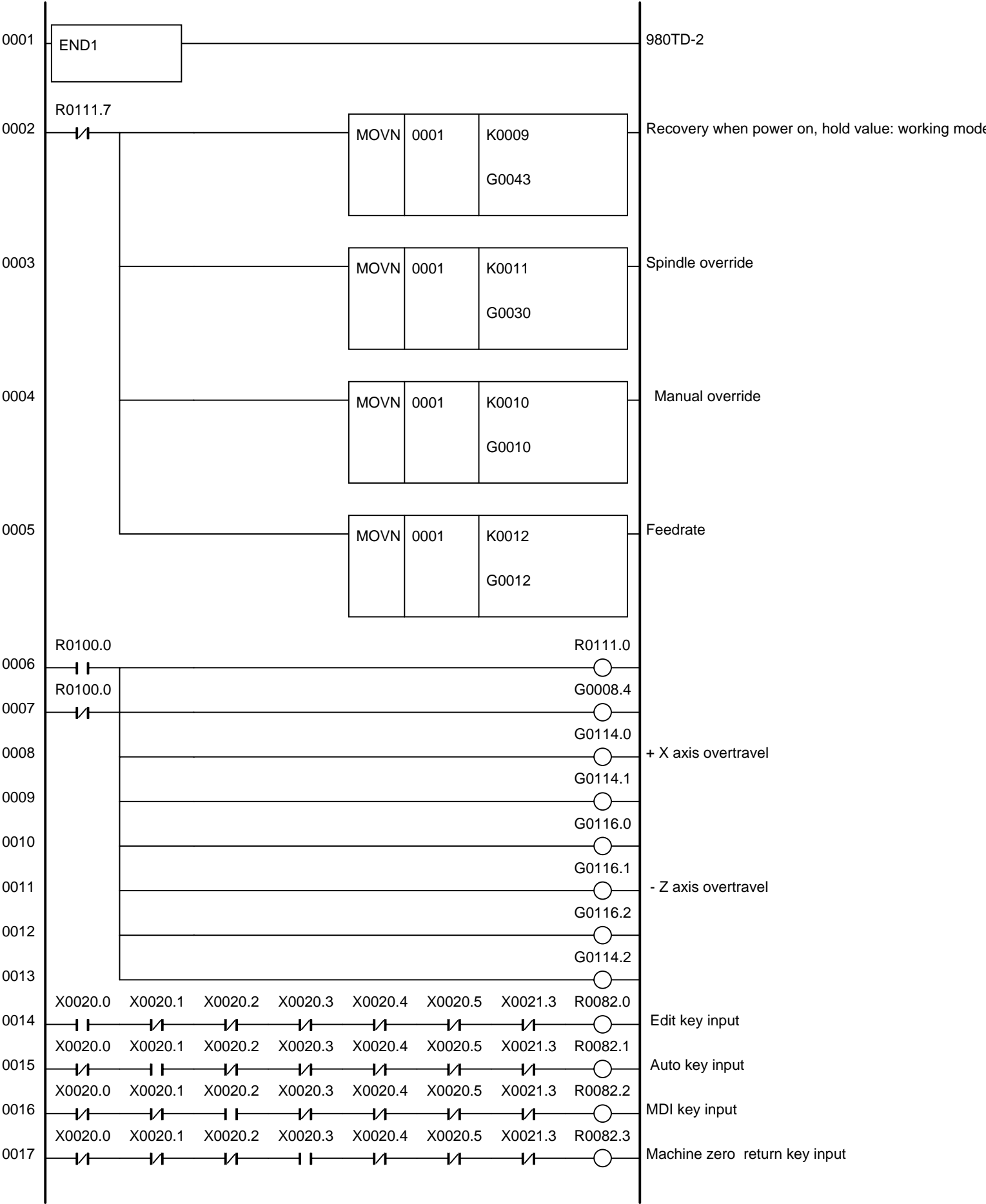
000 ESP ALARM, ESP INPUT OPEN LOOP
 001 PROGRAM NOT EXIST OR OPEN FAIL
 002 - OR DECIMAL POINT IN G CODE
 003 WORD CHARACTERS < 2 OR > 11
 004 COMMAND ADDRESS ERROR (SHOULD BE A~Z)
 005 ILLEGAL COMMAND VALUE
 006 - OR DECIMAL POINT IN BLOCK
 007 ILLEGAL G CODE
 008 I, J, K ON AXIS OTHER THAN PLANE SELECTED
 009 G CODES NOT INPUT AND MOVE SPECIFIED
 010 SAME ADDRESS IN A BLOCK
 011 WORDS OVER 20 IN A BLOCK
 012 VALUE OVER RANGE OR WORD NEEDED OMITTED
 013 S CODE OTHER THAN S00~S99 INPUT
 014 G CODES OF 00 & 01 GROUP IN A SAME BLOCK
 015 M CODE FOR AUTO GEAR SHIFT EXECUTED
 016 TOOL OFFSET NO. BEYOND RANGE(0~32)
 017 TOOL NO. NOT IN BIT PARA NO.055 RANGE
 018 ARC CAN'T BE DRAWN BY G02 OR G03 DATA
 019 TOOL GROUP NO. BEYOND RANGE(1~32)
 020 TOOL RADIUS OFFSET NO. OVER RANGE(1~32)
 021 INCORRECT I, J, K IN G02 OR G03
 024 NO G11 IN PROGRAM
 025 NO TOOL IN CURRENT GROUP
 026 CURRENT TOOL GROUP UNDEFINED
 027 TOOLS OVER 8 IN CURRENT GROUP
 028 G10 L13 COMMAND UNALLOWED
 029 G11 CAN'T BE PRIOR TO G10
 030 OFFSET PLANE CHANGED IN C OFFSET
 031 OFFSET PLANE CHANGED IN AUTO CHAMFERING
 032 RADIUS R DEFINED BUT MOVE NOT SPECIFIED
 033 MOVE NOT SPECIFIED IN 07 GROUP G CODES
 034 ARC DATA ERROR IN C OFFSET
 035 G31 CODE UNALLOWED IN C OFFSET
 036 G31 CODE UNALLOWED IN CHAMFERING
 037 BLOCK CHARACTERS OVER 256
 095 SUBPROG NO. NOT INPUT OR ILLEGAL IN M98
 096 SUBPROG NESTING LEVELS OVER 4
 097 CURRENT (MAIN) PROGRAM CALLED BY M98
 098 M98 OR M99 CODE USED IN MDI MODE
 099 M98 OR M99 CODE USED IN C OFFSET
 101 H11,H12,H13,H25 IS NOT BINARY IN G65
 102 H24 IS OVER 1023 IN G65
 103 DIVIDED BY 0 IN G65
 104 H NOT SPECIFIED OR ILLEGAL IN G65

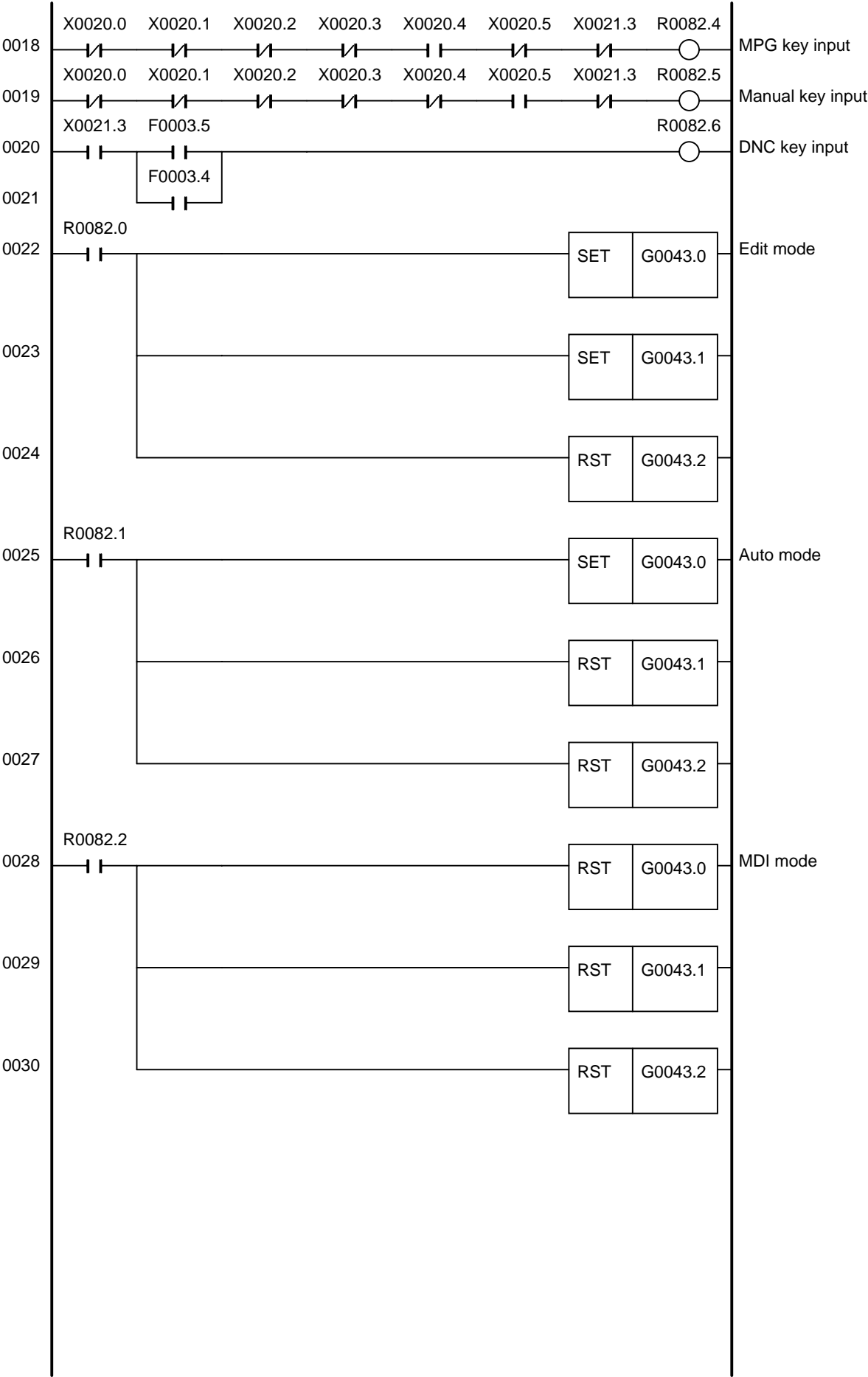
105 ILLEGAL MACRO NO. (ERROR) IN G65
106 P IS 0 OR NOT SPECIFIED IN G65
107 Q NOT INPUT OR ILLEGAL IN G65
108 R NOT INPUT OR ILLEGAL IN G65
109 P IS NOT VARIABLE IN G65
110 VALUE TO THE POWER 1/2 IS NEGATIVE IN G65
111 H99 USER ALARM NO. OVER RANGE IN G65
112 SKIP OR M99 BLOCK NO. OVER RANGE IN G65
113 SKIP OR M99 BLOCK NO. NOT EXIST IN G65
114 ILLEGAL ADDRESS IN G65
115 G65 WITH G43, G44, G49 IN A BLOCK
116 G65 WITH M00, 01, 02,30,98,99 IN A BLOCK
117 G65 CODE USED IN C OFFSET
121 SPINDLE ENCODER P/R OVER 100--5000!
205 K VALUE NOT DEFINED
206 I VALUE NOT DEFINED
207 I VALUE TOO SMALL
208 J VALUE NOT DEFINED
209 J VALUE TOO SMALL
210 CORNER RADIUS TOO BIG OR I,J VALUE TOO SMALL
211 J VALUE TOO BIG
212 K VALUE TOO SMALL
213 U VALUE SMALLER THAN TOOL RADIUS
214 OVERCUT DUE TO BIG K OR SMALL I, J
215 NO J OR START AND END POINT SAME
216 CANNED CYCLE PUNCH MODE NEEDED (G73~G89)
251 C OFFSET ERROR BY PROGRAMMING
252 END POINT NOT IN ARC BY PROGRAMMING
253 SAME COORDINATES OF 2 ADJACENT POINTS
254 ARC CENTER AND START POINT IDENTICAL
255 ARC CENTER AND END POINT IDENTICAL
256 ARC RADIUS LESS THAN TOOL NOSE RADIUS
257 NO INTERSECTION OF 2 ARC PATH
258 ARC SPECIFIED IN C OFFSET SETUP
259 ARC SPECIFIED IN C OFFSET CANCEL
260 OVERCUT IN C OFFSET INTERFERENCE CHECK
261 NO INTERSECTION OF LINEAR TO ARC PATH
262 NO INTERSECTION OF ARC TO LINEAR PATH
263 OFFSET BUFFER OVERFLOW BY MANY NON-MOVE
281 LINEAR CHAMFERING TOO LONG
282 CORNER ROUNDING RADIUS TOO LARGE
283 LINEAR CHAMFERING LONG OR ARC DATA ERROR
284 ROUND RADIUS TOO BIG OR ARC DATA ERROR
285 LINE CHAMFER TOO LONG OR ARC DATA ERROR
286 CORNER RADIUS TOO LARGE OR ARC ERROR
287 CHAMFER TOO LONG OR INTERSECT NOT IN ARC
288 RADIUS TOO LARGE OR INTERSECT NOT IN ARC

