GSK980TD

Turning Machine CNC System

User Manual

广州数控设备有限公司

GSK CNC EQUIPMENT CO., LTD.

Warning!

- Please read the user manual and a user manual from machine manufacturer completely before installation, programming and operation, and operate the system and machine according to user manuals, otherwise which may damage the system and machine, workpiece and even injure the operator.

Notice!

- Functions, technical indexes described in the user manual are for the system. Actual functions and technical performance of CNC machine tool with the system are defined by machine manufacturer, and refer to its user manual；
- The system is employed with intergrated machine control panel and press keys on machine control panel are defined by PLC program. Functions of press keys in the user manual are for standard PLC program. Please notice it!
- Refer to user manual from machine manufacturer about functions and meanings of press keys on machine control panel.

This manual suits for software version:V06.03.23.

All specification and designs are subject to change without notice.

Notice

■ Delivery and storage

- There are 6-layer packing box at most in pile
- Never climb the packing box, neither stand on it, nor place heavy items on it
- Do not use cable connected with product to drag or move it
- Forbid crash, hurt panel and display
- Packing box is protected from damp, sun and rain

■ Open packing box to check

- Ensure things in packing box are the required ones
- Ensure it is not damaged in delivery
- Ensure things in packing box are these of order
- Contact with us in time if its type is inconsistent with the order, there is short of accessories, or it is damaged in delivery

■ Connection

- Only qualified persons can connect the system or check the connection.
- The system must be earthed, its resistance must be less than 4 Ω and the ground wire cannot be replaced by zero wire
- Connection must be correct and firm to avoid the product to be damaged or other unexpected result
- Connect with surge diode in the specified direction to avoid to damage the system
- Switch off power supply before pulling out plug or opening electric box

■ Troubleshooting

- Switch off power supply before troubleshooting or changing components
- Troubleshoot and then startup the system when there is short circuit or overload
- Do not switch off it and a meantime is 1 minute at least after it is switched on again.

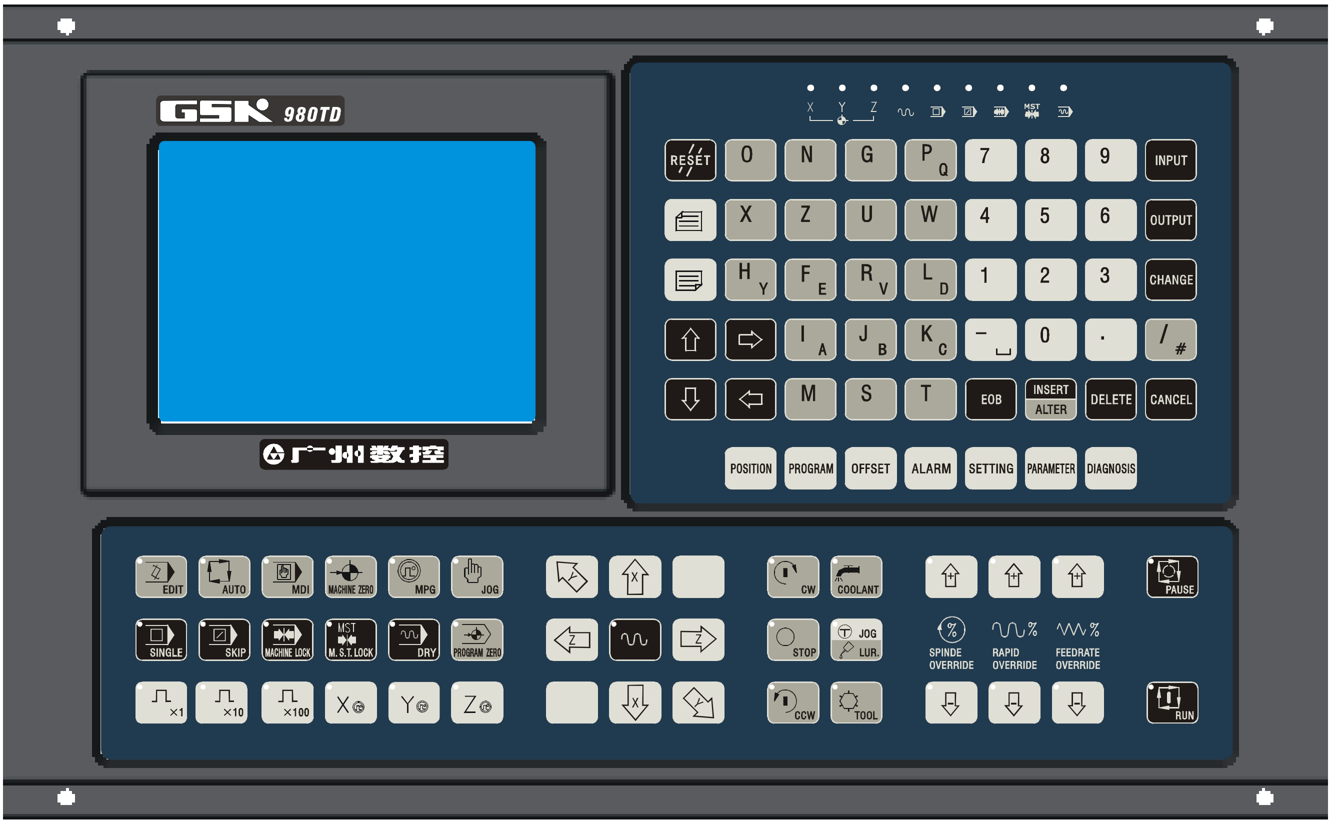
BOOK 1

PROGRAMMING

Chapter 1 PROGRAMMING FUNDAMENTALS

1.1 INTRODUCTION of GSK980TD

With 32-bit high performance CPU and super-large-scale programmable FPGA, the new generation widespread GSK980TD Turning Machine CNC System developed by us (GSK CNC Equipment Co., Ltd.) is the upgraded product of GSK980TA, applying the real time multitasking control and hardware interpolation technology to realize μm-level precise motion and PLC logic control.



Technical characteristics:

- ✓ Link axes (X, Z), μm-level interpolation precision and max. rapid traverse speed 16 m/min(option: 30m/min)
- ✓ Embedded PLC to control various of automatic toolposts and spindle automatic shifting gear, edit, transmit and download ladder diagrams; expendable I/O interfaces(option function)
- ✓ Pitch error compensation, backlash compensation, tool length compensation and tool nose radius compensation
- ✓ S, exponential acceleration/deceleration control to meet high speed and high precision machining
- ✓ Tapping to machine metric/inch single/multiple straight, taper thread, end face thread, variable pitch thread, high speed thread run out with set retraction distance, angle and speed

✓ Chinese and English display interface selected by parameters

✓ Large memory capacity(6144KB,384 part programs) with full screen edit

✓ Convenient management for the system with multilevel operation password

✓ Bidirectional communication between CNC and PC, CNC and CNC; communication upgrading CNC software and PLC programs

✓ Installing dimension, electric interfaces, instruction system and operating windows being compatible with those of GSK980TA Turning CNC System

Technical specifications

<table>
<tr><td rowspan="9">Motion control</td><td>Controllable axes: 2(X, Z); simultaneous controllable axes: 2 (X, Z)</td></tr>
<tr><td>Interpolation: linear, arc interpolation in X, Z direction</td></tr>
<tr><td>Dimension for programs: -9999.999～9999.999mm; min. unit: 0.001mm</td></tr>
<tr><td>Electronic gear：instruction multiplying 1～32767 and dividing 1～32767</td></tr>
<tr><td>Rapid traverse speed: max. 16000mm/min(option:30000mm/min)
Rapid override: time real tuning F0, 25%, 50%,100%</td></tr>
<tr><td>Cutting feedrate: max.8000mm/min(option:15000mm/min) or 500mm/rev (feedrate per rev)
Feedrate override: 16 steps real time tuning for 0～150%</td></tr>
<tr><td>Manual feedrate: 16 steps real time tuning for 0～1260mm/min</td></tr>
<tr><td>Handwheel feedrate: 0.001, 0.01, 0.1mm</td></tr>
<tr><td>Acceleration/deceleration: S acceleration/deceleration for rapid traverse movement and exponential acceleration/deceleration for cutting feed</td></tr>
<tr><td>G
instructions</td><td>28 kinds of G instructions: G00, G01, G02, G03, G04, G28, G32, G33, G34, G40, G41, G42, G50, G65, G70, G71, G72, G73, G74, G75, G76, G90, G92, G94, G96, G97, G98, G99 and macro instruction G65 to execute 27 kinds of calculation, logic operation and program skipping</td></tr>
<tr><td rowspan="3">Thread machining</td><td>Tapping to machine metric/inch single/multiple straight thread, taper thread, end face thread, variable pitch thread. High speed thread run out with set retraction distance, angel and speed; pitch: 0.001～500mm or 0.06～25400 tooth/inch</td></tr>
<tr><td>Spindle encoder: lines can be set （100～5000p/r）</td></tr>
<tr><td>Drive ratio between encoder and spindle: （1～255）:（1～255）</td></tr>
<tr><td rowspan="3">Precision compensation</td><td>Backlash compensation: （X, Z）0～2.000mm</td></tr>
<tr><td>Pitch error compensation: 255 compensation points with ±0.255mm × compensation override for each one in X, Z direction</td></tr>
<tr><td>Tool compensation: 32 groups tool length compensation, tool nose radius compensation (tool compensation C)
Toolsetting method: fixed-point toolsetting, trial cutting toolsetting
Tool compensation executing methods: traversing tool or coordinate offset</td></tr>
<tr><td rowspan="2">M instructions</td><td>M instructions(no repetition): M02, M30, M98, M99, M9000～M9999
Other M□□ instructions are defined and executed by PLC programs</td></tr>
<tr><td>M instructions defined by standard PLC program: M00, M03, M04, M05, M08, M09, M10, M11, M12, M13, M32, M33, M41, M42, M43, M44</td></tr>
</table>

T instruction	Most 32 tool selections（T01□□～T32□□）, the time sequence of tool change is defined by PLC programs. The tool selection is set to 1 and the tool change is not executed by PLC when the line-up toolpost is employed. The standard PLC programs is s is optional to 2～8 tool selections toolpost, clockwise rotation for selecting tools and counterclockwise rotation for clamping toolpost.	
Spindle speed	Speed switching value control: S□□ instruction is defined and executed by PLC programs, direct output of S1, S2, S3, S4 is controlled by the standard PLC programs and S0 is used for stopping output of S1, S2, S3, S4	
	Speed analog voltage control: S instructions specifying the spindle speed per minute or the cutting surface speed (constant surface speed control) , the system outputting 0～10V voltage to spindle converter, 4 gears spindle speed with stepless shifting gear	
PLC function	9 kinds of elementary instruction, 23 kinds of function instruction, 2 grades PLC program, max. 5000 steps and 2μs for each step, refresh cycle for the first grade program is 8ms, ladder diagram editing software, PLC program communication download	
	Integrated machine control panel: 41 input points (press keys), 42 output points (LED) Basic I/O interfaces: 16 input points /16 output points (optional I/O interface: 16 input points /16 output points)	
Displaying window	Display: 320×240 lattice, 5.7" monochrome liquid crystal display(LCD), CCFL in a poor light	
	Display method: Chinese or English window is set by parameter, displaying machining path of workpiece	
Program editing	Program capacity: 6144KB, max. 384 programs, supporting user macro program calling and four-embedded subprogram	
	Editing method: incremental coordinates, absolute coordinate and compound coordinates programming with full screen edit	
Communication	Bidirectional communication for programs and parameters between CNC and PC, CNC and CNC; communication upgrading and downloading CNC software and PLC programs	
Optional driving	DA98 Series Digital AC Servo or DY3 Series Stepper Driver with input pulse and direction signal	