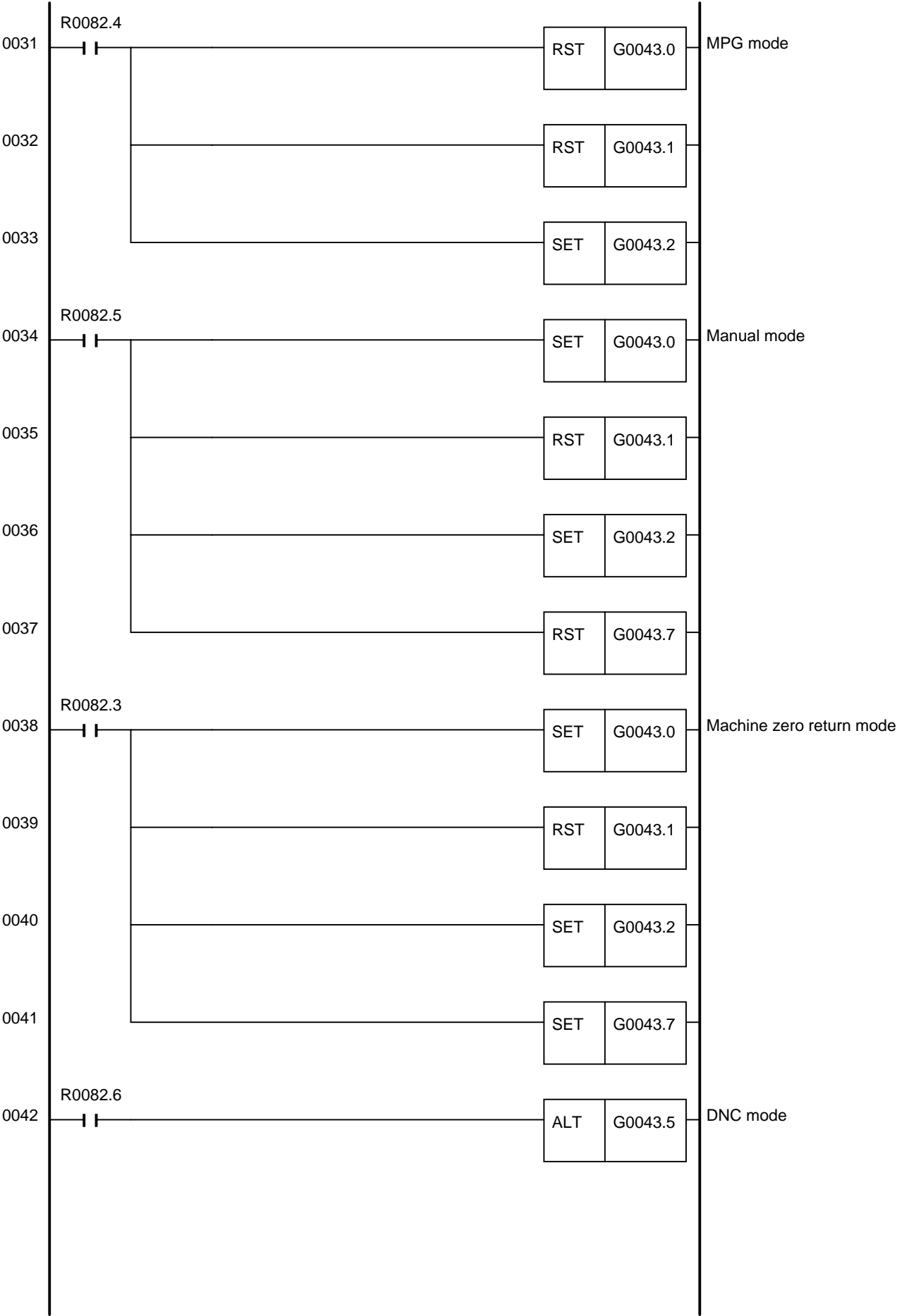
289 CHAMFERING UNALLOWED AT END POINT
 301 PARAMETER SWITCH OPENED
 302 CNC INITIATION FAIL
 303 PART PROGRAM OPEN FAIL
 304 PART PROGRAM SAVE FAIL
 305 ROWS BEYOND RANGE (69993) OPEN UNALLOWED
 306 ILLEGAL WORDS
 307 MEMORY CAPACITY FULL
 308 PROGRAM NO. BEYOND RANGE
 309 MACRO EDIT UNALLOWED IN CURRENT OPERATE
 310 PLC (LADDER) OPEN FAIL
 311 PLC (LADDER) EDIT VERSION ERROR
 312 PLC (LADDER) 1ST LEVEL PROGRAM TOO LONG
 313 EDIT KEYBOARD OR PANEL FAULT
 314 MEMORY FAULT, REPAIR OR REPOWER IT
 315 DNC ERROR, CHECK CONNECTION AND BAUDRATE
 316 PARAMETER FILE SAVE FAIL
 317 FILE SYSTEM ERROR
 350 PARA FILE OPEN FAIL, USE FACTORY SETTING
 351 PARA LOADING ERROR, USE FACTORY SETTING
 352 DATA ERROR, RESET, REOPERATE AFT ZR RETURN
 353 DATA ERROR, RESUME, REOPERATE AFT ZR RETURN
 401 G29 INTERMEDIATE POINT NOT DEFINED
 402 NO GEAR MAX. SPEED, CHECK PAR NO.037~040
 403 RUNNING TOO FAST
 404 FEEDING STOPPED FOR SPINDLE HALT
 405 SPINDLE SPEED TOO LOW IN THREADING
 406 SPINDLE DIRECTION REVERSE TO COMMAND
 407 SPINDLE FLUCTUATION OVER PAR 69# LIMIT
 409 REFERENCE POINT NOT SETUP
 411 BEYOND +X SOFTWARE STROKE
 412 BEYOND -X SOFTWARE STROKE
 413 BEYOND +Z SOFTWARE STROKE
 414 BEYOND -Z SOFTWARE STROKE
 416 + X AXIS OVERTRAVEL
 417 - X AXIS OVERTRAVEL
 418 + Z AXIS OVERTRAVEL
 419 - Z AXIS OVERTRAVEL
 421 X AXIS DRIVER UNREADY
 422 Z AXIS DRIVER UNREADY
 423 Y AXIS DRIVER UNREADY
 426 X AXIS DRIVER ALARM
 427 Z AXIS DRIVER ALARM
 428 Y AXIS DRIVER ALARM
 431 BEYOND +Y SOFTWARE STROKE
 432 BEYOND -Y SOFTWARE STROKE
 433 + Y AXIS OVERTRAVEL
 434 - Y AXIS OVERTRAVEL

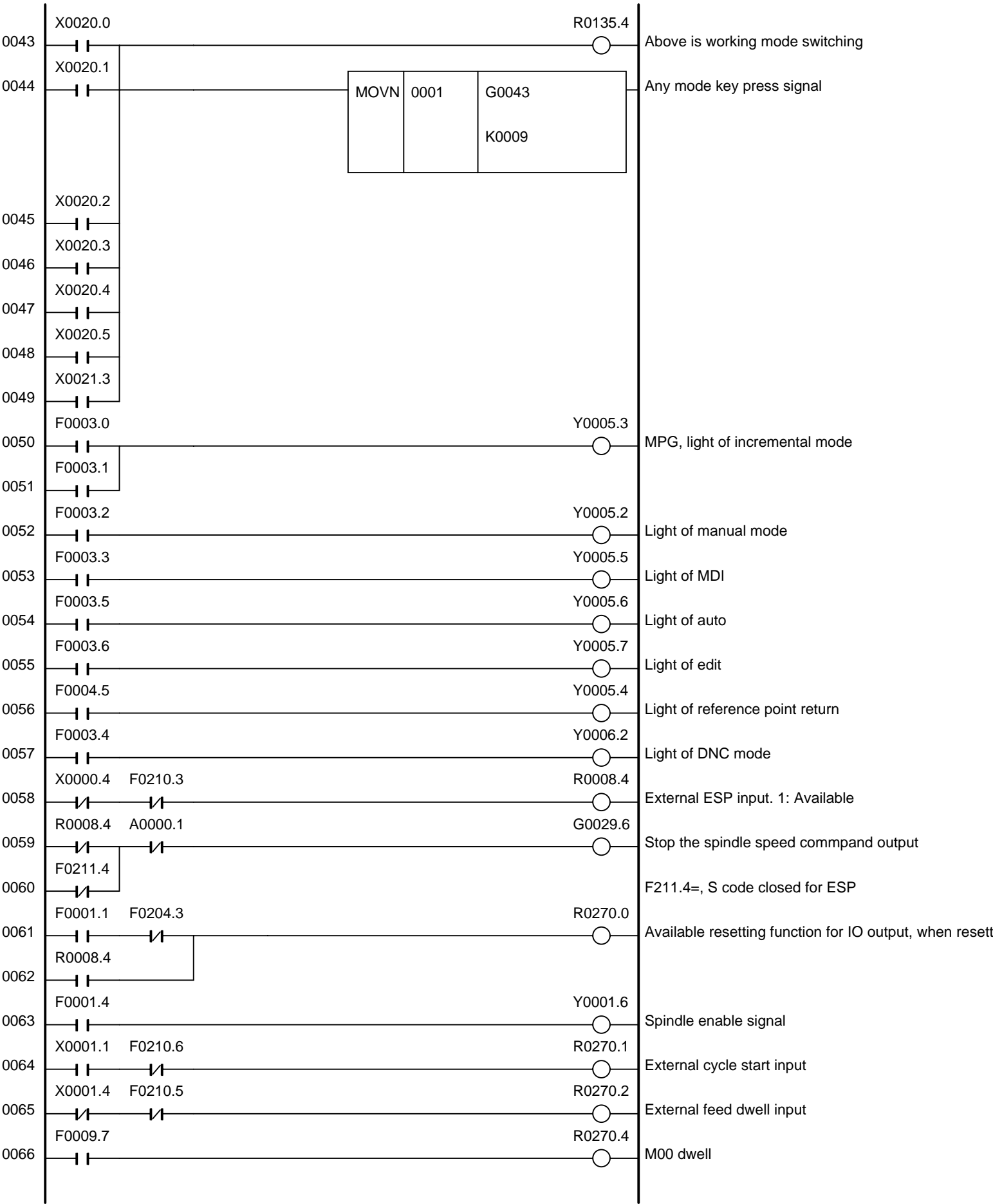
Appendix 5 GSK980MD Standard Ladder Diagram