G instructions

Instructions	Functions	Instructions	Functions
G00	Rapid traverse movement	G70	Finishing cycle
G01	Linear interpolation	G71	Axial roughing cycle
G02	Circular interpolation （CW）	G72	Radial roughing cycle
G03	Circular interpolation （CCW）	G73	Closed cutting cycle
G04	Dwell time preset	G74	Axial grooving cycle
G28	Machine reference point automatic return	G75	Radial grooving cycle
G32	Thread cutting with constant lead	G76	Multiple thread cutting cycle
G33	Tapping cycle in Z direction	G90	Axial cutting cycle
G34	Thread cutting with variable lead	G92	Thread cutting cycle
G40	Canceling tool nose radius	G94	Radial cutting cycle

	compensation		
G41	Tool nose radius compensation left of contour	G96	Constant surface speed ON
G42	Tool nose radius compensation right of contour	G97	Constant surface speed OFF
G50	Setting workpiece coordinate system	G98	Feed per minute
G65	Macro instruction	G99	Feed per rev

PLC instruction list

Elementary instructions	Functions	Function instructions	Functions
LD	Read normally-open contact	TMRB	Timer
LDI	Read normally-closed contact	CODB	Binary conversion
OUT	Output coil	ROTB	Binary rotation control
AND	Normally-open contact in series	MOVN	Data copy
ANI	Normally-closed contact in series	DECB	Binary decoding
OR	Parallel normally-open contact	JMPB	Program skipping(jumping)
ORI	Parallel normally-closed contact	SP	Subprogram
ORB	Parallel series circuit block	SPE	End of subprogram
ANB	Parallel circuit block in series	ADDB	Binary data adding
		SUBB	Binary data subtracting
Function instructions	**Functions**	ALT	Alternative output
END1	End of grade one program	DIFU	Up set
END2	End of grade two program	DIFD	Down set
SET	Set	MOVE	And
RST	Reset	PARI	Parity check
CMP	Comparative set	LBL	Program skipping label
CTRC	Counter	CALL	Subprogram calling

Type significations

GSK 980TD—□

Assembly form：none：standard panel（420×260mm）
B： boxed assembly

980TD Turning Machine CNC

Symbol of GSK CNC Equipment Co., Ltd.

Type	Specification
GSK980TD	420×260mm aluminium alloy solid operator panel
GSK980TD-B	GSK980TD matching with AP01（445mm×345mm×182mm）

Standard functions

All optional functions without being remarked in the provided technical specifications are as follows: Max. rapid traverse speed 16m/min, max. feedrate 8m/min, pitch error compensation, tool nose radius compensation, spindle analog voltage control(converter spindle),communication, 16 input points, 16 output points, standard PLC ladder, I/O interfaces being compatible with those of GSK980TA CNC system, 4-gear spindle automatic shifting gear(only test 1st and 2nd gear), hydraulic chuck, hydraulic tailstock, 4～8 tool selections toolpost(unidirectional selecting tool), safeguard, low pressure alarm etc.

Note 1: Modify or redesign PLC ladder diagram when other functions including executing the bidirectional tool change or testing 4-gear spindle are incompatible with those of 980TA CNC System.

Note 2: Please remark the detailed control requirements in order lists when special PLC ladder diagram (I/O interfaces are incompatible with those of GSK 980TA CNC System) is required.

Optional functions

1. Max. rapid traverse speed 30m/min and max. feedrate 15m/min;
2. I/O expansion: 16 input points (XS41 interface) and 16 output points (XS42 interface);

Standard accessories

Power switch: GSK-PB (assembled)

Connector: CNC interfaces are connected by one set of plug(DB9 female×3, DB15 male×3, DB25 female×1, DB25 male×1)

Note: Corresponding plugs along with cables are supplied when they along with other components including driver are delivered.

Accessory cables: 12m 10-core shield cable (3m for each X axis, Z axis, input interface XS40, output interface XS39);
9m 8-core shield cable with (3m for spindle encoder, input interface XS40, output interface XS39);

3m 4-core shield cable (converter interface);

Note: The above-mentioned cables as wires are supplied. Signal cables with welded plugs are supplied when a whole set of driver and toolpost controller is delivered. The requirements for cable length and welding should be remarked in the order list.

Anti-interference components: 1N4007×8、0.1μF/630V×6

Technical documents: *GSK980 Turning Machine CNC System User Manual*(without *PLC User Manual*)

Optional accessories

Communication components: one piece of 5m communication cable and one installation diskette of communication software TDComm2;

Power filter：FN2060-6-06

Handwheel: Dongxin RE45T1SO5B1(option: AP01) or Changchun LGF-001-100(option: AP02);

Additional panel：AP01（aluminum alloy 420×71 ㎜） can be assembled under of GSK980TD operator panel;

AP02（aluminum alloy 100×260 ㎜） can be assembled at the side of GSK980TD operator panel;

Emergent stop button: LAY3-02ZS/1(it has been installed when GSK980TD-B is delivered);

No self-locking button: KH-516-B11(blue or red);

Self-locking button: KH-516-B21(blue or red);

GSK980TD PLC User Manual

Ladder diagram programming software: one GSKCC installation diskette

Note 1: Communication functions are standard ones but communication components are optional accessories;

Note 2: Optional accessories as product ones (without being installed and connected) are supplied and it should be remarked in the order list when they are required to install and connect.

1.2 CNC SYSTEMS of MACHINE TOOLS and CNC MACHINE TOOLS

CNC machine tool is an electro-mechanical integrated product, composed of Numerical Control Systems of Machine Tools, machines, electric control components, hydraulic components, pneumatic components, lubricant, coolant and other subsystems (components), and CNC systems of machine tools are control cores of CNC machine tools. CNC systems of machine tools are made up of computerized numerical control(CNC), servo (stepper) motor drive devices, servo (or stepper) motor and etc.

Operational principles of CNC machine tools: according to requirements of machining technology, edit user programs and input them to CNC, then CNC outputs motion control instructions to the servo (stepper) motor drive devices, and last the servo (or stepper) motor completes the cutting feed of machine tool by mechanical driving device; logic control instructions in user programs to control spindle start/stop, tool selections, coolant ON/OFF, lubricant ON/OFF are output to electric control systems of machine tools from CNC, and then the electric control systems control output components including buttons, switches, indicators, relays, contactors and so on. Presently, the electric control systems are employed with Programmable Logic Controller (PLC) with characteristics of compact, convenience and high reliance. Thereof, the motion control systems and logic control systems are the main of CNC machine tools

GSK980TD Turning Machine CNC system has simultaneously motion control and logic control function to control two axes of CNC machine tool to move, and has embedded PLC function. Edit PLC programs (ladder diagram) according to requirements of input and output control of machine tool and then download them to GSK980TD Turning Machine CNC system, which realizes electric control requirements of required machine tool, is convenient to electric design of machine tool and reduces lost of CNC machine tool.

Software used for controlling GSK980TD Turning Machine CNC system is divided into system software (NC for short) and PLC software (PLC for short). NC system is used for controlling display, communication, edit, decoding, interpolation and acceleration/deceleration, and PLC system for controlling explanations, executions, inputs and outputs of ladder diagrams.

Standard PLC programs are loaded(except for the special order) when GSK980TD Turning Machine CNC System is delivered, concerned PLC control functions in following functions and operations are described according to control logics of standard PLC programs, marking with "*Standard PLC functions*" in *GSK980TD Turning Machine CNC System User Manual*. Refer to *Operation Manual* from machine manufacturer about functions and operations of PLC control because the machine manufacturer may modify or edit PLC programs again.

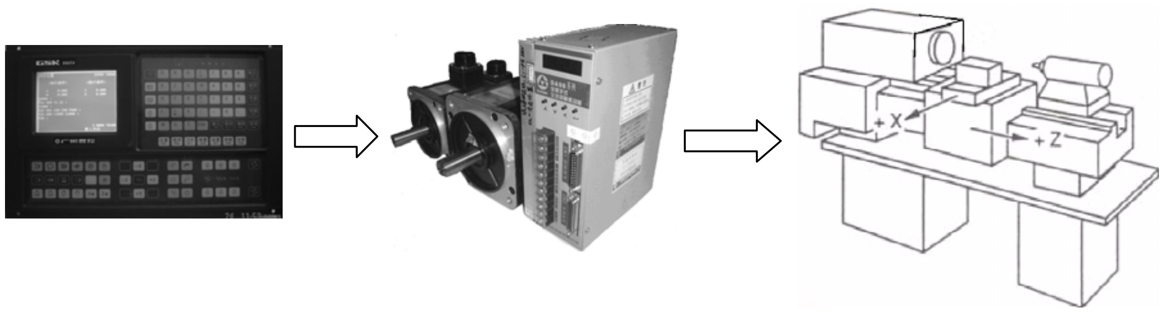
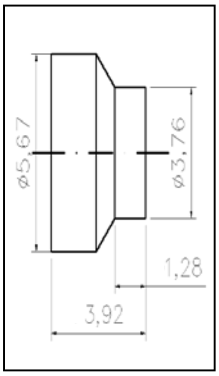


Fig. 1-1

Programming is a course of workpiece contours, machining technologies, technology parameters and tool parameters being edit into part programs according to special CNC programming instructions. CNC machining is a course of CNC controlling a machine tool to complete machining of workpiece according requirements of part programs. Technology flow of CNC machining is as follows Fig. 1-2.



Confirm machining technologies after analyzing workpiece drawings

Edit part programs and record into CNC

```
O0001;
G00 X3.76 Z0;
G01 Z-1.28 F50;
…
M30;
%
```

Test part programs and execute trial running

Execute toolsetting and set tool offsets and coordinates

Run part programs and machine workpieces

Check part dimension and modify part programs and compensations

The machining ends and the workpiece is formed

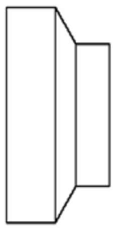


Fig. 1-2

1.3 PROGRAMMING FUNDAMENTALS

1.3.1 Coordinates Definition

Sketch map of CNC turning machine is as follows:

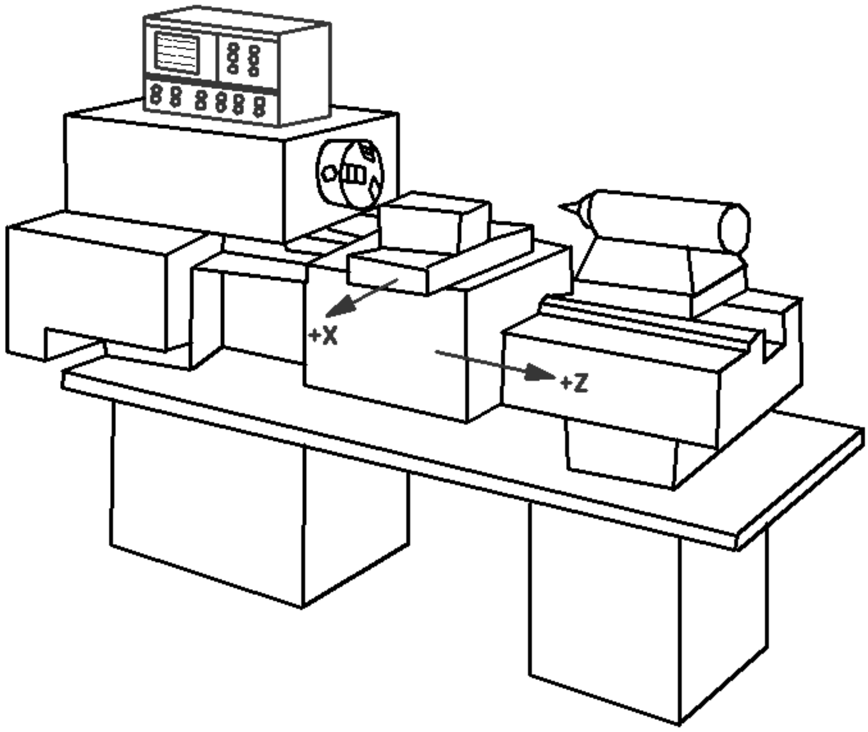


Fig. 1-3

The system is employed with a rectangular coordinate system composed of X, Z axis. X axis is perpendicular with axes of spindle and Z axis is parallel with axes of spindle; negative directions of them approach to the workpiece and positive ones are away from it.

There are a front toolpost and a rear toolpost of NC turning machine according to their relative position between the toolpost and the spindle, Fig. 1-4 is a coordinate system of the front toolpost and Fig. 1-5 is a rear toolppost one. It shows exactly the opposite direction in X direction but the same direction in Z direction from figures. In the manual, it will introduce programming application employed with the front toolpost coordinate system in following figures and examples.

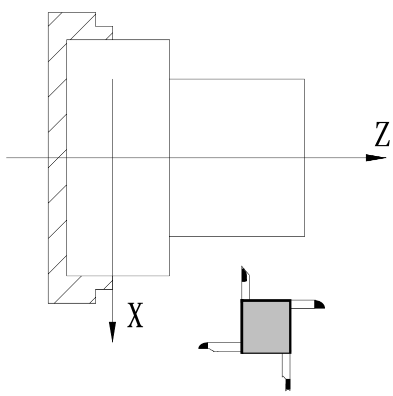


Fig.1-4 Front toolpost coordinate system

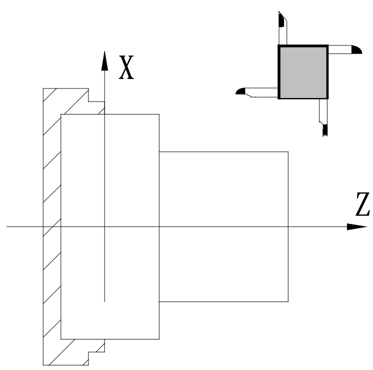


Fig. 1-5 Rear toolpost coordinate system

1.3.2 Machine Coordinate System and Machine Reference Point

Machine tool coordinate system is a benchmark one used for CNC counting coordinates and a fixed one on the machine tool. **Machine tool origin** is named **machine reference point** or **machine zero**. The position of machine reference point is specified by a reference point return switch on the machine tool. Usually, the reference point return switch is installed on max. stroke in X, Z positive direction. The system considers the current coordinates of machine tool as zeroes and sets the machine tool coordinate system according to the current position as the coordinate origin after having executed the machine reference point return.

Note: Do not execute the machine reference point return without the reference point switch installed on the machine tool.

1.3.3 Workpiece Coordinate System and Program Reference Point

Workpiece coordinate system is set to a rectangular coordinate system according to part drawings named floating coordinate system. After the workpiece is clamped on the machine tool, G50 is executed to set an absolute coordinates of tool's current position according to the relative position of tool and workpiece, and so the workpiece system has been created. The current position of tool is named program reference point and the tool returns to the position after executing the program reference point return. Usually, Z axis is consistent with the axes of spindle and X axis is placed on the heading or the ending of workpiece. The workpiece will be valid until it is replaced by a new one.

The current position of workpiece coordinate system set by G50 is named the program reference point and the system returns to it after executing the program reference point return.

Note: Do not execute the machine reference point return without using G50 to set the workpiece coordinate system after power on.

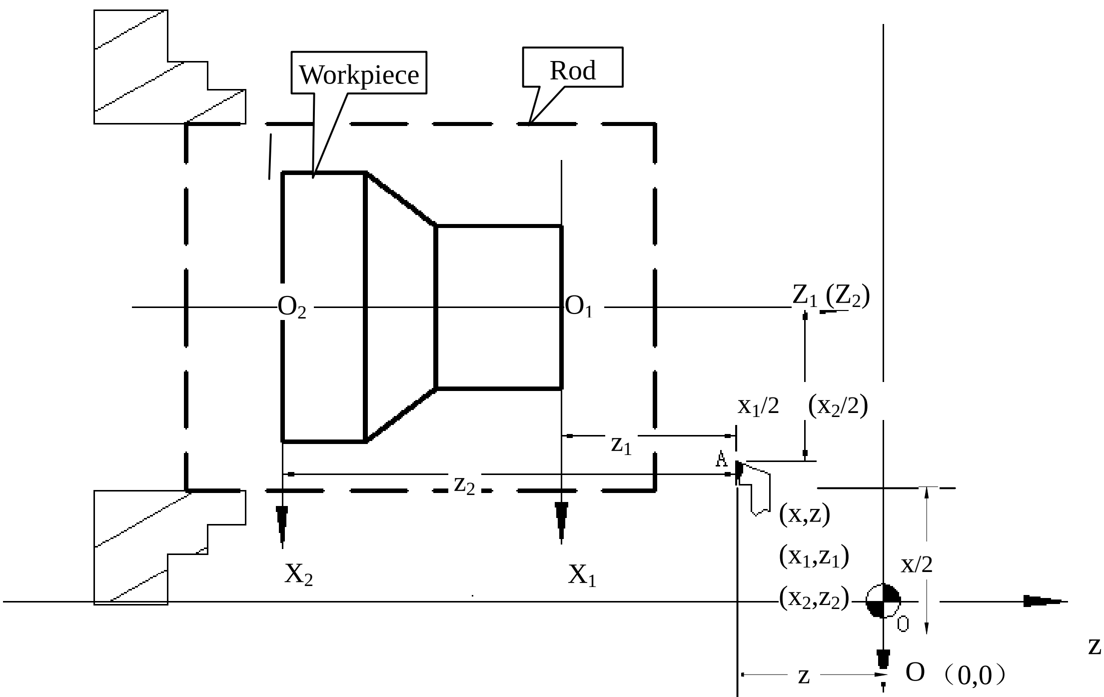


Fig.1-6

In the above figure, XOZ is the coordinate system of machine tool, $X_1O_1Z_1$ is the workpiece coordinate system of X axis located at the heading of workpiece, $X_2O_2Z_2$ is the one of X axis located at the ending of workpiece, O point is the machine reference point, A point is the tool nose and coordinates of A point in the above-mentioned coordinate systems is as follows:

A point in the machine tool coordinate system: (x,z);
A point in $X_1O_1Z_1$ coordinate system: (x_1,z_1);
A point in $X_2O_2Z_2$ coordinate system: (x_2,z_2);

1.3.4 Interpolation Function

Interpolation is defined as a planar or three dimensional contour formed by path of 2 or multiple axes moving at the same time, also called **Contour control**. The controlled moving axis is called link axis when the interpolation is executed. The moving distance, direction and speed of it are controlled synchronously in the course of running to form the required complex motion path. Fixed point control is defined that the motion path in the course of running are not controlled but end point of one axis or multiple axes moving.

X and Z in the system are link axes and 2 axes link CNC system. The system possesses linear, circular and thread interpolation function.

Linear interpolation: Complex motion path in X, Z direction is a straight line from starting point to end point.

Circular interpolation: Complex motion path in X, Z direction is arc radius defined by R or the circle center (I, K)
from starting point to end point.

Thread interpolation: Moving distance in X or Z direction or X and Z direction is defined by rotation angle of spindle to form spiral cutting path on the workpiece surface to realize the thread cutting. For thread interpolation, the feed axis rotates along with the spindle, the long axis moves one pitch when the spindle rotates one rev, and the short axis and the long axis directly interpolate.

Example:

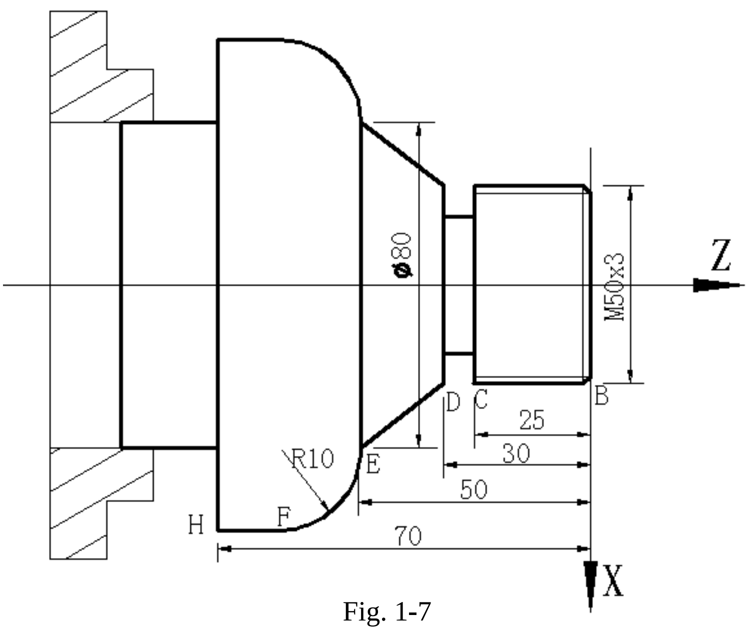


Fig. 1-7

...

G32 W-27 F3；　　　（B→C；thread interpolation）
G1 X50 Z-30 F100；
G1 X80 Z-50；　　　（D→E；linear interpolation）
G3 X100 W-10 R10；　　（E→F；circular interpolation）
…
M30；

1.3.5 Absolute Programming and Incremental Programming

Specify coordinate values of path's end point or target position in programming and there are 3 kinds of programming method according to coordinate values in programming: absolute programming, incremental programming and compound programming

Absolute coordinate value to program（present with X, Z） in X, Z direction is absolute programming;
Incremental movement to program （present with U, W）in X, Z direction is incremental programming;
In the system, X,Z axis is separately employed with absolute programming and incremental program, which is called compound programming

Example：A→B linear interpolation

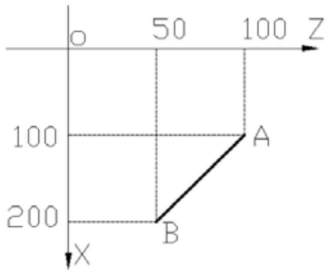


Fig. 1-8

Absolute programming：G01 X200. Z50.；
Incremental programming：G01 U100. W-50.；
Compound programming：G01 X200. W-50.；or G01 U100. Z50.；

Note: When there are instruction address X, U or Z, W simultaneously, X,Z are valid.