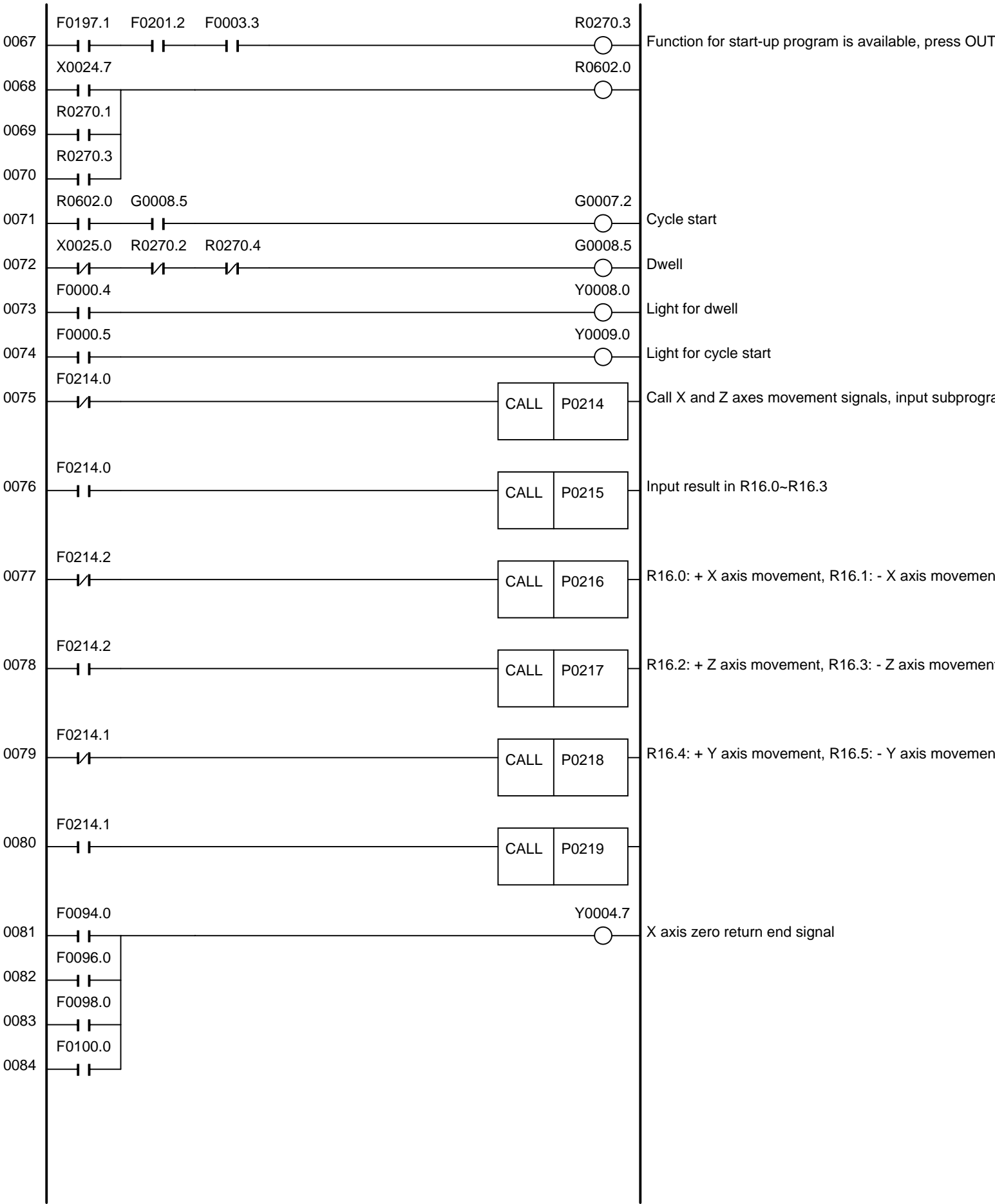


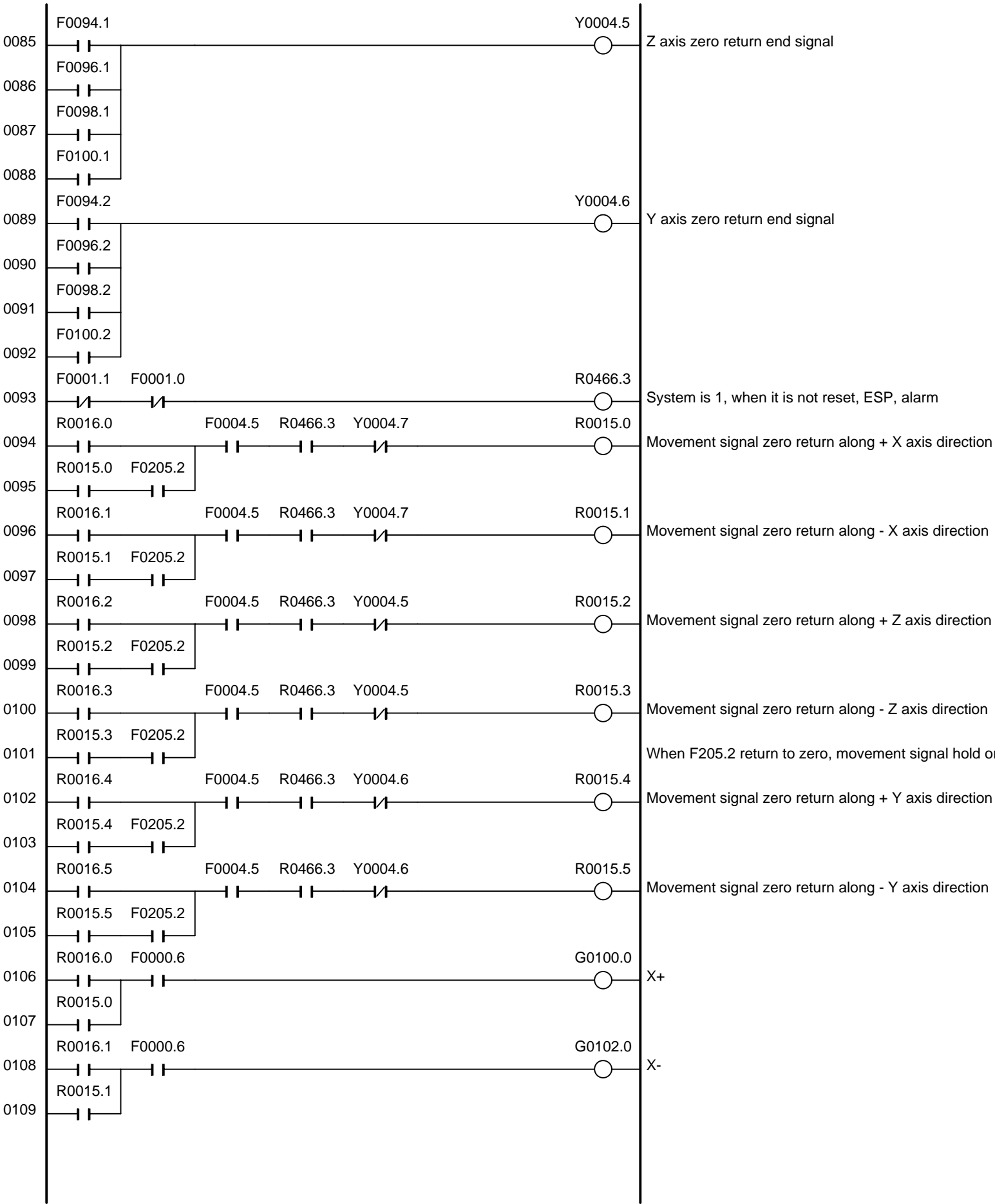


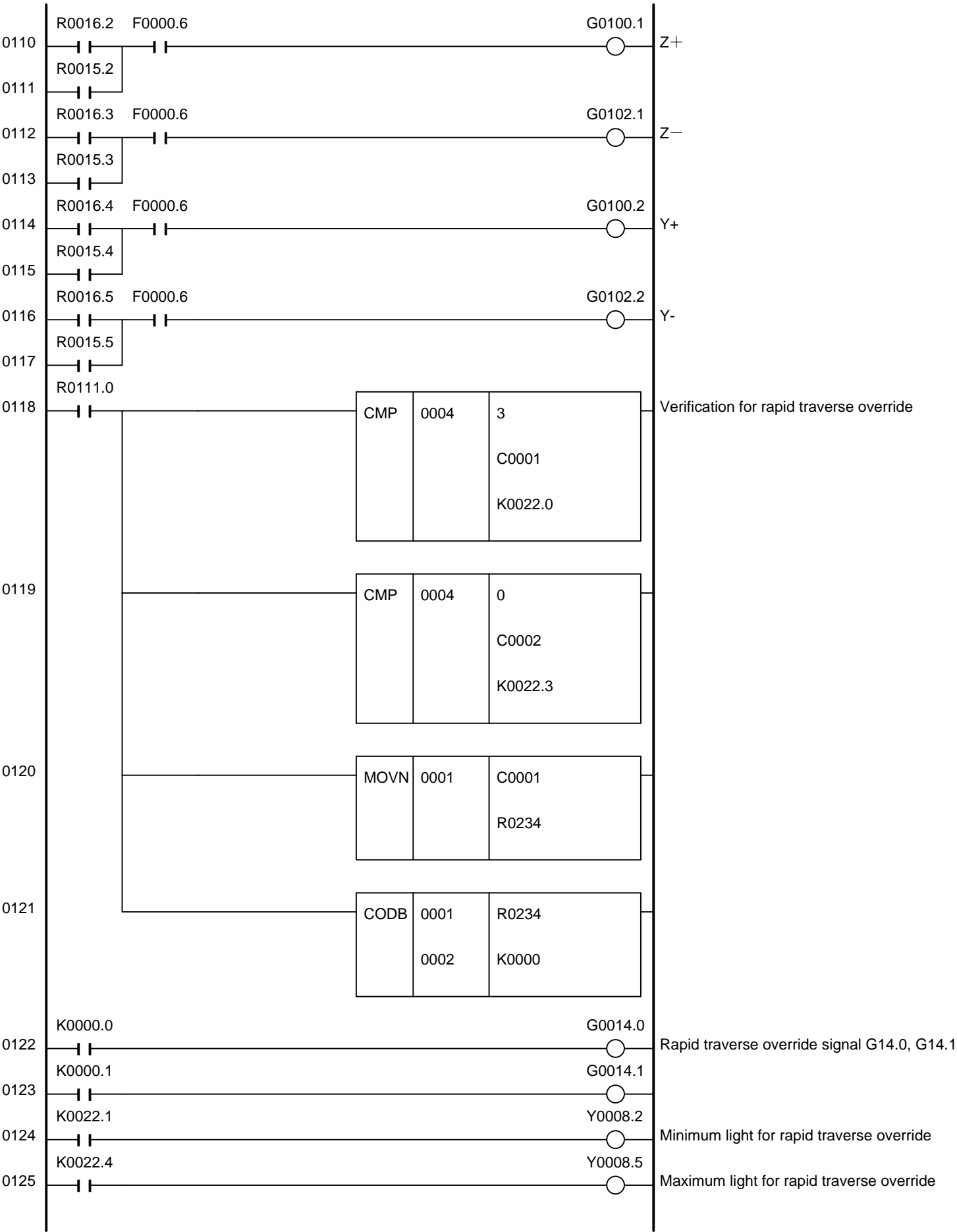


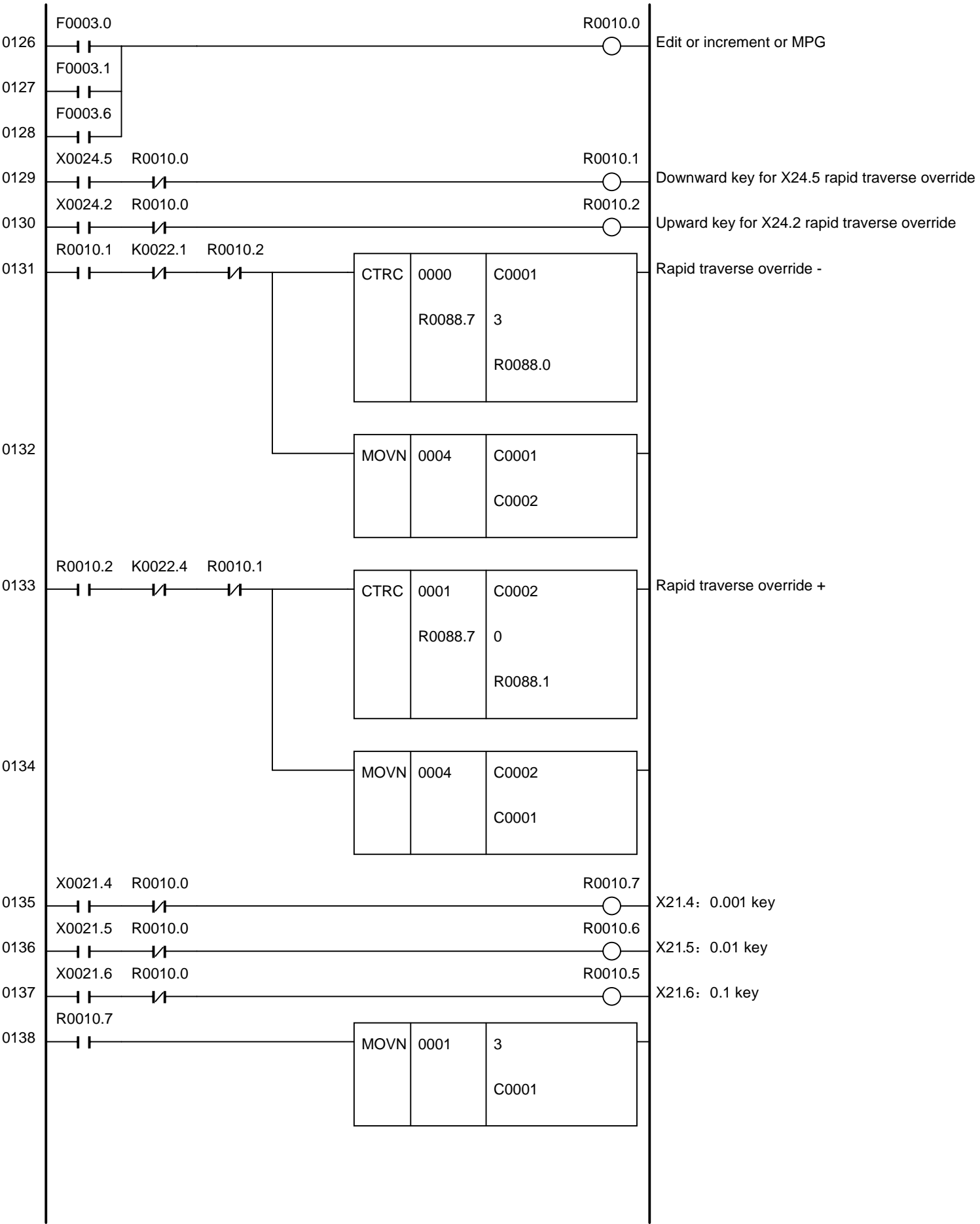


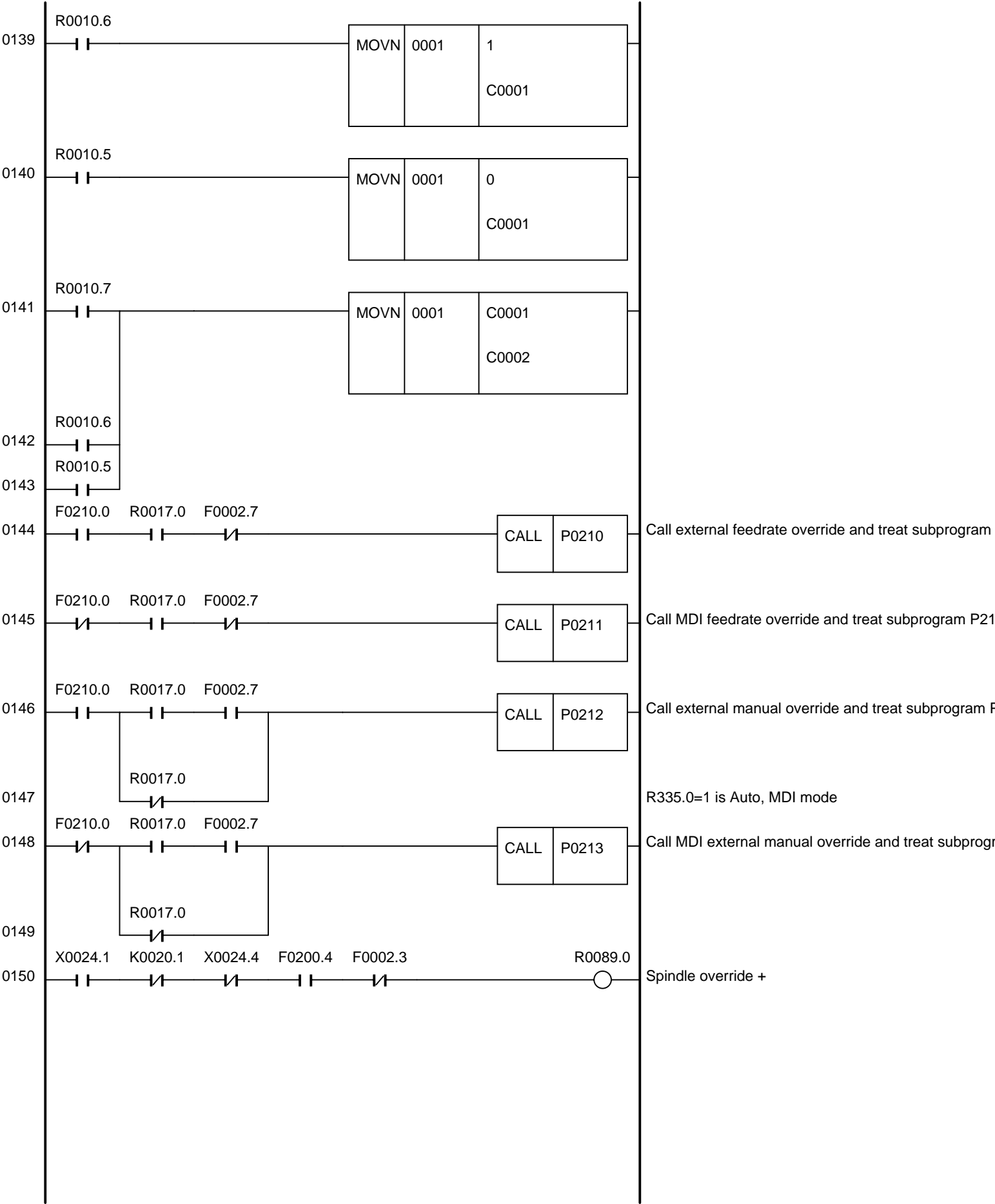


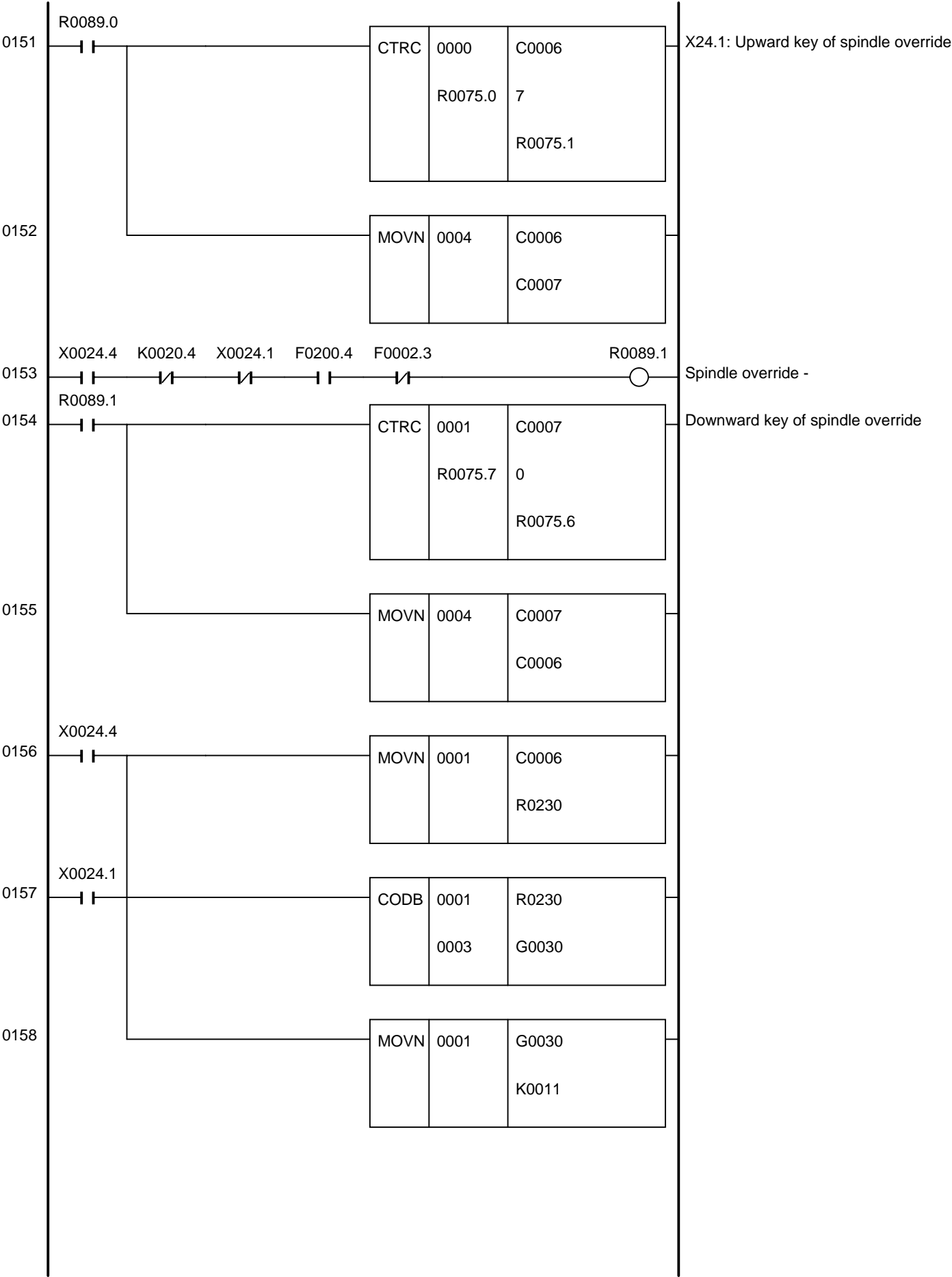