Example：G50 X10. Z20.;
G01 X20. W30. U20. Z30.;【End point of the block（**X20，Z30**）】

1.3.6 Diameter and Radius Programming

Diameter programming: when NO.001 Bit2 is 0, input instruction value in diameter in X direction and coordinate in X direction is in diameter at the moment;

Radius programming: when NO.001 Bit2 is 1, input instruction value in diameter in X direction and coordinate in X direction is diameter at the moment

Table 1-1: Address, data related to diameter or radius programming

	Address, data	Explanation	Diameter programming	Radius programming
Address, data related to diameter or radius programming	X	Coordinate in X direction	In diameter	In radius
		G50 setting X axis		
	U	Increment in X direction	In diameter	In radius
		Allowance of finishing in X direction in G71、G72、G73	In diameter	In radius
	R	Moving distance of tool retraction after cutting in G75	In diameter	In radius
		Moving distance of tool retraction when cutting to the end point in G74	In diameter	In radius

Except for addresses and data in Table 1-1, others (arc radius, taper in G90) are unrelated to diameter or radius programming, and their input values in X direction are defined by the radius.

It is employed with the diameter programming except for the special indication in the following explanation.

1.4　STRUCTURE of an NC PROGRAM

User needs to compile part programs (called program) according to instruction formats of CNC system. CNC system executes programs to control the machine tool movement, the spindle starting/stopping, the coolant and the lubricant ON/OFF to complete the machine of workpiece.

Program example:

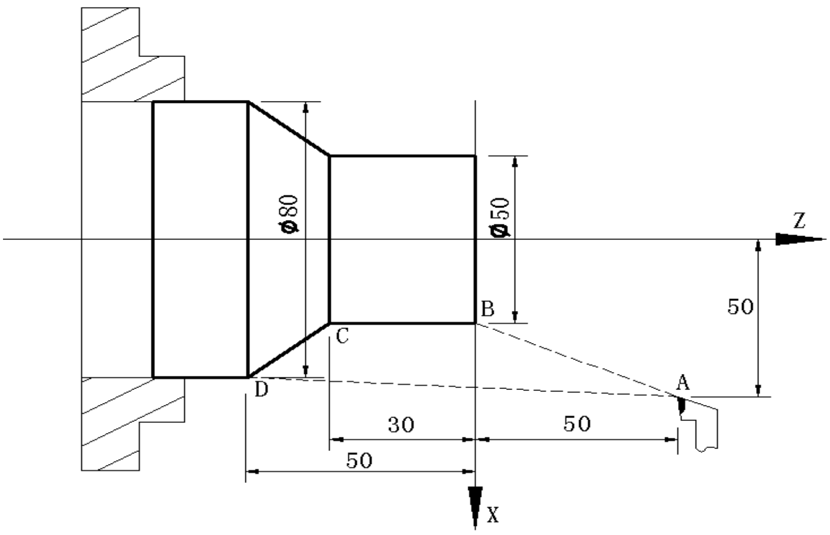


Fig. 1-9

O0001	；	（Program name）
N0005	G0 X100 Z100；	（Rapid positioning to A point）
N0010	M12；	（Workpiece clamped）
N0015	T0101；	（Changing No.1 tool and execute its offset）
N0020	M3 S600；	（Starting the spindle with 600 rev/min）
N0025	M8	（Coolant ON）
N0030	G1 X50 Z0 F600；	(Approaching B point with 600mm/min）

N0040	W-30 F200；	（Cutting from B point to C point）
N0050	X80 W-20 F150；	（Cutting from C point to D point）
N0060	G0 X100 Z100；	（Rapid retracting to A point）
N0070	T0100；	（Canceling the tool offset）
N0080	M5 S0；	（Stopping the spindle）
N0090	M9；	（Coolant OFF）
N0100	M13；	（Workpiece unclamped）
N0110	M30；	（End of program, spindle stopping and coolant OFF）

The tool leaves the path of A→B→C→D→A after the above-mentioned programs are executed.

1.4.1 General Structure of Program

A program consists of a sequence of blocks, beginning with “OXXXX”（program name）and ending with “%”; a block begins with block number (omitted) and ends with “;” or “*”. See the general structure of program as follows:

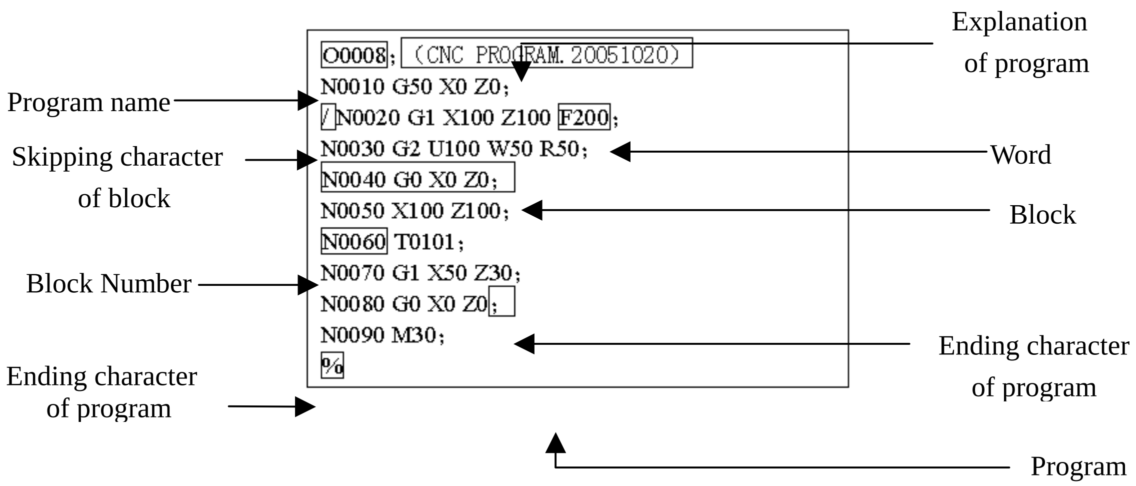
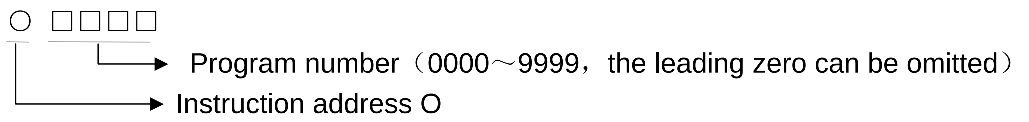


Fig. 1-10 General structure of program

Program name

There are most 384 programs stored in the system. To identify it, each program has only one program name(there is no the same program name)beginning with instruction address O and the following 4-bit digits.



Word

A word is the basic instruction unit to command CNC system to complete the control function, composed of an English letter (called instruction address) and the following number (operation instruction with/without sign). The instruction address describes the meaning of its following operation instruction and there may be different meaning in the same instruction address when the different words are combined together. See Table 1-2 words in the system.

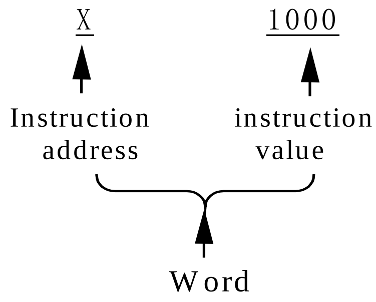


Table 1-2 Word list

Instruction address	Range of instruction value	Function
O	0～9999	Program name
N	0～9999	Block number
G	00～99	Preparatory function
X	-9999.999～9999.999(mm)	Coordinate in X direction
	0～9999.999(s)	Dwell time
Z	-9999.999～9999.999（mm）	Coordinate in Z direction
U	-9999.999～9999.999（mm）	Increment in X direction
	0～9999.999(s)	Dwell time
	-99.999～99.999（mm）	Finishing allowance in X direction in G71, G72, G73
	0.001～99.999（mm）	Cutting depth in G71
	-9999.999～9999.999（mm）	Moving distance of tool retraction in X direction in G73
W	-9999.999～9999.999（mm）	Increment in Z direction
	0.001～9999.999（mm）	Cutting depth in G72
	-99.999～99.999（mm）	Finishing allowance in Z direction in G71,G72, G73
	-9999.999～9999.999（mm）	Moving distance of tool retraction in Z direction in G73
R	-9999.999～9999.999（mm）	Arc radius
	0.001～9999.999（mm）	Moving distance of cycle tool retraction in G71,G72
	1～9999 (times)	Cycle times of roughing in G73
	0.001～9999.999（mm）	Moving distance of tool retraction after Cutting in G74, G75
	0.001～9999.999（mm）	Moving distance of tool retraction after cutting to the end point in G74, G75
	0.001～9999.999（mm）	Finishing allowance in G76
	-9999.999～9999.999（mm）	Taper in G90, G92, G94, G96
I	-9999.999～9999.999（mm）	Vector of arc center relative to starting point in X direction
	0.06～25400（tooth/inch）	Inch thread tooth
K	-9999.999～9999.999（mm）	Vector of arc center relative to starting point in Z direction

Instruction address	Range of instruction value	Function
F	0～8000（mm/min）	Feedrate per minute
	0.0001～500(mm/r)	Feedrate per rev
	0.001～500（mm）	Metric thread lead
S	0～9999（rev/min）	Specified spindle speed
	00～04	Multi-gear spindle output
T	01～32	Tool function
M	00～99	Auxiliary function output, program executed flow, subprogram call
	9000～9999	Subprogram call
P	0～9999999（0.001s）	Dwell time
	0～9999	Called subprogram number
	0～999	Calling times of subprogram
	0～9999999（0.001mm）	Circular moving distance in X direction in G74, G75
		Thread cutting parameter in G76
	0～9999	Initial block number of finishing in the compound cycle instruction
Q	0～9999	Terminative block number of finishing in the compound cycle instruction
	0～9999999（0.001mm）	Circular moving distance in Z direction in G74, G75
	1～9999999（0.001mm）	The first cutting depth in G76
	1～9999999（0.001mm）	Min. cutting depth in G76
H	01～99	Operator in G65

Block

A block which is basic unit of CNC program consists of a sequence of words, ending with “;” or “*” . There is the character “;” or “*” between blocks. “;” is employed to separate blocks in the manual as follows:

/ N0030 G0 X20 Z30 ；

Ending character of block

Block number

Skipping character of block

One block may be with a number of words or only with “；” (ending character) instead of words. There must be one or more blank space between words.

There is only one for other addresses except for N, G, S, T, H, L in one block, otherwise the system alarms. The last word in the same address is valid when there are more N, G, S, T, H, L in the same block. The last G instruction is valid when there are more G instructions which are in the same group in one block.

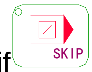
Block number

A block number consists of an address N and its following 4-bit digit as N0000～N9999,and the leading zero can be omitted. The block number must be at the beginning of block, otherwise the block is invalid.

The block number can be omitted, but there must be the block number when the program calls/skips the target block. The increment of block number is at will and it better to increase or decrease the sequence of block number in order to conveniently search and analyze programs.

When *"Block number"* is set to "ON", block numbers will be automatically created incrementally and their increment is defined by №42.

Block skipping character

Insert "/" in the front of block and startup  when some block cannot be executed (cannot be deleted), and the system skips the block and executes the next one. The block with "/" in the front of it will be executed if  is not started.

Ending character of program

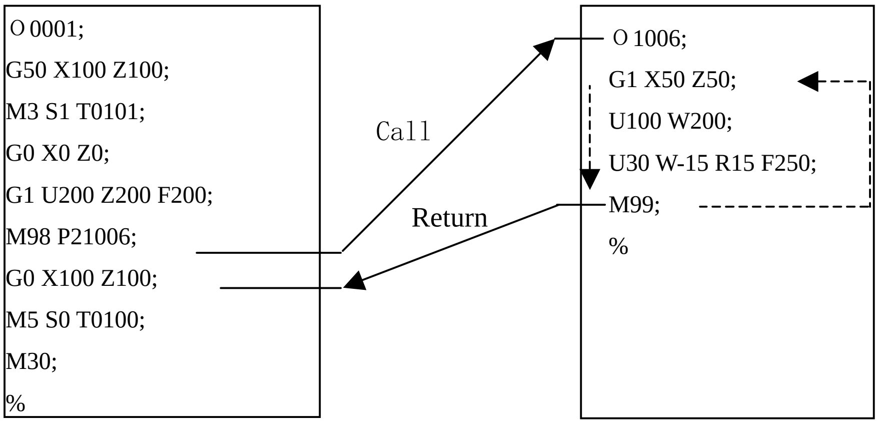
" %" is an ending character of program. "%" is a mark of communication ended when the program is transmitted. The system will automatically insert "%" at the end of program.

Program annotation

A program annotation has less than 20 characters (10 Chinese characters) for each program, lies in a bracket following its program name and is expressed only in English and digitals in CNC system; it can be edit in Chinese in PC and displayed in Chinese in CNC system after being downloaded.

1.4.2 Main Program and Subprogram

To simply the programming, when the same or similar machining path and control procedure is used many times, its program instructions are edited to a sole program to call. The main program is defined to call others and the subprogram is to be called. They both take up the program capacity and storage space of system. The subprogram has own name, and can be called at will by the main program and also can run separately. The system returns to the main program to continue when the subprogram ends as follows:



Main program　　Subprogram

Fig.1-11

1.5 PROGRAM RUN

1.5.1 Sequence of Program Run

Open the current program in Auto mode. The system only open one program, and so only one run any time. When opening the first block, the cursor located in the heading of the first block and can be

moved in Edit mode. The program stops in Auto mode and starts run after the cycle start signal (RUN is pressed or external cycle start signal）startups the program to run from a block pointed by current cursor, usually blocks are executed one by one according to programming sequence, the program stops run after executing M02 or M30. The cursor moves along with program and is located at the heading of current block. Sequence and state of program run are changed in the followings:

- The program stops run after pressing RESET or emergent stop button;
- The program stops run when the system or PLC alarms;
- Single block stops run (the current block pauses after it runs completely) in Edit, MDI mode, and then a block pointed by the current cursor starts run after the system switches into Auto mode, RUN is pressed or external cycle start signal is switched on;
- The program stops run in Manual(Jog), Handwheel(MPG), Single Block, Program Reference Point Return, Machine Reference Point Return mode and it continuously runs from current position after the system is switched into Auto mode and RUN is pressed or the external cycle start signal is switched on;
- The program pauses after pressing PAUSE or the external cycle start signal is switched off, and it continuously runs from current position after pressing RUN or the external cycle start signal is switched on;
- When Single Block is ON, the program pauses after every block is executed completely，and then it continuously runs from the next block after RUN is pressed or the external cycle start signal is

switched on;

- Block with "/" in the front of it is not executed when the block skipping switch is ON;
- The system skips to the target block to run after executing G65;
- Please see Section Three G Instructions about execution sequence of G70~73;
- Call corresponding subprograms or macro program to run when executing M98 or M9000~M9999; the system returns to main program to call the next block when executing M99(if M99 specifies a target block number, the system returns to it to run) after the subprograms or macro programs run completely;
- The system return to the first block to run and the current program is executed repetitively when M99 is executed in a main program.

1.5.2 Execution Sequence of Word

There are many words(G, X, Z, F, R, M, S, T and so on) and most of M, S, T is transmitted to PLC by NC explanation and others is directly executed by NC. M98, M99, M9000～M9999, S word for specifying spindle speed (rev/min, m/min) is directly executed by NC.

NC firstly executes G and then M instructions(without transmitting M signal to PLC) when G instructions and M98, M99, M9000～M9999 are in the same block.

When G instructions and M, S, T executed by PLC are in the same block, PLC defines M, S, T and G to be executed simultaneously, or execute M, S ,T after G instructions. Please see User Manual of machine manufacturer for execution sequence of instructions.

Execution sequence of G, M, S, T in the same block defined by GSK980TD standard PLC program is as follows:

M3, M4, M8, M10, M12, M32, M41, M42 ,M43,M44,S□□, T□□□□ and G instructions are executed simultaneously;

M5, M9, M11, M13, M33 after G instructions are executed;

M00, M02, M30 after other instructions of current block are executed.

Chapter 2   M.S.F.T INSTRUCTION

2.1   M INSTRUCTION（AUXILIARY FUNCTION）

M instruction consists of instruction address M and its following 1～2 or 4 bit digits, used for controlling the flow of executed program or outputting M instructions to PLC.

M □□□□
Instruction value（00~99,   9000~9999，the leading zero can be omitted）
Instruction address

M98, M99, M9000～M9999 is executed by NC separately and NC does not output M instructions to PLC.

Nc defines M02, M03 end of programs and outputs M instructions to PLC which can control spindle, coolant and so on.

M98, M99, M9000～M9999 are defined to call programs, M02, M30 are defined to end of program which are not changed by PLC. Other M instructions output to PLC and their function are defined by PLC. Please refer to *User Manual* from machine manufacturer.

There is only one M instruction in one block, otherwise the system alarms.

Table 2-1 M instructions

Instructions	Functions
M02	End of program
M30	End of program
M98	Call subprograms
M99	Return from a subprogram; it is executed repeatedly when the program ends in M99(the current program is not called by other programs)
M9000～M9999	Call macro programs(their program numbers are more than 9000)

2.1.1   End of Program M02

Instruction format: M02 or M2

Instruction function: In Auto mode, after other instructions of current block are executed, the automatic run stops, and the cursor stops a block in M02 and does not return to the start of program. The cursor must return to the start of program when the program is executed again.

Except for the above-mentioned function executed by NC, function of M002 is also defined by PLC ladder diagram as follows: current output of CNC is reserved after executing M02.

2.1.2   End of Program Run M30

Instruction format: M30

Instruction function: In Auto mode, after other instructions of current block are executed in M30, the automatic run stops, the amount of workpiece is added 1, the tool nose radius compensation is cancelled and the cursor returns to the start of program (whether the cursor return to the start of program or not is defined by parameters).

If NO.005 Bit 4 is set to 0, the cursor does not return to the start of program, and the cursor returns immediately after the program is executed completely when NO.005 Bit 4 is set to 1.

Except for the above-mentioned function executed by NC, the function of M30 is also defined by PLC ladder diagram as follows: the system closes M03, M04 or M08 signal output and outputs M05 signal after executing M30.

2.1.3 Subprogram Call M98

Instruction format：

M98 P○○○ □□□□

Called subprogram number（0000～9999）. The leading zero of subprogram number can be omitted when the calling times is not input; the subprogram number must be with 4-bit digits when the calling times is input.

Calling times: 1-9999. The calling times cannot be input when it is 1.

Instruction function: In Auto mode, after other instructions are executed in M98, CNC calls subprograms specified by P, and subprograms are executed 9999 times at most. M98 is invalid in MDI mode.

2.1.4 Return from Subprogram M99

Instruction format： M99 P○○○○

Executed block after returning to the main program is 0000～9999，and its leading zero can be omitted.

Instruction function: After other instructions of current block in the subprogram are executed, the system returns to the main program and continues to execute next block specified by P, and calls a block following M98 of current subprogram when P is not input. The current program is executed repeatedly when M99 is defined to end of program (namely, the current program is executed without calling other programs). M98 is invalid in MDI mode.

Example: Execution path of calling subprogram (with P in M99) as Fig. 2-1.

Execution path of program without P in M99.

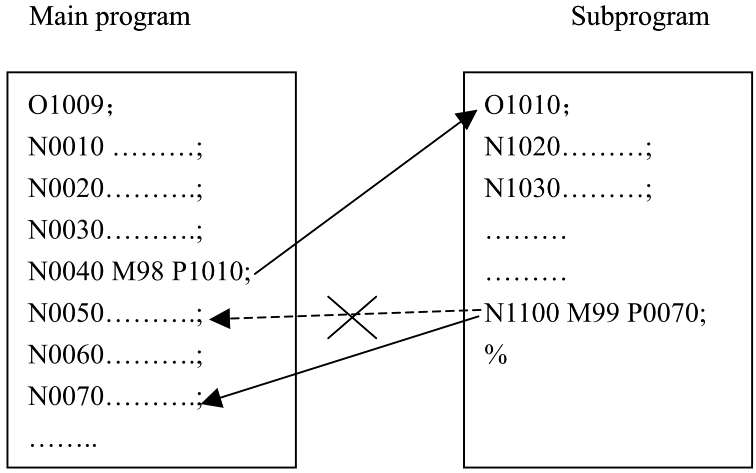


Fig.2-1

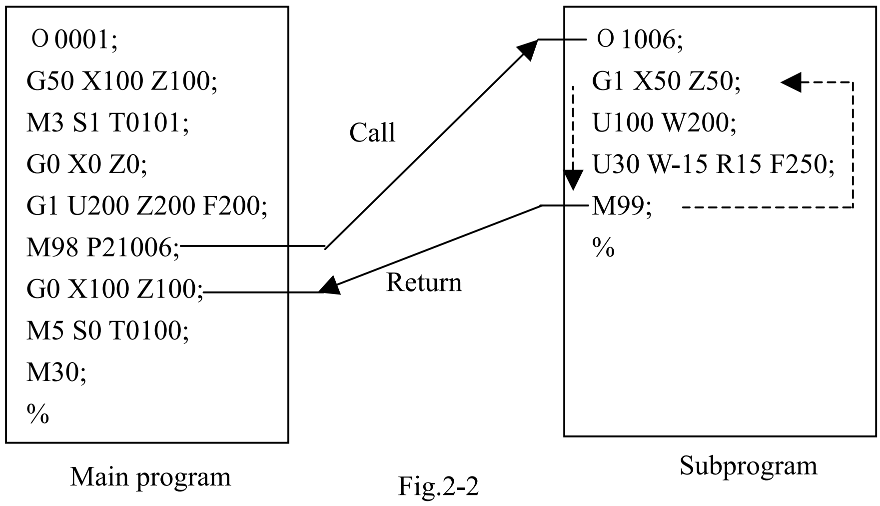


Fig.2-2

The system can call fourfold-embedded subprograms, namely can call other subprograms in another subprogram as Fig. 2-3.