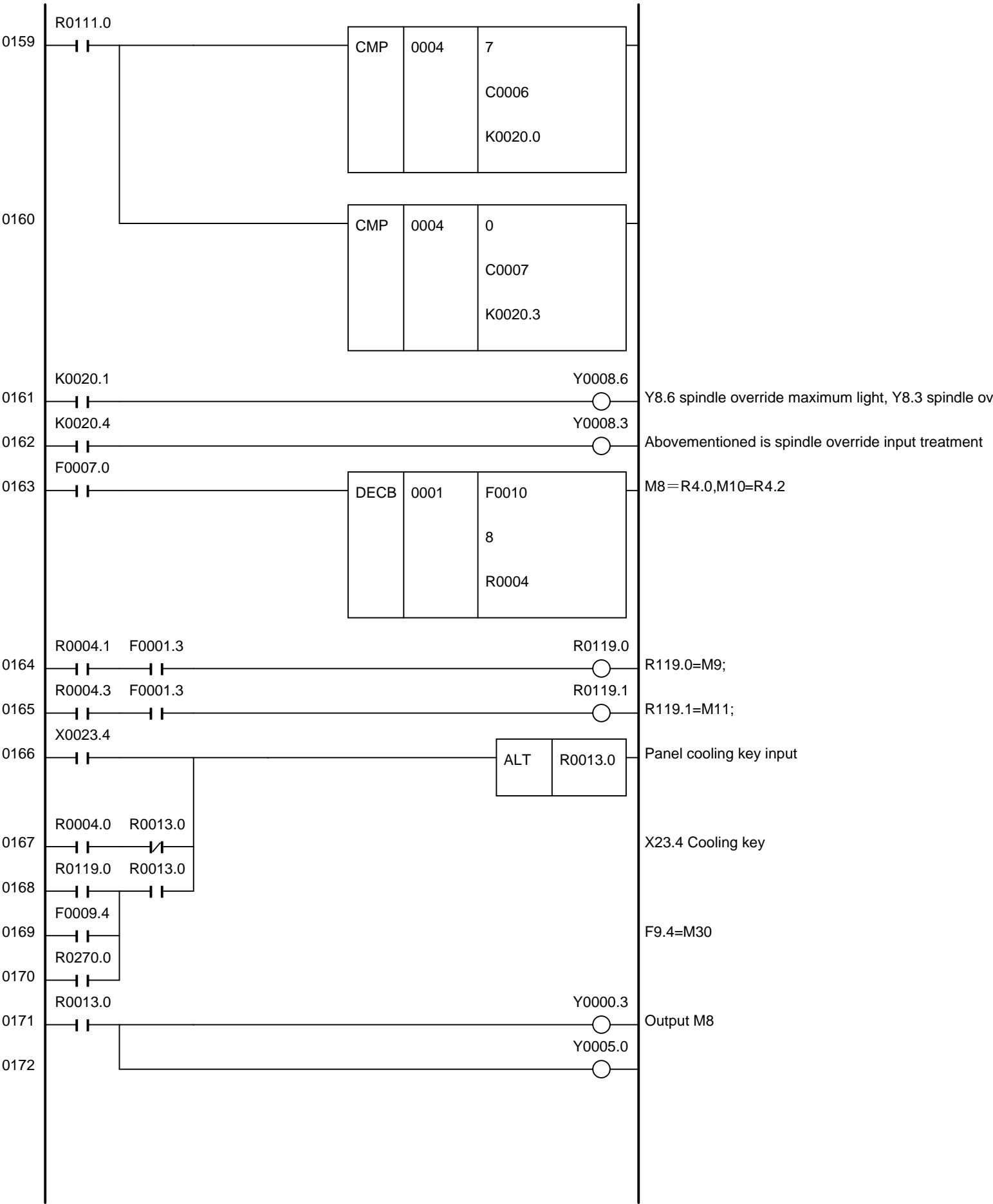


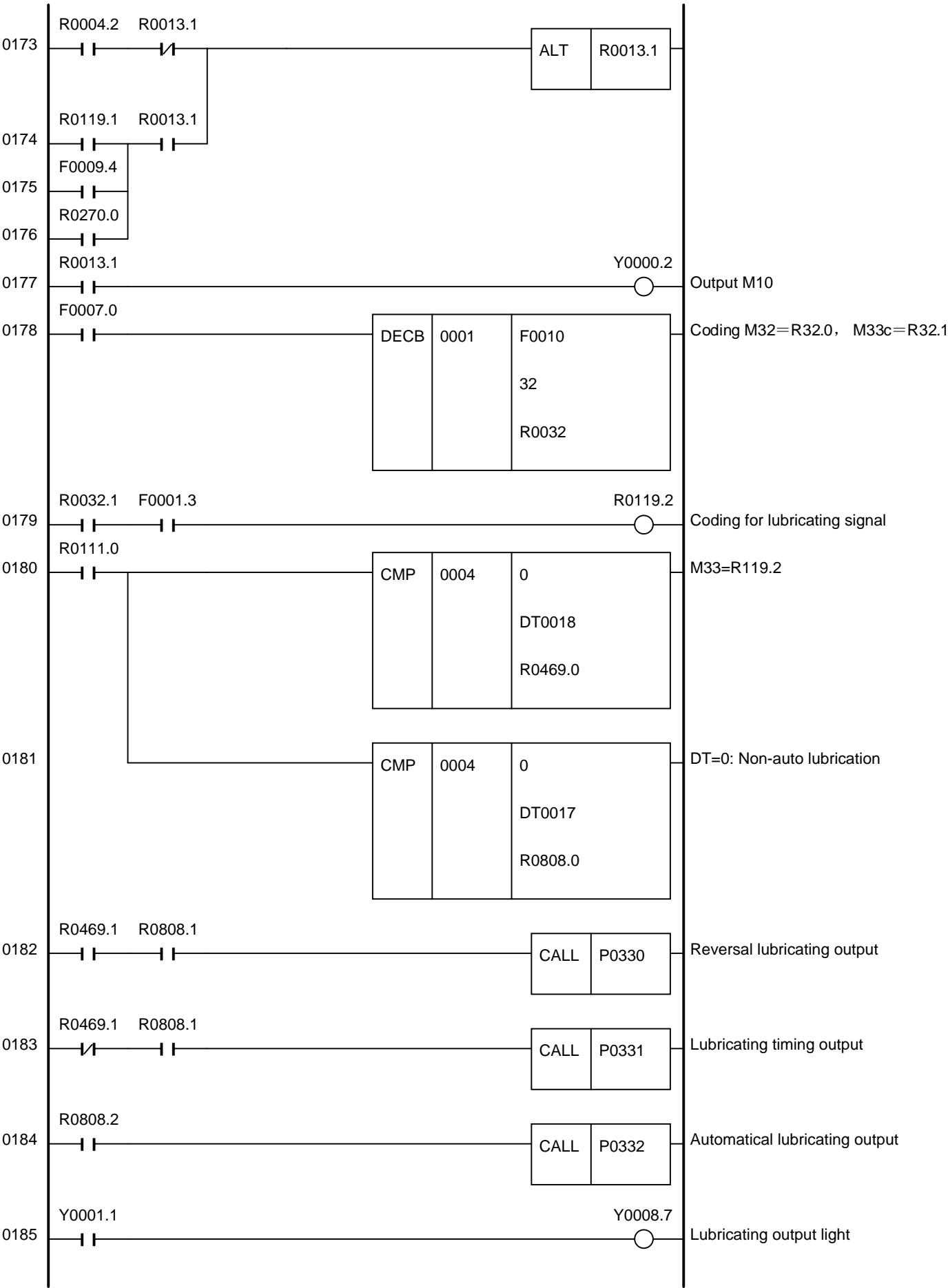


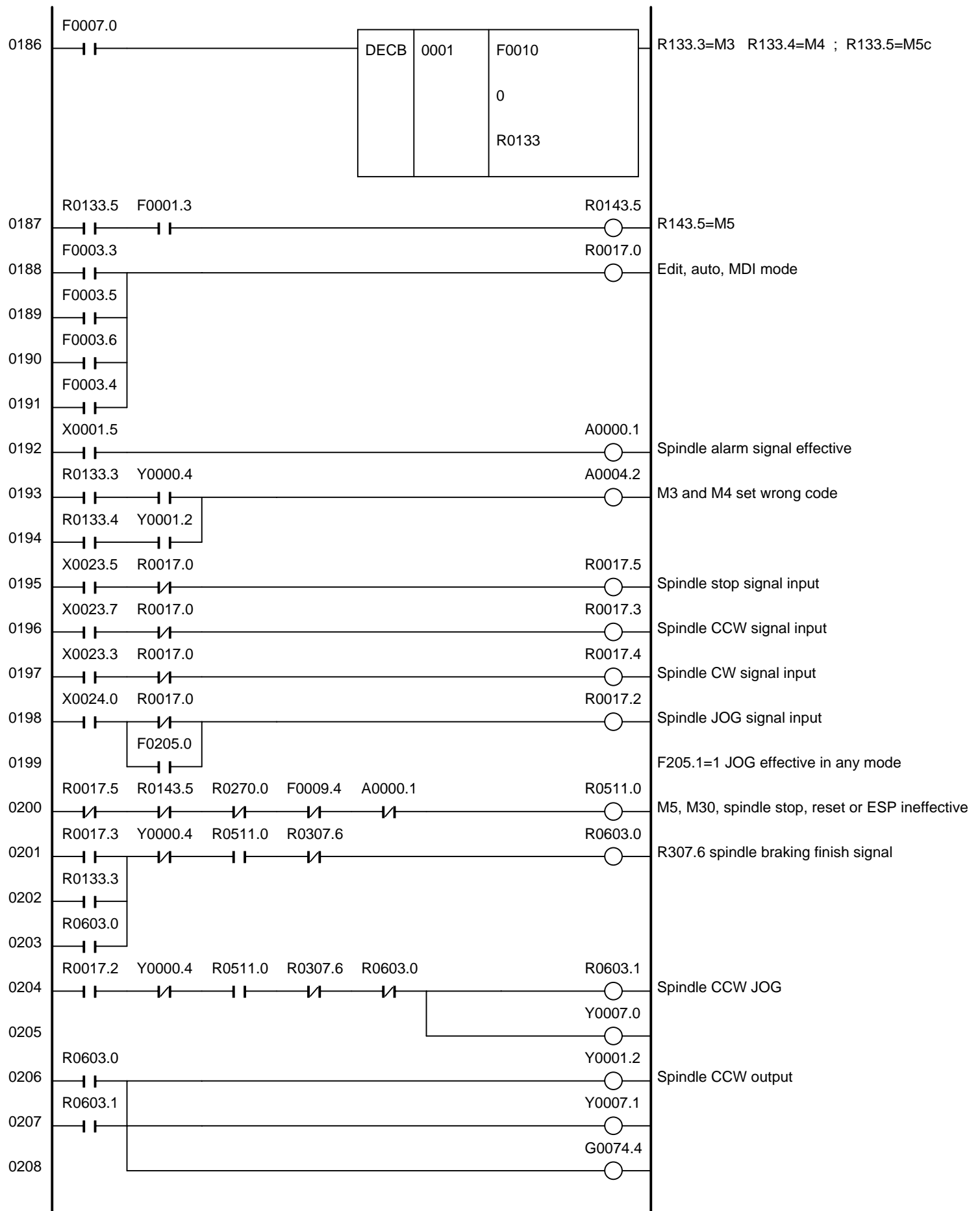


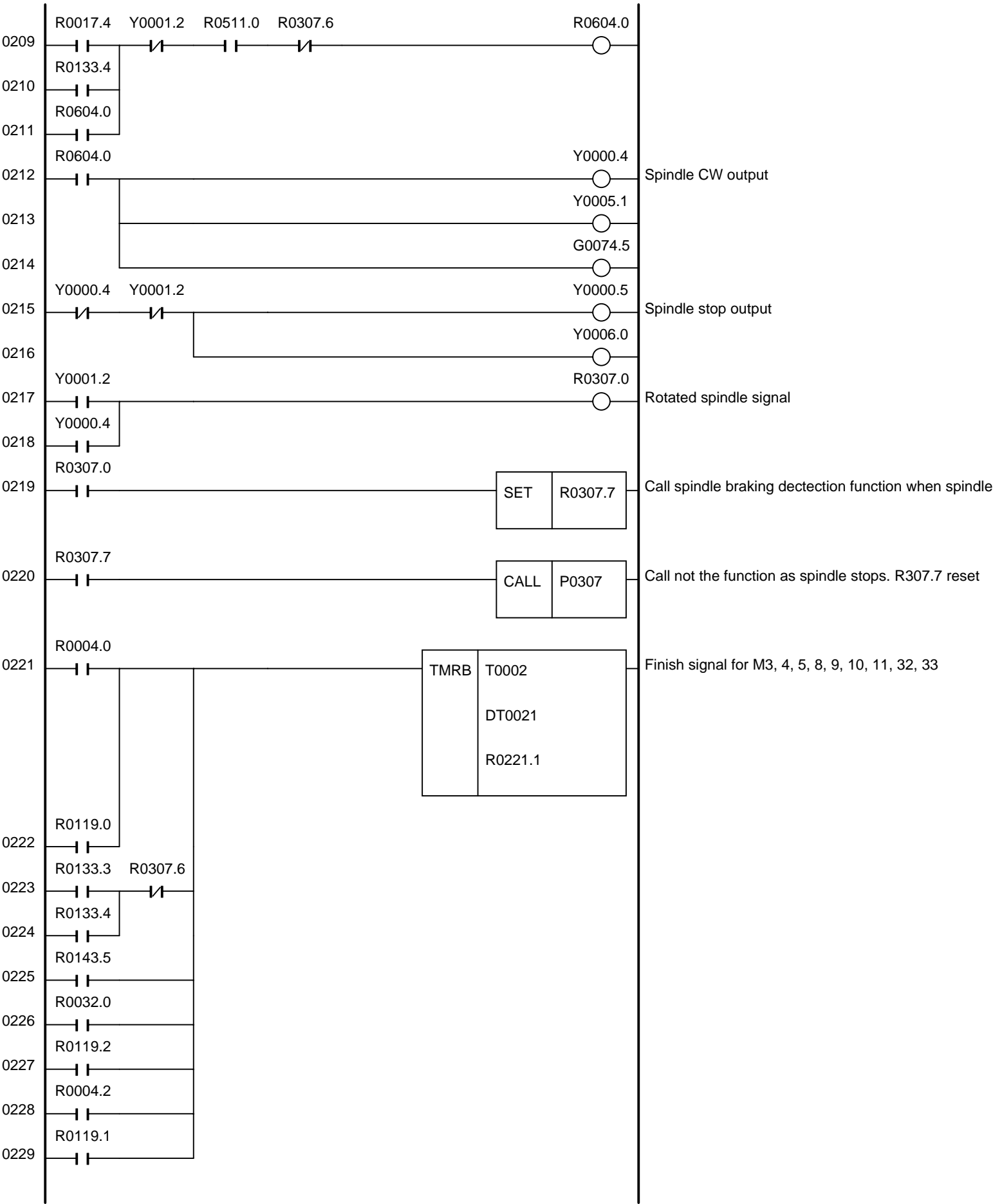


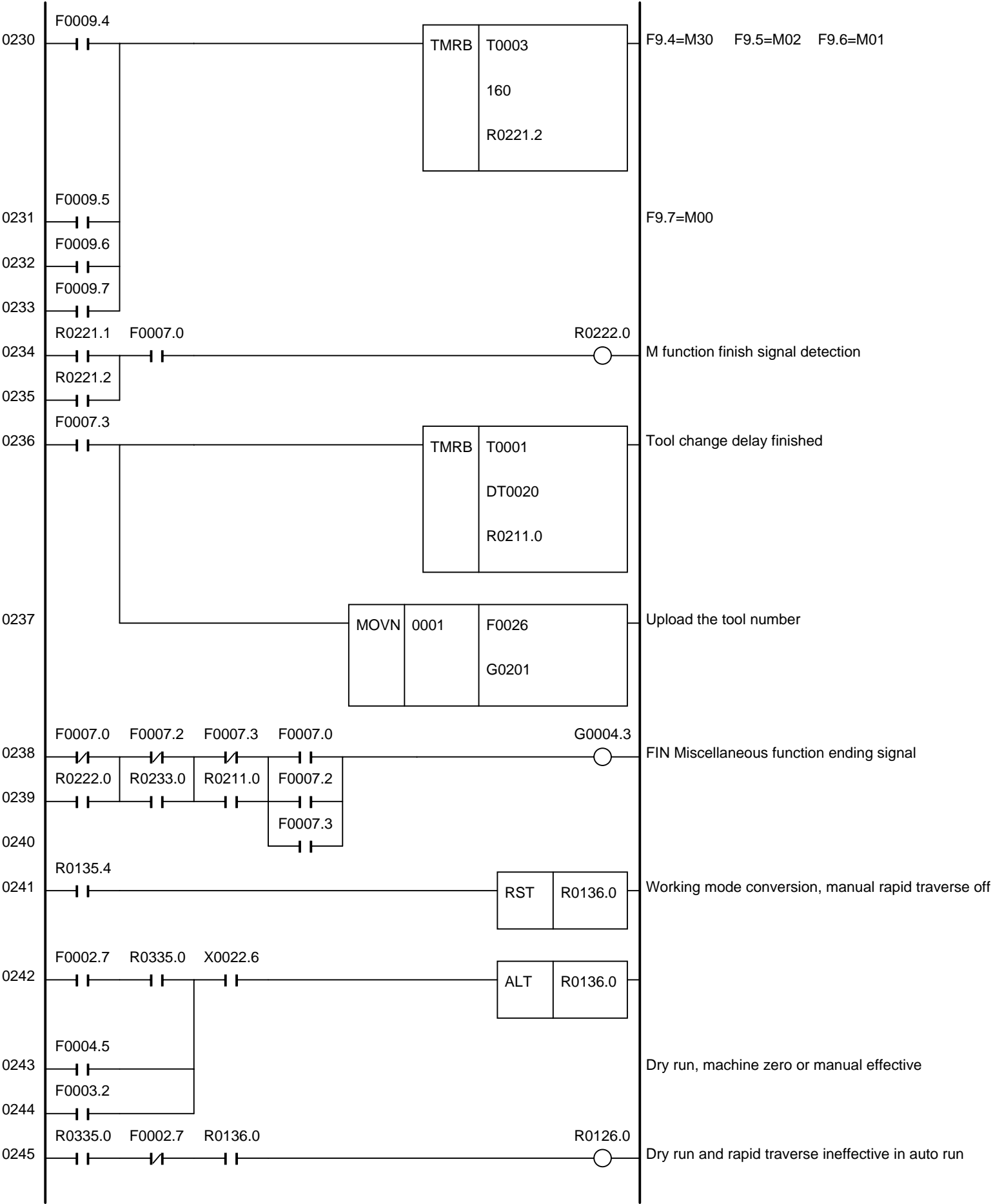


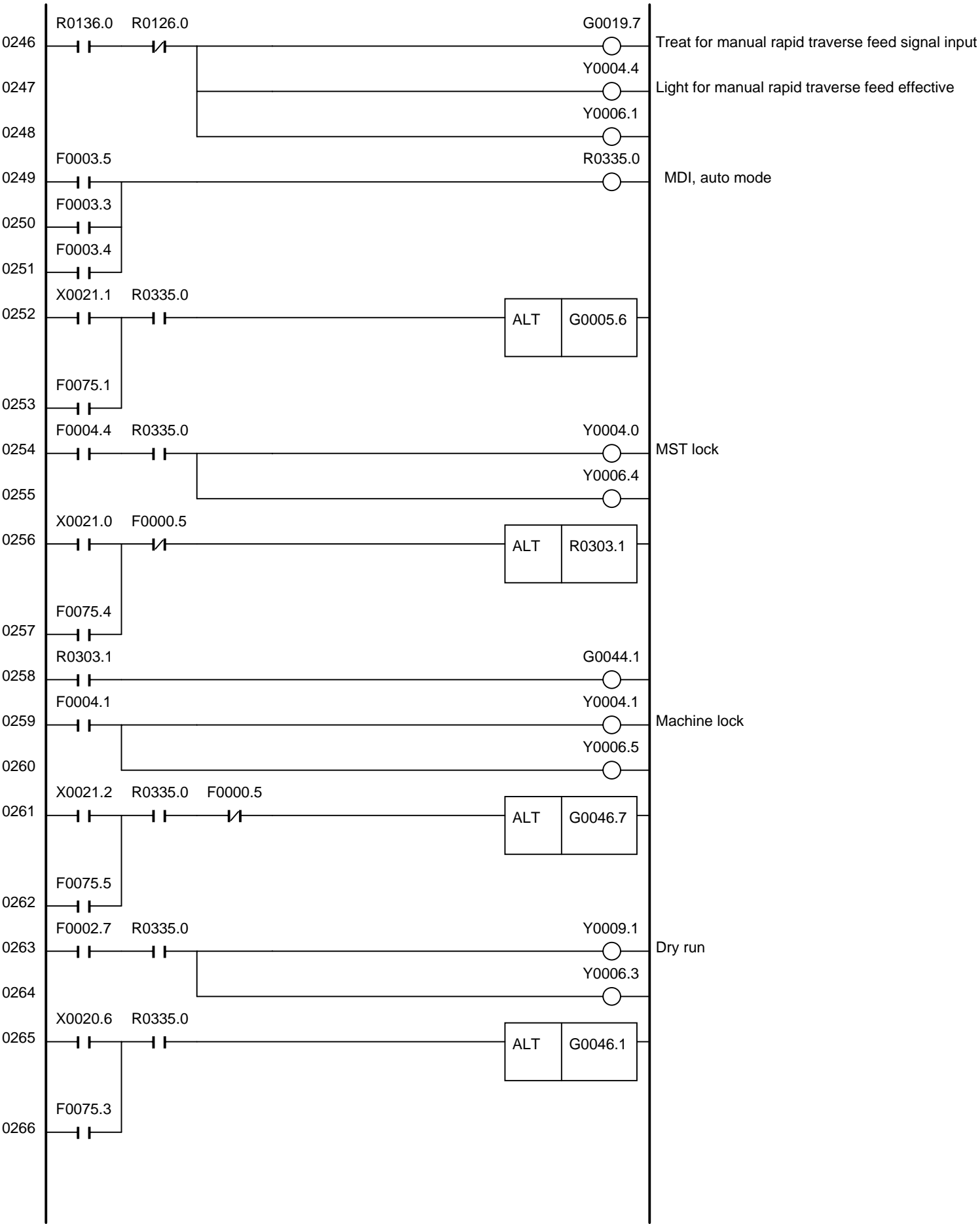