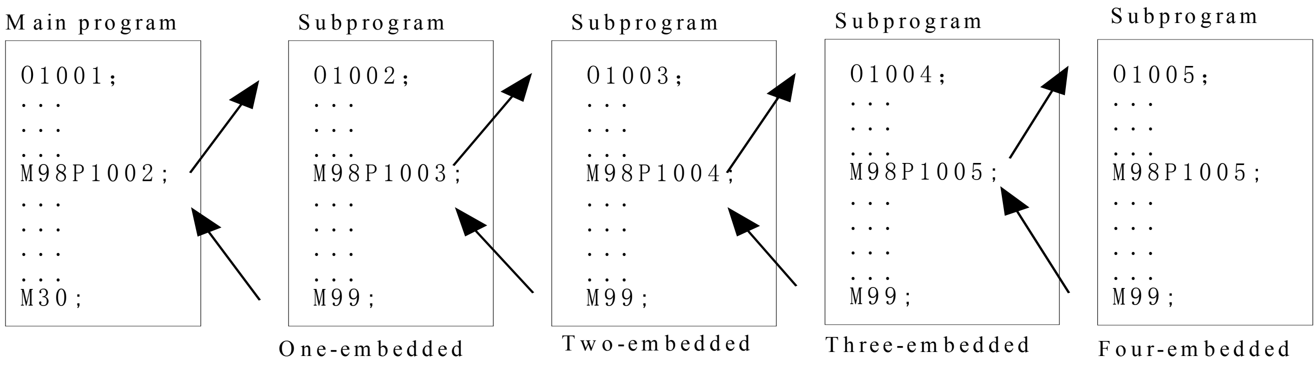


Fig. 2-3 Subprogram embedding

2.1.5 Macro Program Call

Instruction format: M□□□□ — 9000～9999

Instruction function: call macro programs corresponding to instruction values（O9000～O9999）.

Macro programs: O9000～O9999 programs

Macro programs: O9000～O9999 programs are for machine manufacturer, used for editing subprogram with special functions, called macro programs. The system must have 2 grades operation legal power(machine manufacturer)when editing O9000～O9999, and macro programs calling instructions are executed to call with 3～5 grades operation legal. M9000～M9999 are invalid in MDI mode.

2.1.6 M Instructions Defined by Standard PLC Ladder Diagram

Other M instructions are defined by PLC except for the above-mentioned ones（M02、M30、M98、M99、M9000～M9999）. The following M instructions are defined by standard PLC, and GSK980TD Turning Machine CNC system is used for controlling machine tool. Refer to instructions of machine manufacturer about functions, significations, control time sequence and logic of M instructions.

M instructions defined by standard PLC ladder diagram

Instruction	Function	Remark
M00	Program pause	
M03	Spindle clockwise	Functions interlocked and states reserved
M04	Spindle counterclockwise	
*M05	Spindle stop	
M08	Coolant ON	Functions interlocked and states reserved
*M09	Coolant OFF	
M10	Tailstock forward	Functions interlocked and states reserved
M11	Tailstock backward	
M12	Chuck clamping	Functions interlocked and states reserved
M13	Chuck unclamping	
M32	Lubricant ON	Functions interlocked and states reserved
*M33	Lubricant OFF	
*M41、M42、M43、M44	Spindle automatic gear shifting	Functions interlocked and states reserved

Note: Instructions with "" defined by standard PLC is valid when power on.*

2.1.7 Program Stop M00

Instruction format: M00 or M0

Instruction function: After executing M00, the program stops with "*Pause*", and continuously runs after pressing the cycle start key.

2.1.8 Spindle Control M03，M04，M05

Instruction format: M03 or M3

M04 or M4；

M05 or M5.

Instruction function: M03：Spindle rotation CW；

M04: Spindle rotation CCW;

M05: Spindle stop.

Note: Refer to time sequence of output defined by standard PLC ladder in Ⅳ ***Installation and Connection****.*

2.1.9 Coolant Control M08，M09

Instruction format: M08 or M8;

M09 or M9;

Instruction function: M08: Coolant ON;

M09: Coolant OFF.

Note: Refer to time sequence and logic of M08, M09 defined by standard PLC ladder in Ⅳ ***Installation and Connection***.

2.1.10 Tailstock Control M10，M11

Instruction format: M10；

M11；

Instruction function：M10：tailstock going forward；

M11：tailstock going backward.

Note: *Refer to time sequence and logic of M10, M11 defined by standard PLC ladder in* Ⅳ ***Installation and Connection.***

2.1.11 Chuck Control M12，M13

Instruction format: M12；

M13；

Instruction function: M12：chuck clamping；

M13：chuck unclamping.

Note: Refer to time sequence and logic of M10, M11 defined by standard PLC ladder in Ⅳ ***Installation and Connection.***

2.1.12 Lubrication Control M32，M33

Instruction format: M32；

M33；

Instruction function：M32：lubricant ON；

M33：lubricant OFF.

Note: Refer to time sequence and logic of M32, M33 defined by standard PLC ladder in Ⅳ ***Installation and Connection.***

2.1.13 Spindle Automatic Gear Shifting M41, M42, M43, M44

Instruction format：M4n；(n=1、2、3、4)

Instruction function: the spindle automatically gears to No. n gear when M4n is executed.

Note: Refer to time sequence and logic of M41, M42, M44 defined by standard PLC ladder in Ⅳ ***Installation and Connection.***

2.2 SPINDLE FUNCTION（S FUNCTION）

S instruction is used for controlling spindle speed and there are two methods to control that of GSK980TD: Spindle speed switching value control: S□□（2-bit digits instruction value）is executed by PLC, and PLC outputs switching value signal to machine tool to change spindle speed with grades

Spindle speed analog voltage control: S□□□□（4-bit digits instruction value）specifies actual speed of spindle and NC outputs 0~10V analog voltage signal to spindle servo or inverter to realize stepless spindle speed

2.2.1 Spindle Speed Switching Value CONTROL

Spindle speed is controlled by switching value when NO.001 BIT4 is set to 0. There is only one S instruction in a block, otherwise the system alarms.

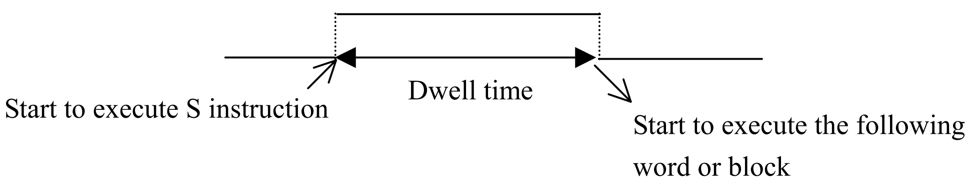
Their executing sequence is defined by PLC when S instruction and word for moving function are in the same block. Please refer to *User Manual* from machine manufacturer.

When spindle speed is control led by switching value, GSK980TD Turning CNC system is used for machine tool and the time sequence and logic of executing S instruction is according to *User Manual* from machine manufacturer. Refer to S instruction defined by standard PLC of GSK980TD as follows:

Instruction format： S□□

00～04（the leading zero can be omitted）：No.1～4 gear of spindle speed is controlled by switching value.

In spindle speed switching value control mode, after S signal transmits to PLC, the system dwells time defined by NO.081, then return FIN signal, and the dwell time is called runtime of S instruction.



S01, S02, S03, S04 output are reserved when resetting CNC.

S1～S4 output are invalid when CNC is switched on. The corresponding S signal output is valid and reserved, and others are cancelled at the same time when executing one of S01, S02, S03, S04. When executing S00, S1～S4 output are cancelled and only one of S1～S4 is valid at the same time.

2.2.2 Spindle Speed Analog Voltage Control

Spindle speed is controlled by analog voltage when NO.001 BIT4 is set to 1.

Instruction format：S OOOO

└── 0000～9999（the leading zero can be omitted.）：Spindle speed analog voltage control

Instruction function: the spindle speed is defined, and the system outputs 0～10V analog voltage to control spindle servo or inverter to realize the stepless timing. S instruction value is not reserved, and it is 0 after the system is switched on.

When the spindle speed analog voltage control is valid, there are 2 methods to input the spindle speed: the spindle fixed speed is defined by S instruction(rev/min), and is invariant without changing S instruction value, which is called constant speed control(G97 modal); other is the tangent speed of tool relative to the outer circle of workpiece defined by S instruction, which is called constant surface speed control (G96 modal), and the spindle speed is changed along with the absolute coordinates value of X absolute coordinates in programming path when cutting feed is executed in the constant surface speed. Please refer to ***Section 2.2.3.***

The system can execute 4 gears spindle speed. Count the analog voltage value corresponding to the specified speed according to setting value(corresponding to NO.037～NO.040) of max. spindle speed (analog voltage is 10V)of current gear, and then output to spindle servo or inverter to ensure that the spindle actual speed and the requirement are the same.

After the system is switched on, the analog output voltage is 0V. The analog output voltage is reserved (except that the system is in cutting feed in the surface speed control mode and the absolute value of X absolute coordinates is changed) after S instruction is executed. The analog output voltage is 0V after S0 is executed. The analog output voltage is reserved when the system resets and emergently stops.

Parameters relative to the analog voltage control of spindle speed:

System parameter NO.21: offset value of output voltage with max. spindle speed (the analog output voltage is 10V);

System parameter NO.36: offset value of output voltage with spindle speed 0 (the analog output voltage is 10V);

System parameter NO.037～NO.040: max. spindle speed (the analog output voltage is 10V) with spindle 1～4 gears.

2.2.3 Constant Surface Speed Control G96, Constant Rotational Speed Control G97

Instruction format：G96 S__；（S0000～S9999，the leading zero can be omitted.）

Instruction function：the constant surface speed control is valid, the cutting surface speed is defined (m/min) and the constant rotational speed control is cancelled. G96 is modal G instruction. If the current modal is G96, G96 cannot be input.

Instruction format：G97 S__；（S0000～S9999，the leading zero can be omitted.）

Instruction function：the constant surface speed control is cancelled, the constant rotational speed control is valid and the spindle speed is defined(rev/min). G96 is modal G instruction. If the current modal is G97, G97 cannot be input.

Instruction format：G50 S__；（S0000～S9999，the leading zero can be omitted.）

Instruction function: define max. spindle speed limit (rev/min) in the constant surface speed control and take the current position as the program reference point.

G96, G97 are the modal word in the same group but one of them is valid. G97 is the initial word and the system defaults G97 is valid when the system is switched on.

When the machine tool is turning it, the workpiece rotates based on the axes of spindle as the center line, the cutting point of tool cutting workpiece is a circle motion around the axes, and the instantaneous speed in the circle tangent direction is called **cutting surface**(for short **surface speed**). There are different surface speed for the different workpeice and tool with different material.