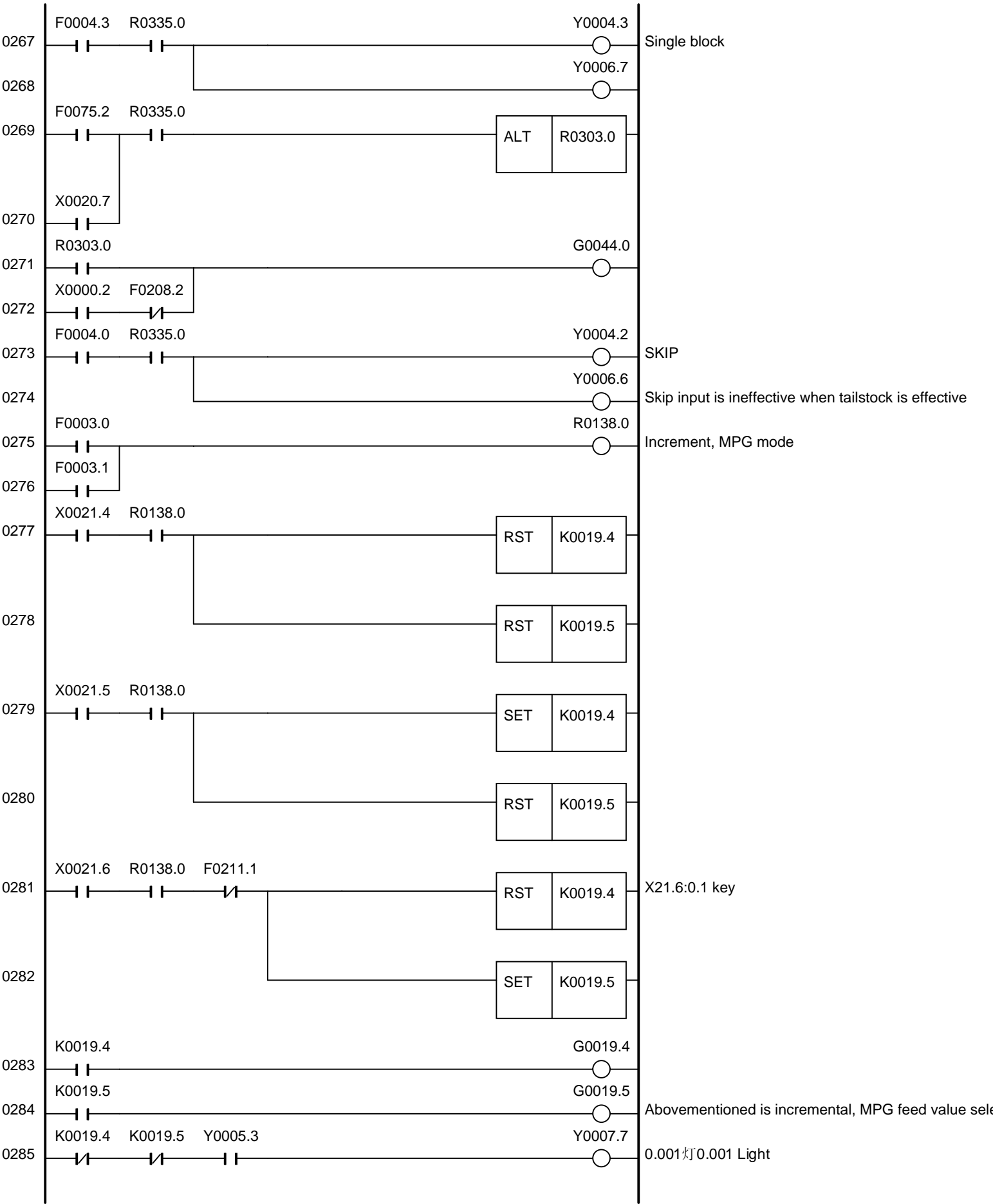


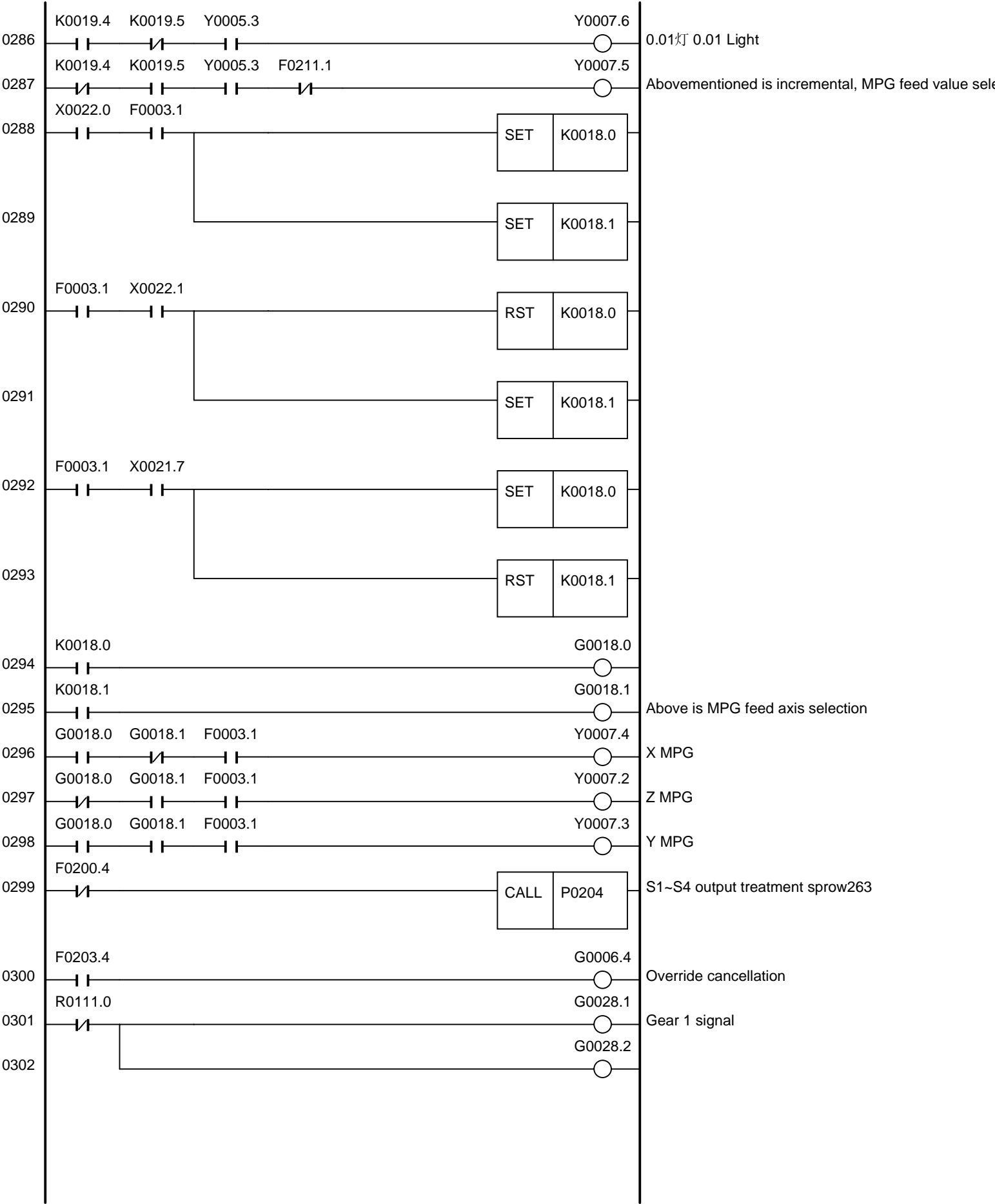


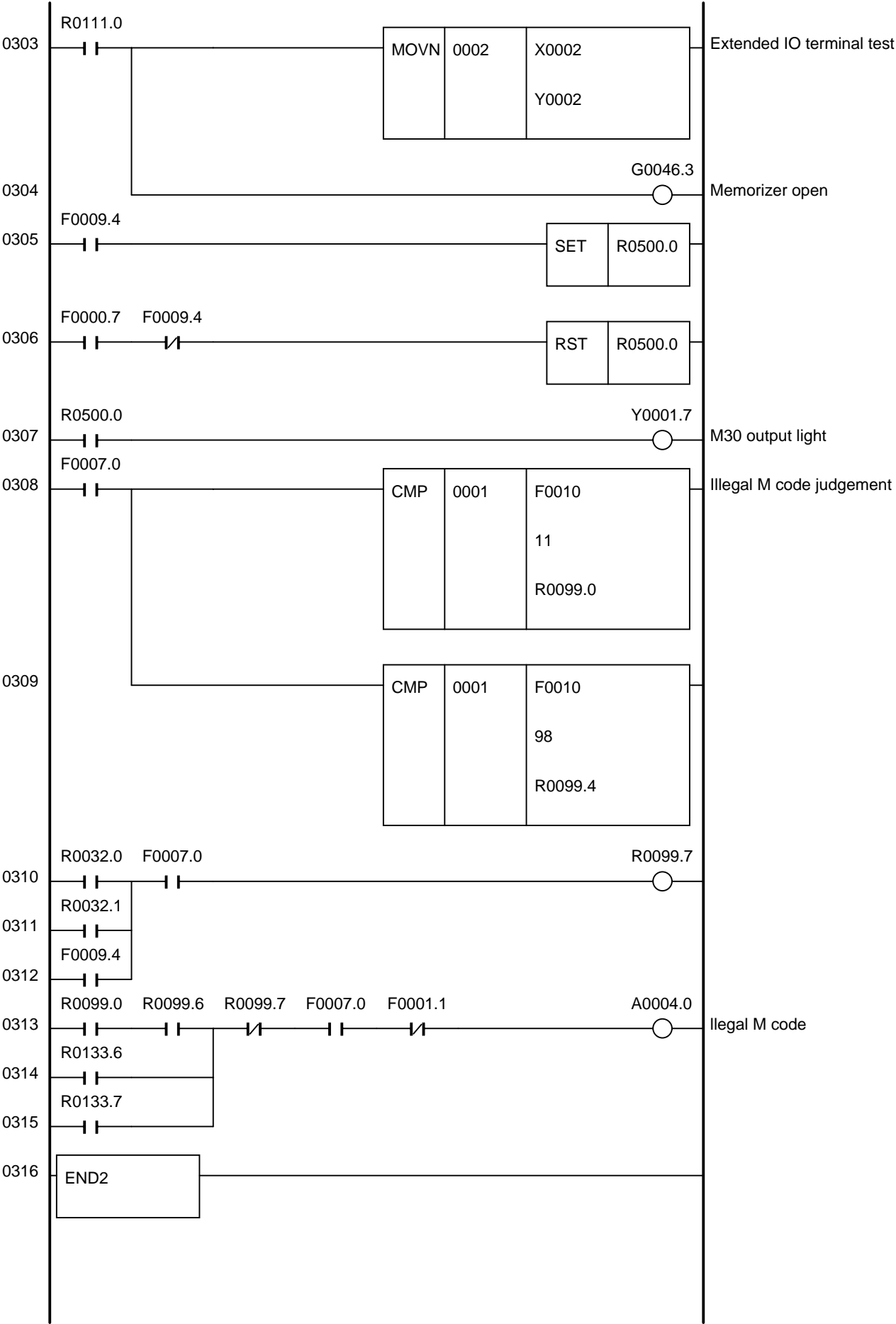


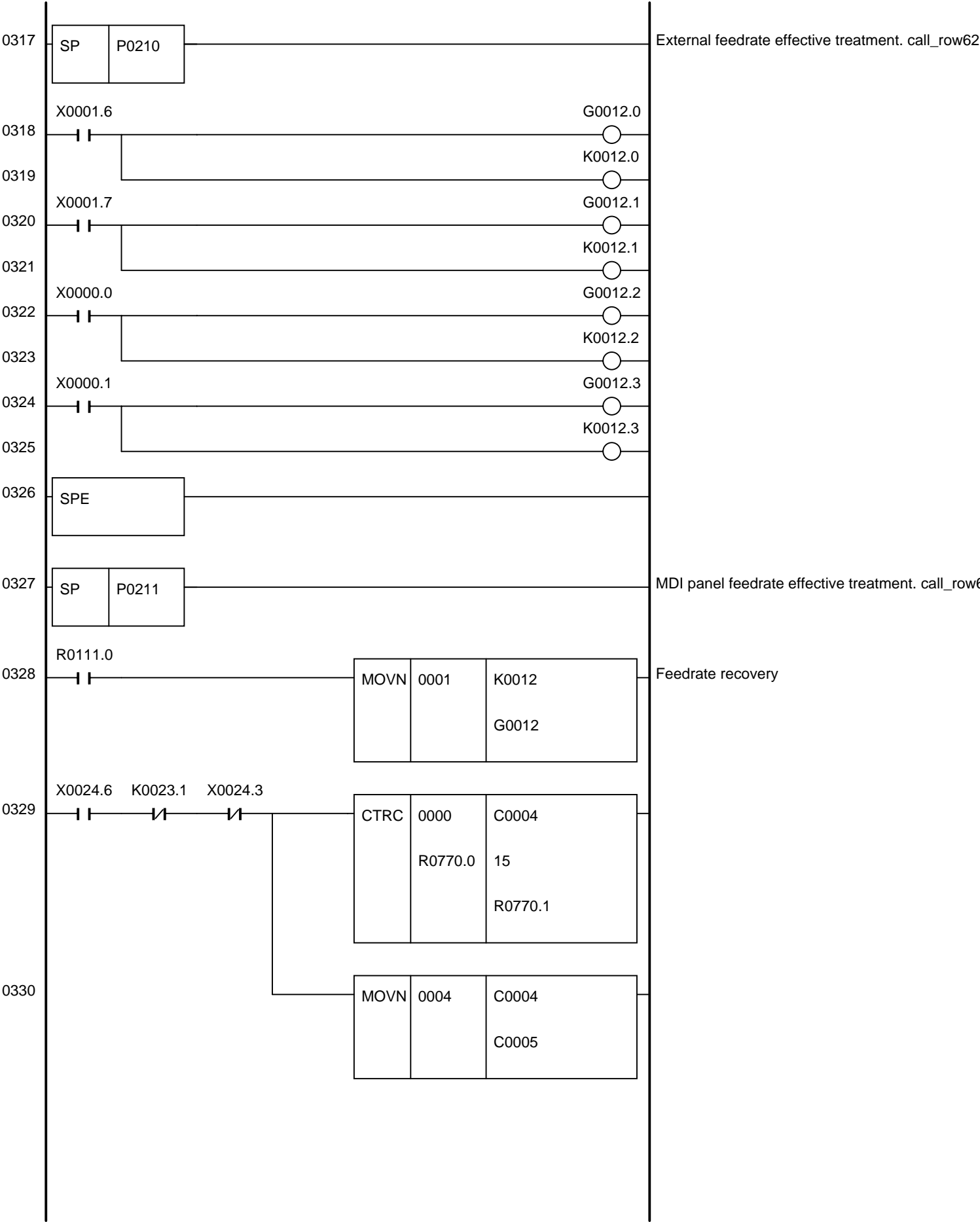


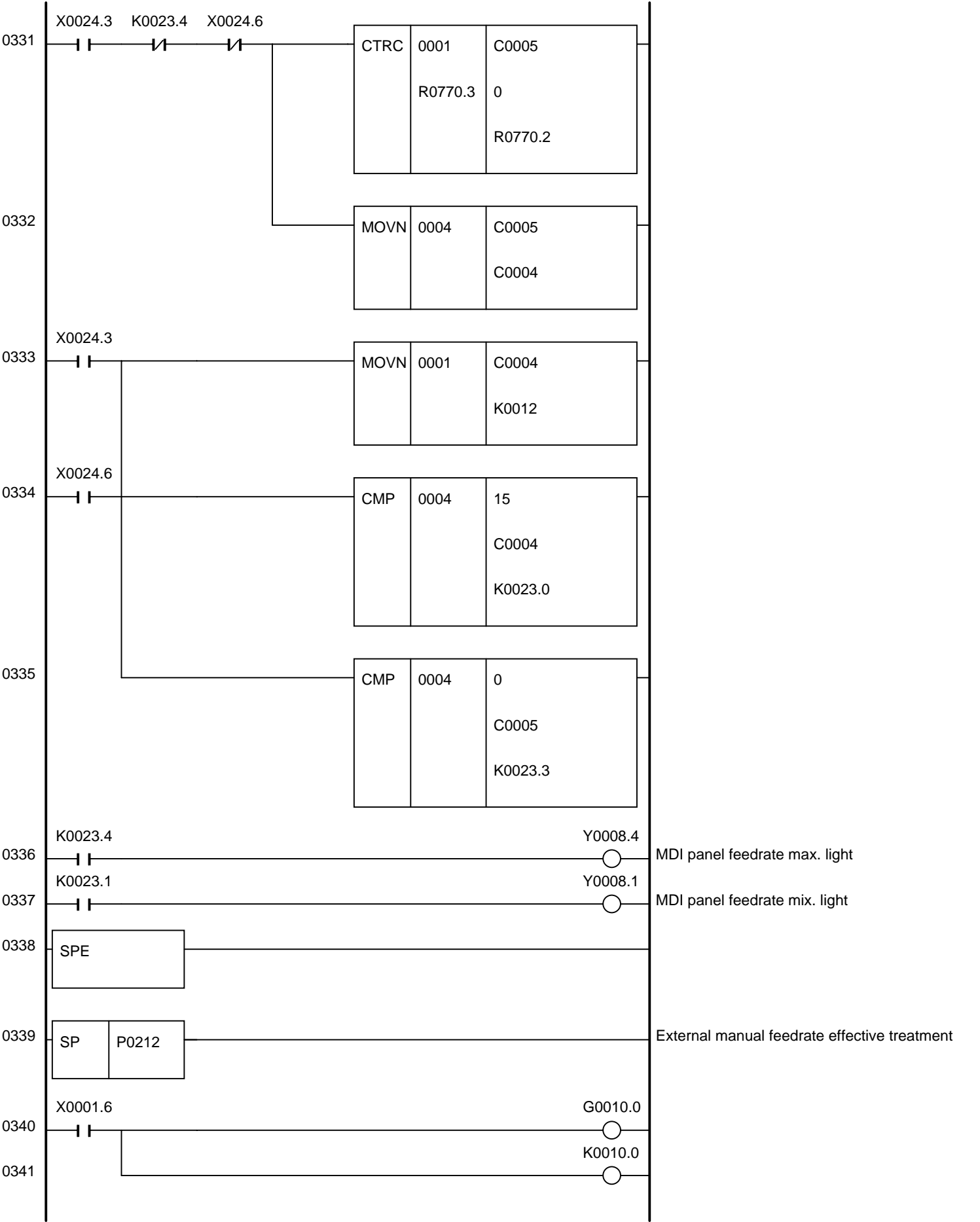


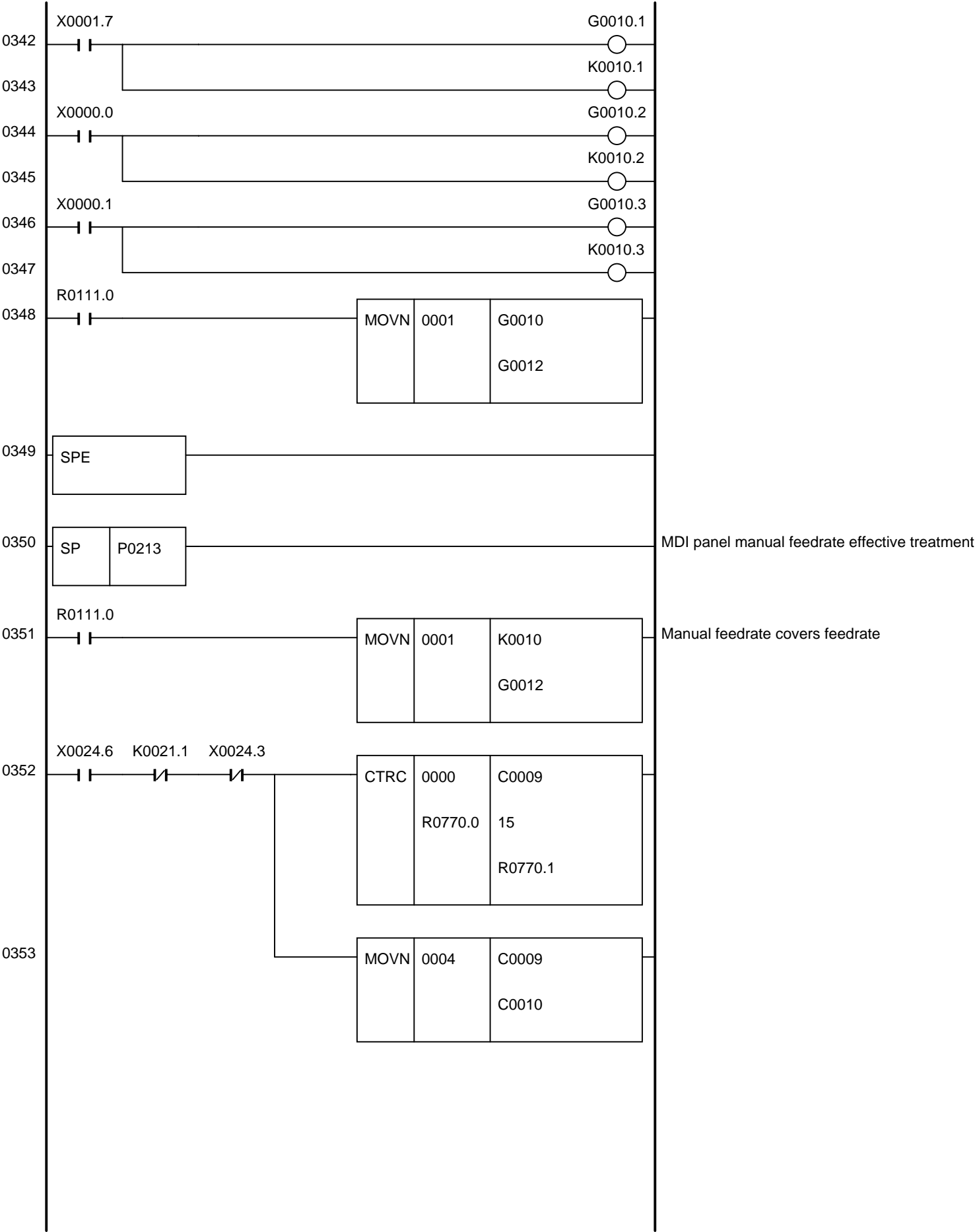


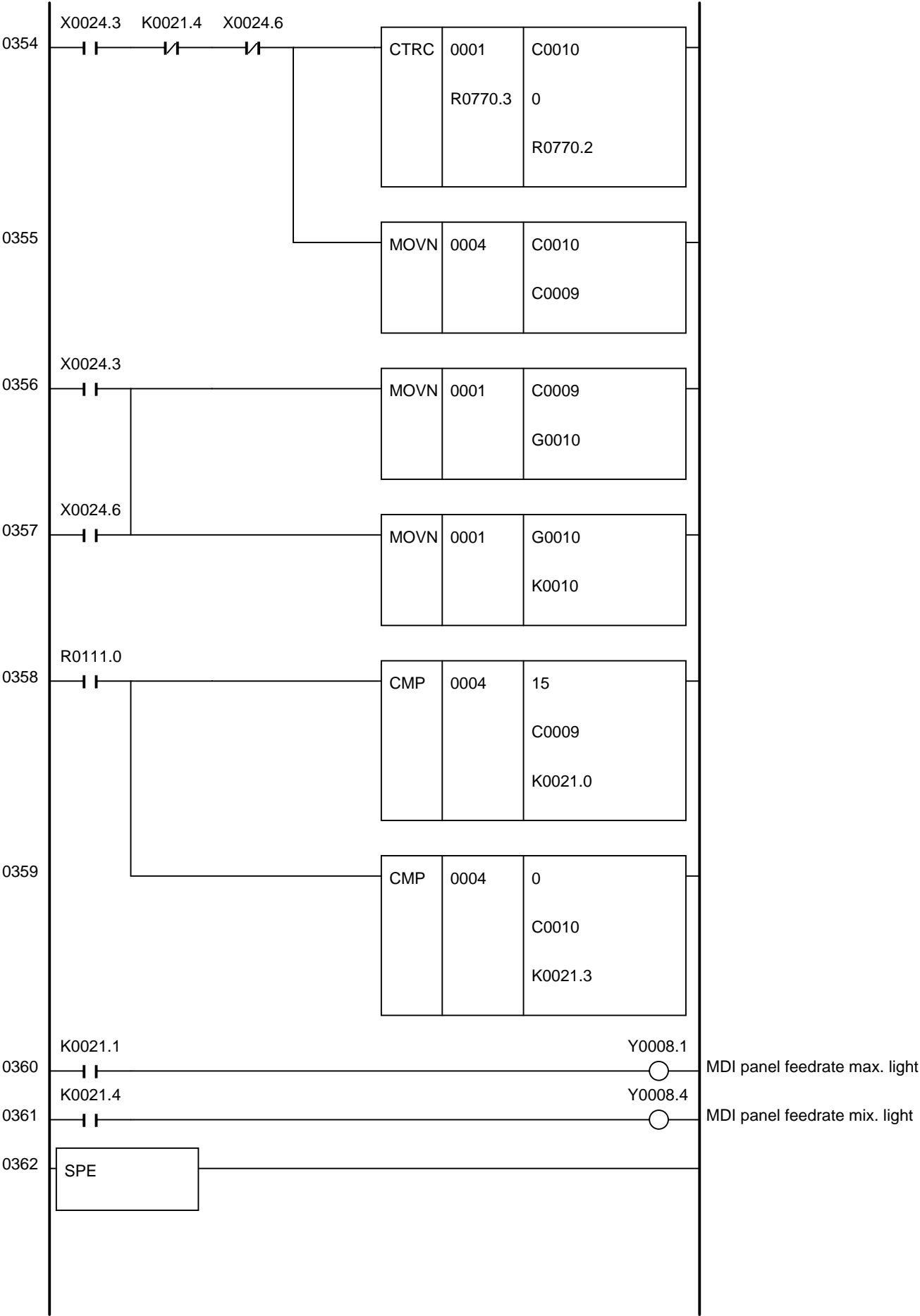


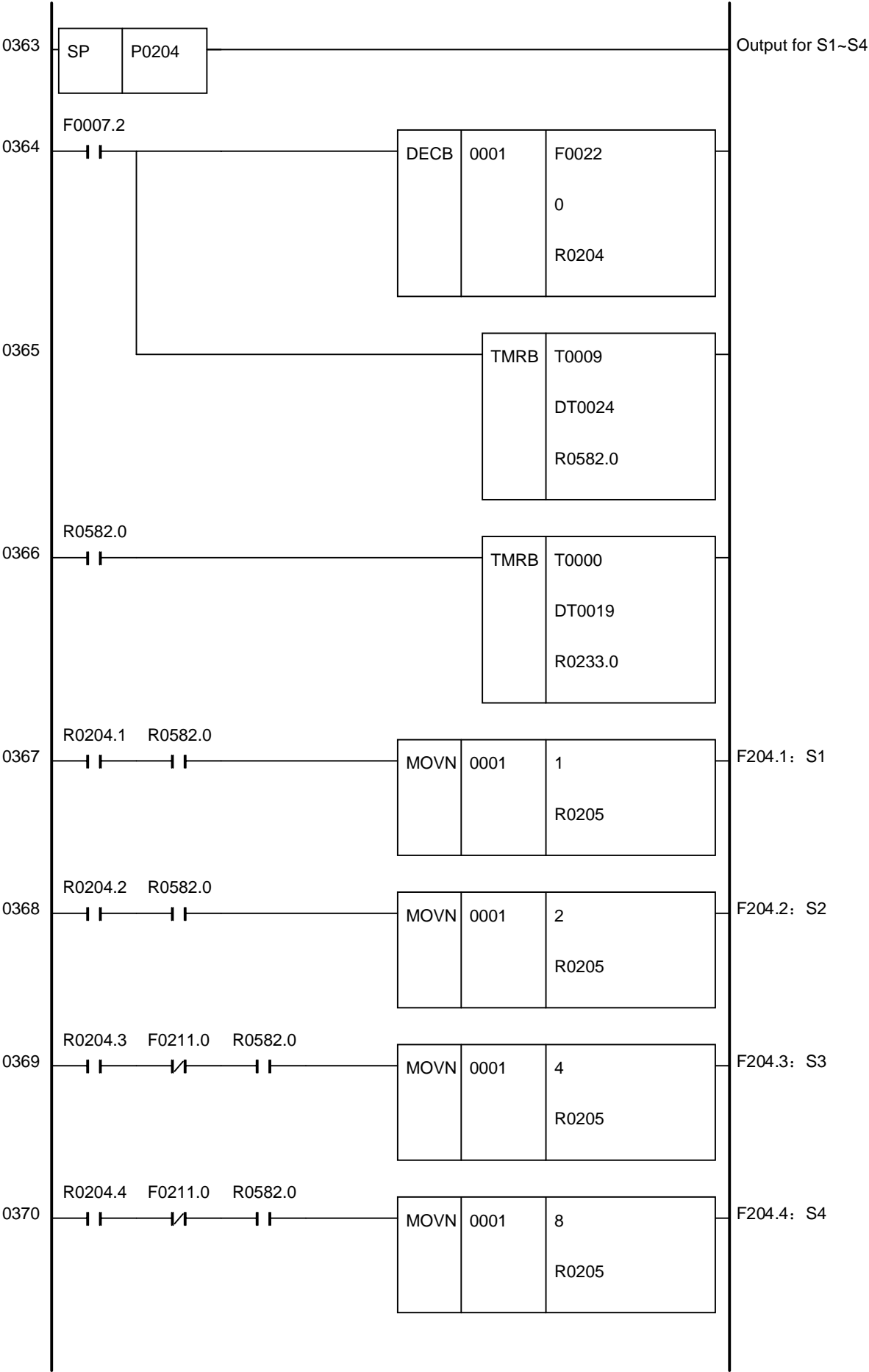


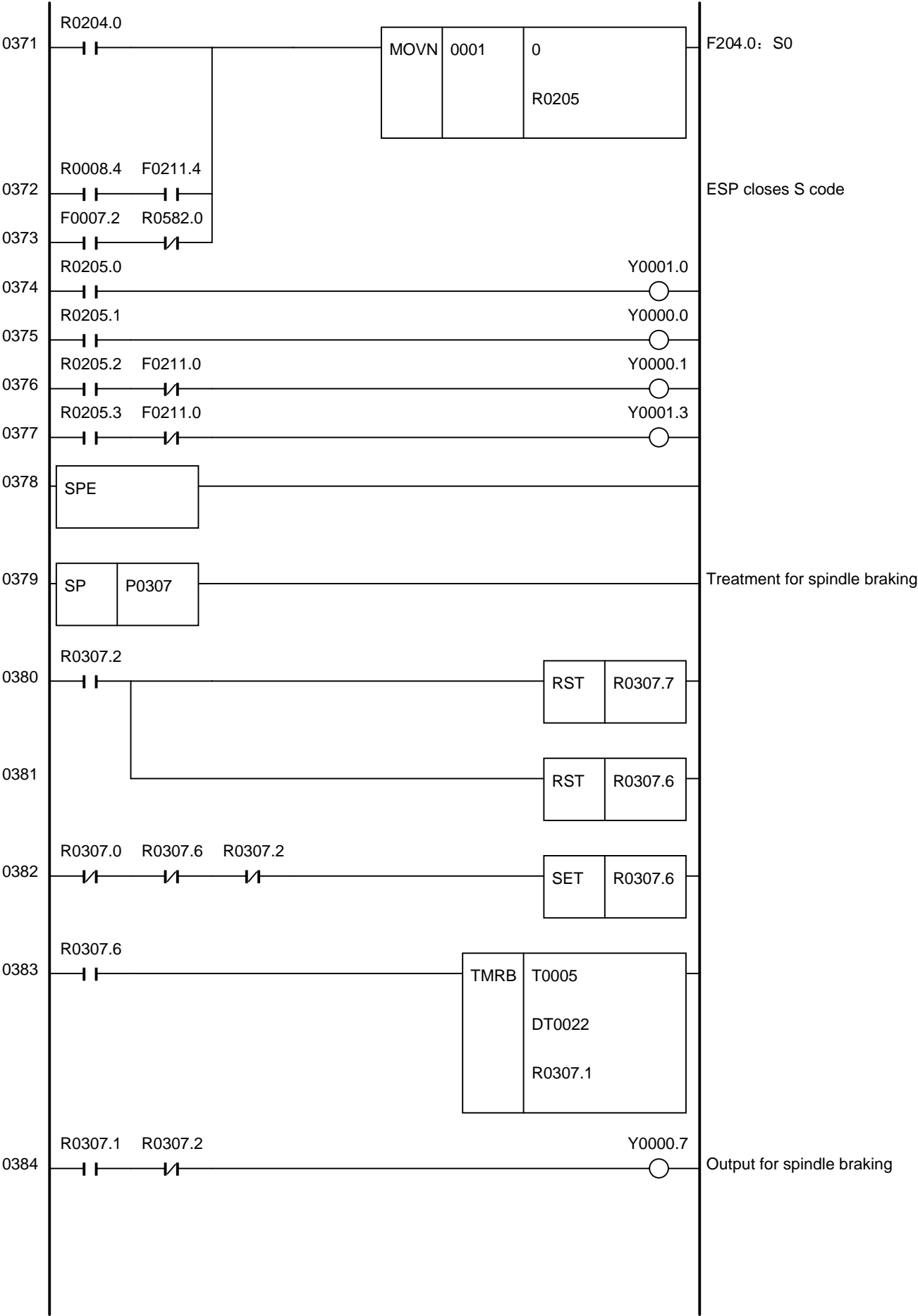


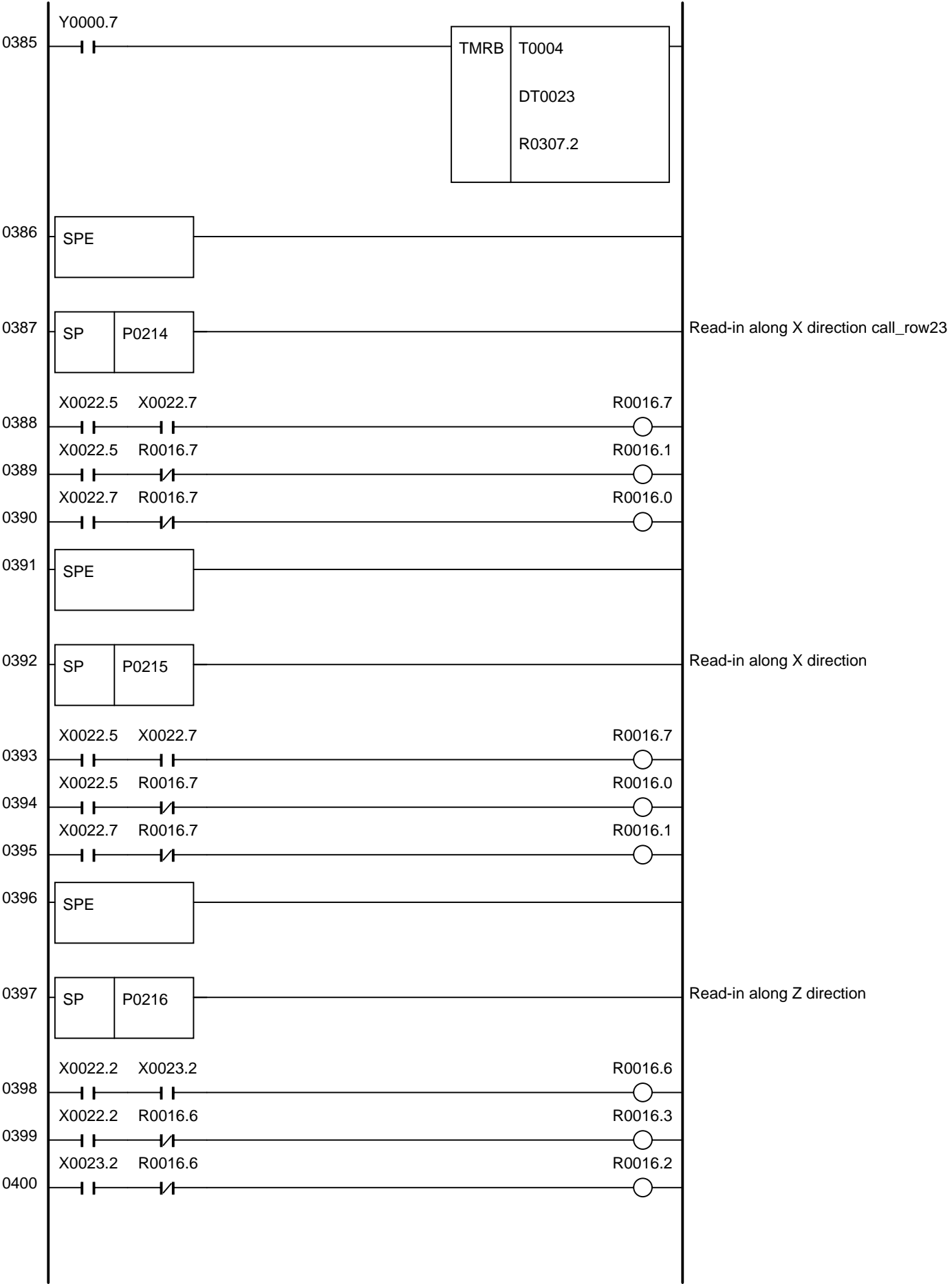


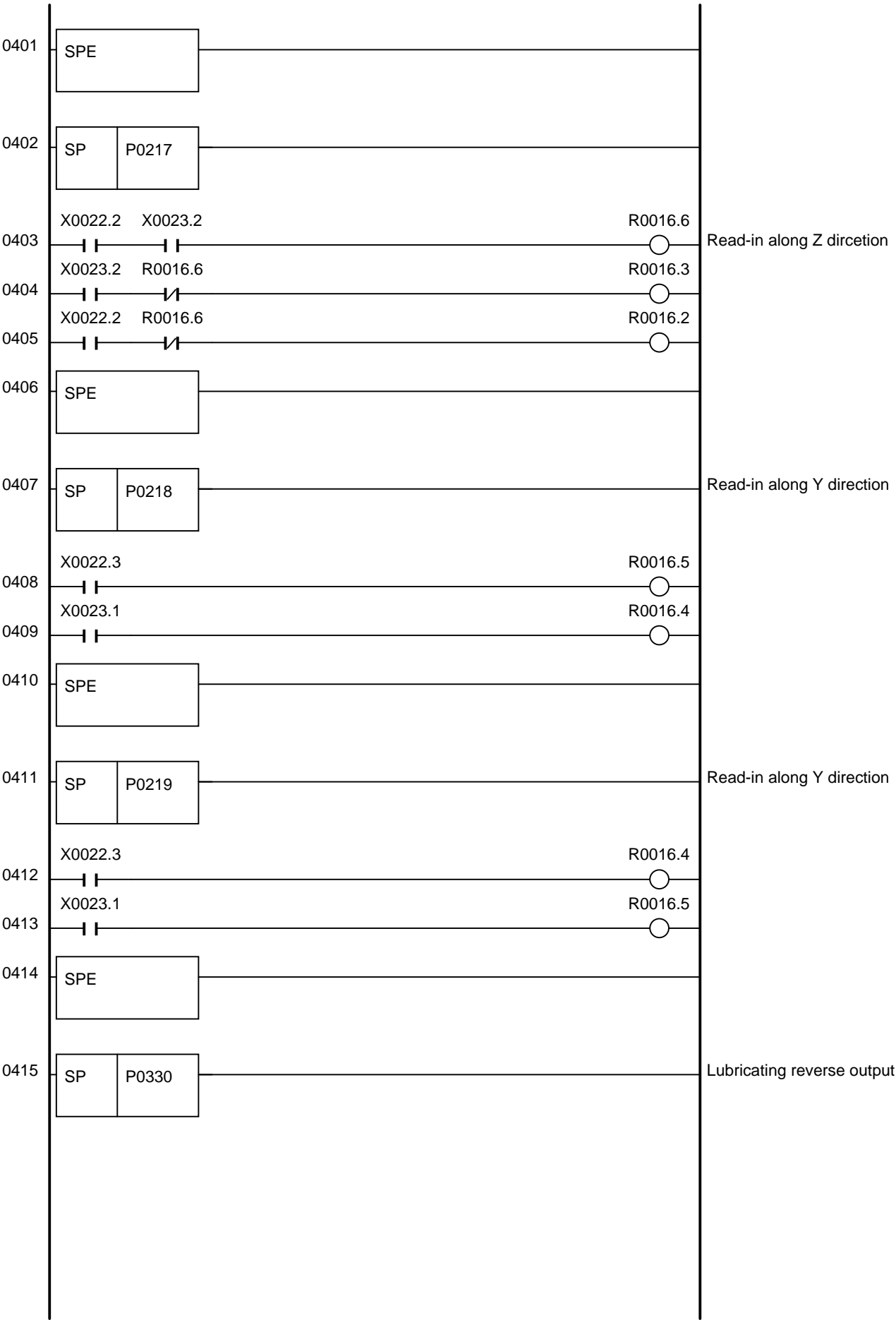