When the spindle speed controlled by the analog voltage is valid, the constant surface control is valid. The spindle speed is changed along with the absolute value of X absolute coordinates of programming path in the constant speed control. If the absolute value of X absolute coordinates adds, the spindle speed reduces, and vice verse, which make the cutting surface speed as S instruction value. The constant speed control to cut the workpiece makes sure all smooth finish on the surface of workpiece with diameter changing.

Surface speed=spindle speed× |X| × π÷1000 (m/min)

Spindle speed: rev/min

|X|：absolute value of X absolute coordinates value (diameter value), mm

π≈3.14

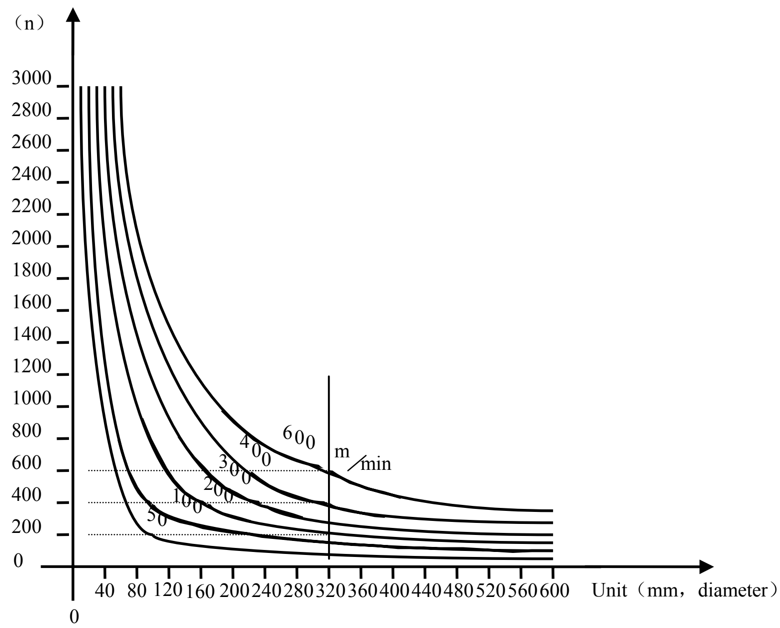


Fig. 2-4

In G96, the spindle speed is changed along with the absolute value of X absolute coordinates value of programming path in cutting feed (interpolation), but it is not changed in G00 because there is no actual cutting and is counted based on the surface speed of end point in the program block.

In G96, Z coordinates axis of workpiece system must consist with the axes of spindle (rotary axis of workpiece), otherwise, there is different between the actual surface speed and the defined one.

In G96, G50 S_ can limit max. spindle speed (rev/min). The spindle actual speed is the limit value of max. speed when the spindle speed counted by the surface speed and X coordinates value is more than the max. spindle speed set by G50 S_. After the system powers on, max. spindle speed limit value is not defined and its function is invalid. Max. spindle speed limit value defined by G50 S_ is reserved before it is defined again and its function is valid in G96. Max. spindle speed defined by G50 S_ is invalid in G97 but its limit value is reserved.

Note: In G96, the spindle speed is limited to 0 rev/min (the spindle does not rotate) if G50, S0 are executed; G50 S_ is executed to set max. spindle speed limit value of constant surface speed and also set the current position to the program reference point at the same time, and the tool returns to the current position after the program reference point return is executed.

When the constant surface speed is controlled by the system parameter NO.043, the spindle speed is lower limit, which is higher than one counted by the surface speed and X axis coordinates value

Example:

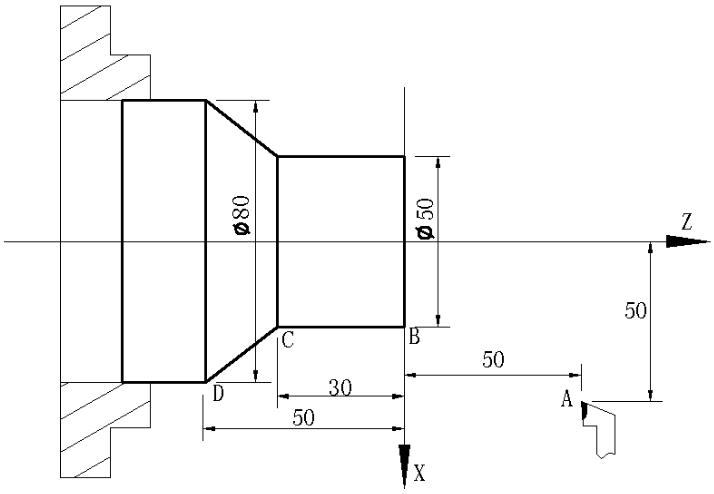


Fig. 2-5

O0001	;	(Program name)
N0010	M3 G96 S300;	(Spindle rotates clockwise, the constant surface speed control is valid and the surface speed is 300m/min)
N0020	G0 X100 Z100;	(Rapid traverse to A point with spindle speed 955 rev/min)
N0030	G0 X50 Z0;	(Rapid traverse to B point with spindle speed 1910 rev/min)
N0040	G1 W-30 F200;	(Cut from B to C with spindle speed 1910 rev/min)
N0050	X80 W-20 F150;	(Cut from C to D with spindle speed 1910 rev/min and surface speed 1194 rev/min)
N0060	G0 X100 Z100;	(Rapid retract to A point with spindle speed 955 rev/min)
N0110	M30;	(End of program, spindle stopping and coolant OFF)
N0120	%	

Note 1: S value commanded in G96 is also reserved in G97. Its value is resumed when the system is in G96 again.

Example:

G96 S50;　(Cutting surface speed 50m/min)

G97 S1000;　(Spindle speed 1000 rev/min)

G96 X3000;　(Cutting surface speed 50m/min)

Note 2: The constant surface speed control is valid when the machine tool is locked (X, Z axis do not move when their motion instruction are executed);

Note 3: To gain the precise thread machining, it should not be adopted with the constant surface speed control *but the constant rotational speed (G97) in the course of thread cutting;*

Note 4: From G96 to G97, if none of S instruction (rev/min) is commanded in the program block in G97, the last spindle speed in G96 is taken as S instruction in G97, namely, the spindle speed is not changed at this time;

Note 5: In G96, when the spindle speed counted by the cutting surface speed is more than max. speed of current spindle gear (system parameter NO.037～NO.040), at this time, the spindle speed is limited to max. one of current spindle gear.

2.2.4 Spindle Override

When the spindle speed analog voltage control is valid, the spindle actual speed can be tuned real time by the spindle override and is limited by max spindle speed of current gear after the spindle override is tuned, and it also limited by limited values of max. and min. spindle speed in constant surface speed control mode.

The system supplies 8 steps for spindle override （50%～120% increment of 10%）. The actual steps and tune of spindle override are defined by PLC ladder and introductions from machine manufacturer is referred when using it. Refer to the following functions of GSK980TD standard PLC ladder.

The spindle actual speed specified by GSK980TD standard PLC ladder can be tuned real time by the spindle override tune key at 8 steps in 50%～120% and it is not reserved when the spindle override is switched off. Refer to the operations of spindle override in *Ⅲ Operation.*

2.3 TOOL FUNCTION （T FUNCTION）

T functions of GSK980TD: automatic tool change and executing tool offset. Control logic of automatic tool change is executed by PLC and tool offset is executed by NC.

Instruction format：

T □□ ○○

Tool offset number（00-32，the leading zero cannot be omitted）

Target tool number（01-32，the leading zero cannot be omitted）

Instruction function: The automatic toolpost rotates to the target tool number and the tool offset of tool offset number commanded is executed. The tool offset number can be the same as the tool number, and also cannot be the same as it, namely, one tool can corresponds to many tool offset numbers. After executing tool offset and then T□□00, the system reversely offset the current tool offset and the system its operation mode from the executed tool length compensation into the non-compensation, which course is called the canceling tool offset, called canceling tool compensation. When the system is switched on, the tool offset number and the tool offset number displayed by T instruction is the state before the system is switched off.

Only one T instruction is in a block, otherwise the system alarms.

Toolsetting is executed to gain the position offset data before machining (called tool offset), and the system automatically executes the tool offset after executing T instruction when programs are running. Only edit programs for each tool according to part drawing instead of relative position of each tool in the machine coordinate system. If there is error caused by the wearing of tool, directly modify the tool offset according to the dimension offset.

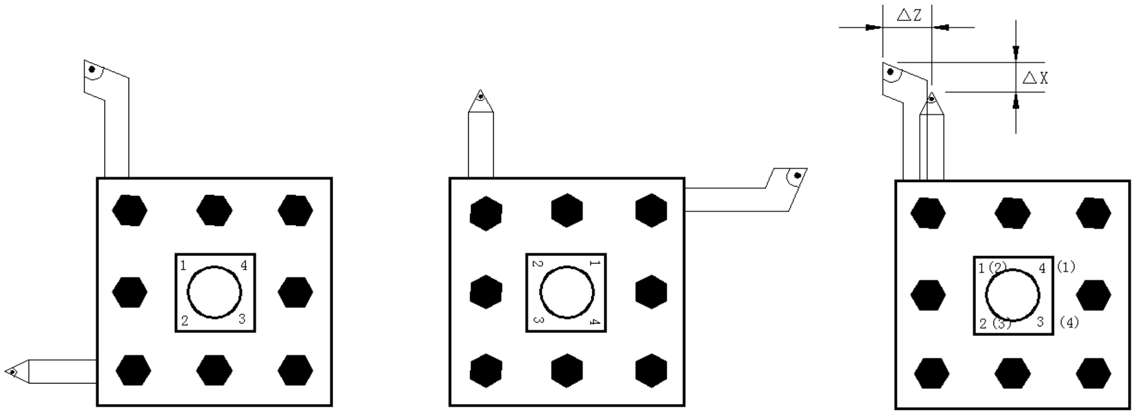


Fig. 2-4 Tool offset

The tool offset is used for the programming. The offset corresponding to the tool offset number in T instruction is added or subtracted on the end point of each block. Tool offset in X direction in diameter or radius is set by NO.004 Bit4. For tool offset in diameter or radius in X direction, The external diameter is changed along with diameter or radius when the tool length compensation is changed.

Example:

Course of creation, execution and cancellation of tool offset by movement is as Fig. 2-5.