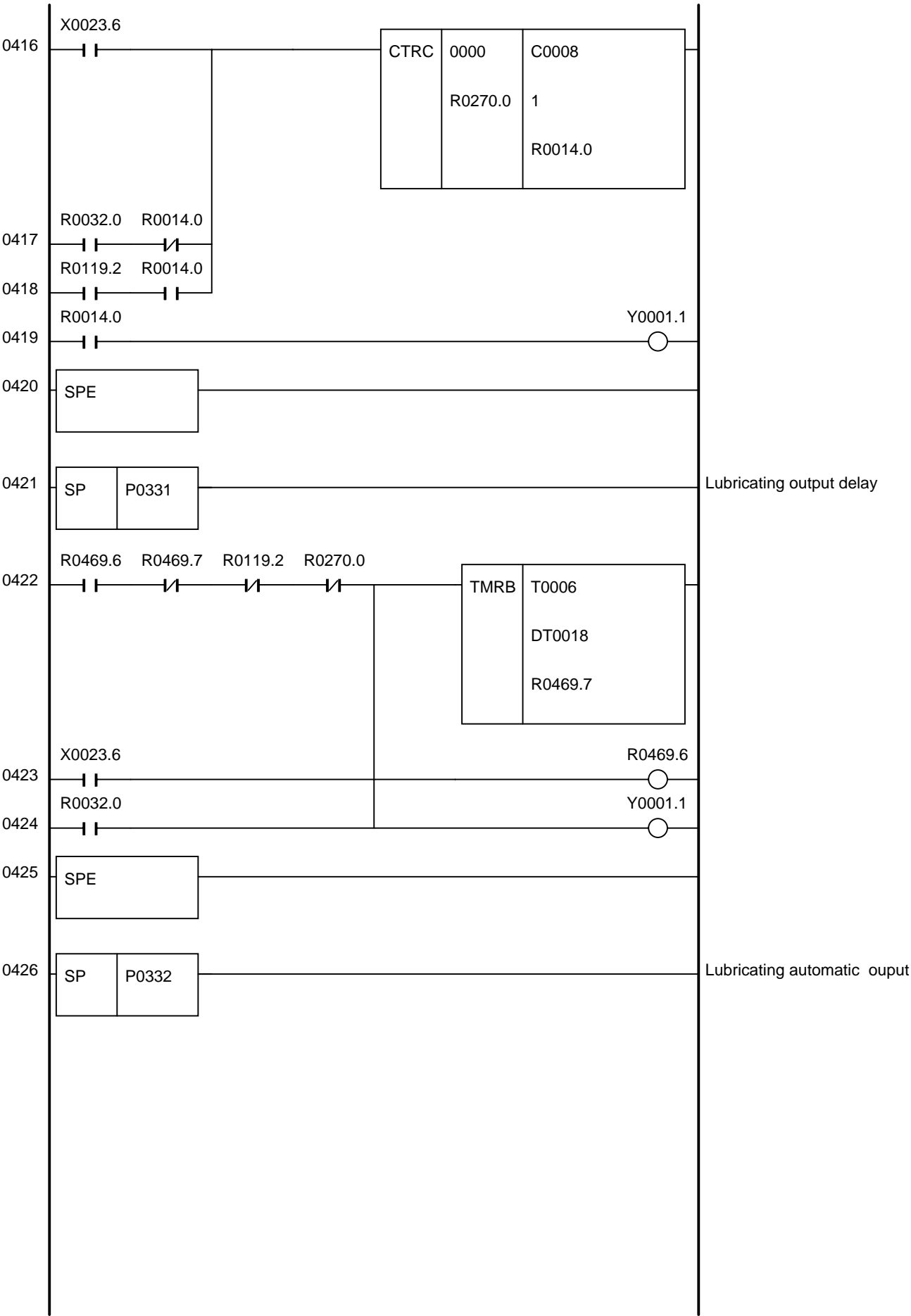


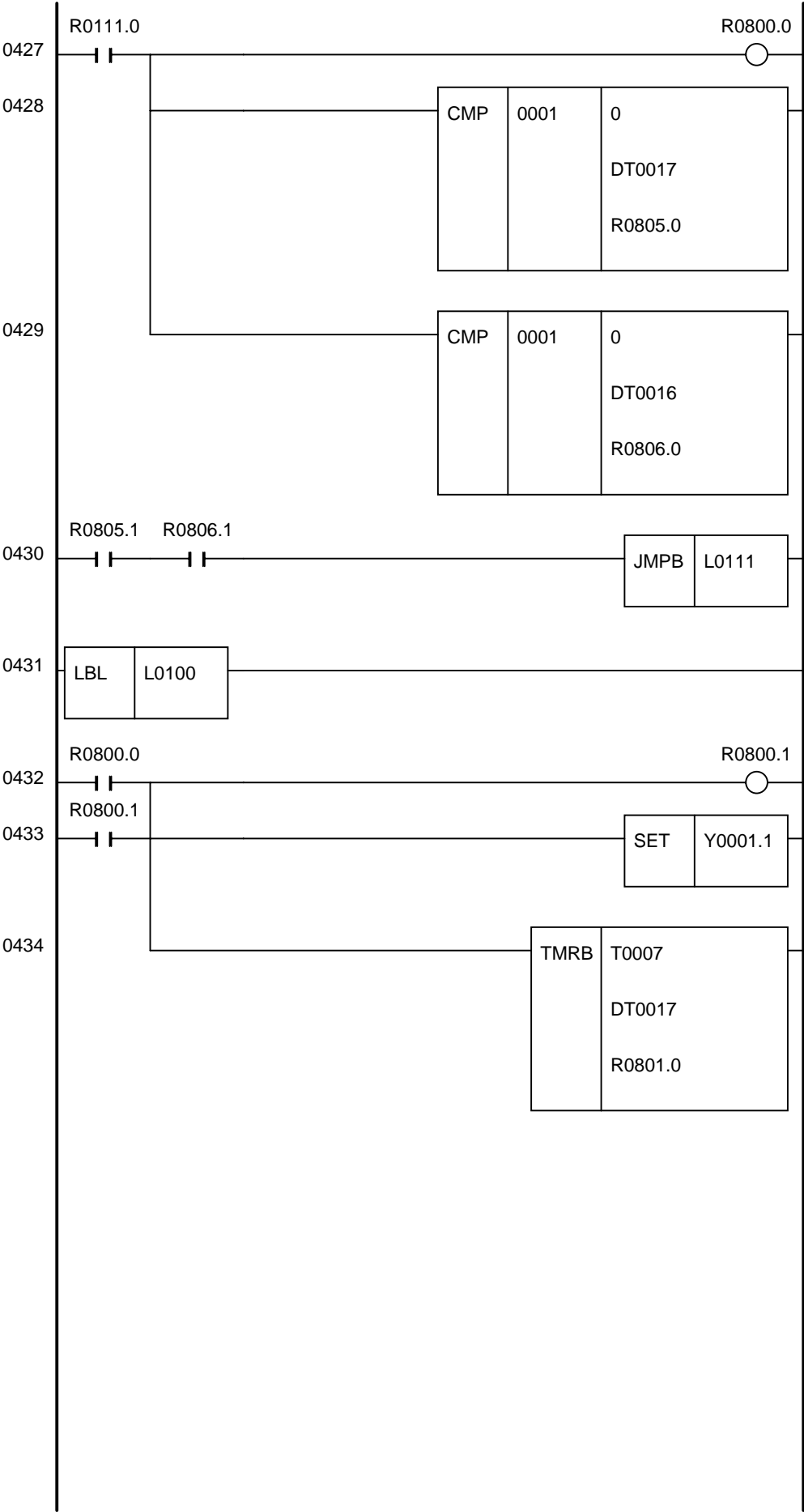


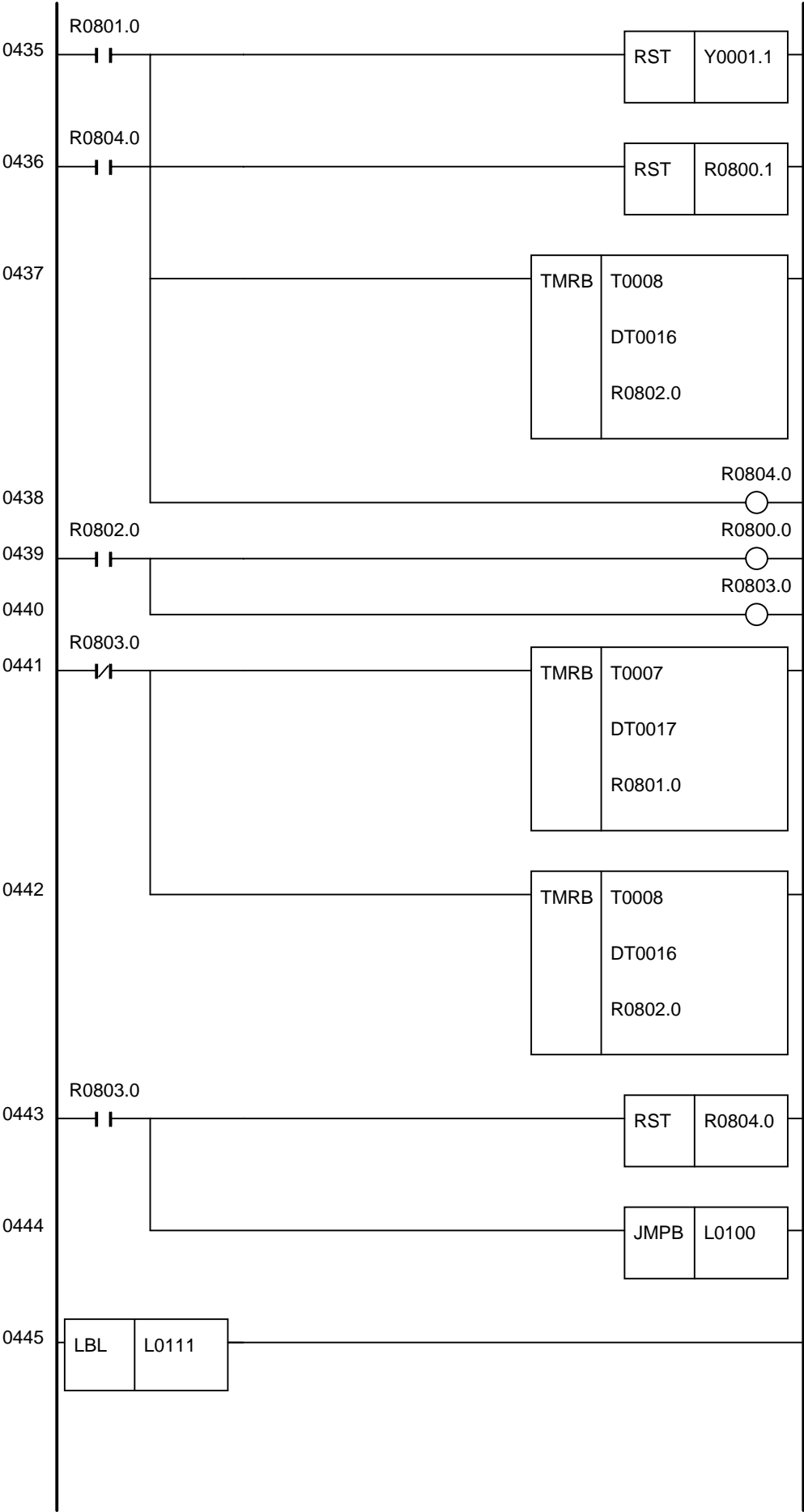












0446

SPE



广州数控设备有限公司
GSK CNC EQUIPMENT CO., LTD.

Add: No.52, 1st . Street, Luochong North Road, Luochongwei, Guangzhou, 510165, China

Website: <http://www.gsk.com.cn>

E-mail: gsk@gsk.com.cn

Tel: (00) 86-20-81796410/81797922

Fax: (00) 86-20-81993683

All specifications and designs are subject to change without notice. Aug. 2007/Edition 2

Mar. 2008/Printing 1