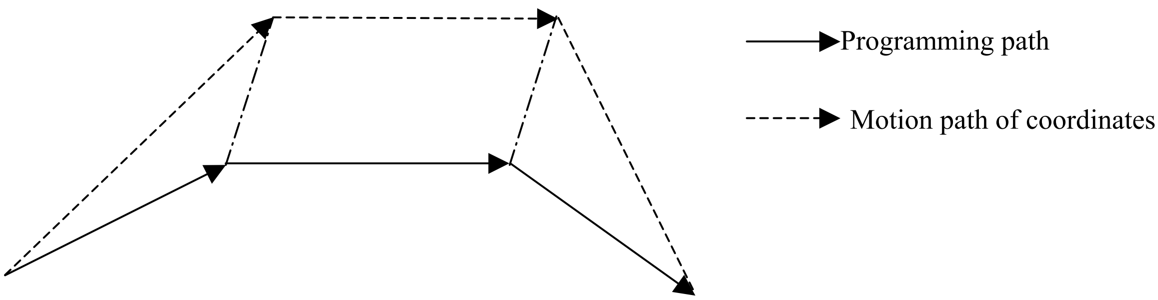


Fig. 2-5 Creation, execution and cancellation of tool length compensation

G01 X100 Z100 T0101; （Block 1， start to execute the tool offset）

G01 W150; （Block 2， tool offset）

G01 U150 W100 T0100; （Block 3, canceling tool offset）

There are two methods defined by NO.003 Bit4 to execute the tool length compensation:

Bit4=0: The tool length compensation is executed by the tool traversing;

Bit4=1: The tool length compensation is executed by modifying the coordinates;

Example:

Table 2-4

Tool offset number	X	Z
00	0.000	0.000
01	0.000	0.000
02	12.000	-23.000
03	24.560	13.452

State of T T0100	State of T T0202	State of T T0303
Coordinates displaying (Incremental coordinates) U：0.000 W：0.000 (Absolute coordinates) X：0.000 Z：0.000	Coordinates displaying (Incremental coordinates) U：12.000 W：-23.000 (Absolute coordinates) X：0.000 Z：0.000	Coordinates displaying (Incremental coordinates) U：24.560 W：13.452 (Absolute coordinates) X：0.000 Z：0.000
No. 01 tool without the tool compensation	Change No.2 tool and execute its offset by the tool traversing with 12mm in X positive direction and 23mm in Z negative direction	Change No.3 tool and execute its offset by the tool traversing with 12.56mm in X positive direction and 36.452mm in Z positive direction

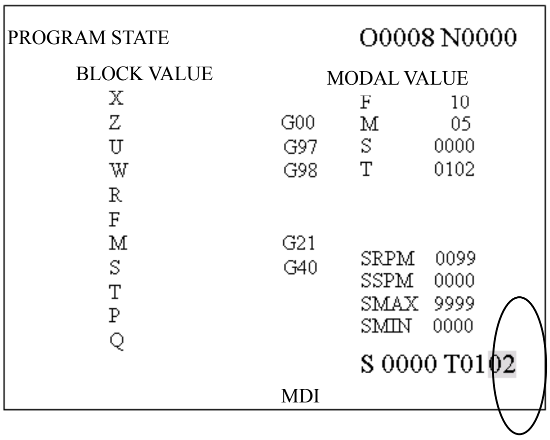
Fi.g 2-6 Tool traversing mode

State of T T0100	State of T T0202	State of T T0303
Coordinates displaying (Incremental coordinates) U：0.000 W：0.000 (Absolute coordinates) X：0.000 Z：0.000	Coordinates displaying (Incremental coordinates) U：0.000 W：0.000 (Absolute coordinates) X：12.000 Z：-23.000	Coordinates displaying (Incremental coordinates) U：0.000 W：0.000 (Absolute coordinates) X：24.560 Z：13.452
No. 01 tool without the tool compensation	Change No.2 tool and execute its offset by modifying the coordinates	Change No.3 tool and execute its offset by modifying the coordinates

Fig. 2-7 Modifying the coordinates mode

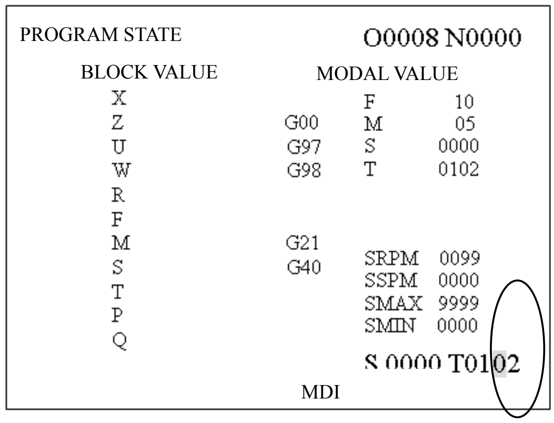
In Edit and Auto mode, a sole T word in executing tool offset (it is not with the motion instruction in the same block) is relative to NO.004 BIT3 setting (as Fig.2-6 and Fig.2-7). When NO.003 Bit4=1 and a sole T instruction is executed, the tool offset number is displayed in poor, which is cleared out(tool offset number is still displayed in poor when tool offset is not executed for one axis, the previous bit of tool offset number is for X axis tool compensation and the next one is for Z axis tool compensation) after executing tool offset.

Example: When NO.003 Bit is 1 and a sole T0102 is executed, the system displays after executing Z axis as follows:



Executing a sole T0102

Tool offset of two axes are not executed



Executing W0 after T0102

tool offset in X direction is not executed and it in Z direction is not

When T instruction and the motion instruction are in the same block and execute tool offset by modifying coordinates, the motion instruction and T instruction are executed at the same time, the system executes by adding the current tool offset to coordinates of motion instruction and whether the traverse speed is employed the cutting feedrate or the rapid traverse speed defined by the motion instruction.

When T instruction and the motion instruction are in the same block and execute tool offset by traversing tool, the motion instruction or T instruction is executed separately. Firstly tool change is executed and then the motion instruction is executed. The tool offset is executed at current rapid traverse speed.

The tool offset is cancelled after one of the following operations is executed:

1. Execute T□□00 instruction;
2. Execute G28 or manual machine reference point return (only the tool offset of coordinate axis which is executed machine reference point return is cancelled, and another one which is not executed machine reference point return is not cancelled);

When NO.084 is not 1 （2～32） and target tool number is not equal to current display tool number, the control sequence and logic of toolpost is defined by PLC ladder diagram after commanding T instruction, please see User Manual of machine tool manufacturer. GSK980TD standard PLC ladder diagram defines as follows: clockwise rotation for selecting tool, counterclockwise rotation for toolpost clamping, directly inputting tool selection signal for tool change. Please refer to ***Ⅲ Installation and Connection.***

When the system is employed with line-up toolpost, NO.084 should be set to 1 and different tool number is executed by different tool offset as T0101、T0102、T0103.

2.4 FEEDRATE FUNCTION (F FUNCTION)

2.4.1 Cutting Feed（G98/G99，F Instruction）

Instruction format: G98 F__；(F0001～F8000, the leading zero can be omitted, feedrate per minute is specified, mm/min)

Instruction function: cutting feedrate is specified as mm/min, G98 is the modal G instruction. G98 cannot be input if the current instruction is G98 modal.

Instruction format: G99 F__；(F0.0001～F500, the leading zero can be omitted).

Instruction function: cutting feedrate is specified as mm/min, G99 is a modal G instruction. G99 cannot be input if the current instruction is G98 modal. When G99F is executed, the arithmetic product of F instruction value(mm/rev) and current spindle speed(rev/min) is taken as feedrate to command actual cutting feedrate which is changed along with spindle speed. The cutting feedrate per rev specified by G99 F_ is contributed to the equable cutting line on the surface of workpiece. In G99, the machine tool must be employed with the spindle encoder to machine the workpiece on the machine tool.

G98, G99 are the modal G instruction in the same group and only one is valid. G98 is the initial state G instruction and the system defaults G98 is valid when the system switched on.

Reduction formula of feed between per rev and per min:

$$F_m = F_r \times S$$

F_m：feed per min （mm/min）；

F_r：feed per rev（mm/r）；

S：spindle speed （r/min）

After the system is switched on, the feedrate is 0 and F value is reserved after F is commanded. The feedrate is 0 after F0 is executed. F value is reserved when the system resets and emergently stops.

Note: In G99 modal, there is the uneven cutting feedrate when the spindle speed is lower than 1 rev/min; there is the follow error in the actual cutting feedrate when there is the swing in the spindle speed. To gain the high machining quality, it is recommended that the selected spindle speed should be not lower than min. speed of spindle servo or inverter.

Cutting feed: The system can control the motions in X, Z direction contributed that the motion path of tool and the defined path by instructions (line straight, arc)is consistent, and also instantaneous speed on the tangent of motion path and F word is consistent, which motion control is called cutting feed or interpolation. The cutting feedrate is specified by F, the system divides the cutting feedrate specified by F according to the programming path into vector in X, Z direction, also controls the instantaneous speed in X, Z direction to contributed that the combined speed of vector in X, Z direction is equal to F instruction value.

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

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

F is the combined speed of vector of instantaneous speed in X, Z direction;
d_x is the instantaneous(d_t) increment in X direction,
f_x is the instantaneous speed in X direction;
d_z is the instantaneous(d_t) increment in Z axis, fz is the instantaneous speed in Z direction

Example: In Fig. 2-8, the data in the brackets are the coordinates for each point (it is the diameter in X direction), the system parameter NO.022 is 3800, the system parameter NO.023 is 7600, the rapid override and feedrate override are 100%.

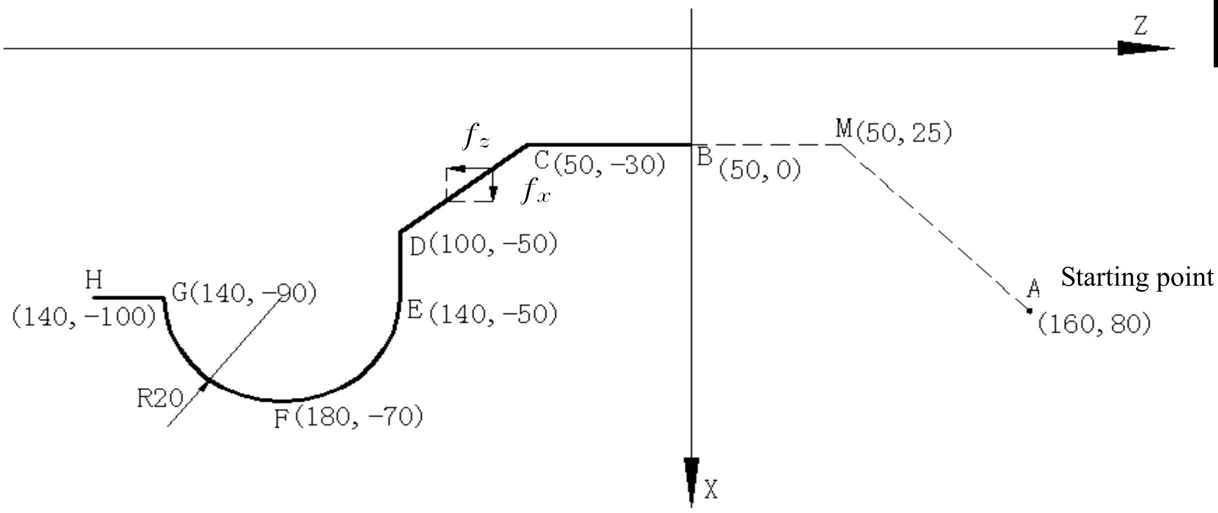


Fig. 2-8

Program as follows:

G50 X160 Z80; (Create a workpeice coordinates system)

G0 G98 X50 Z0; (Rapid traverse from A to B through M point. A→M: rapid traverse speed 7600mm/min in X direction, 7600mm/min in Z direction, M→B: rapid traverse speed 0mm/min in X direction, 7600mm/min in Z direction)

G1 W-30 F100; (B→C, rapid traverse speed 0mm/min in X direction, 100mm/min in Z direction)

X100 W-20; (C→D, rapid traverse speed 156mm/min in X direction, 62mm/min in Z direction)

X140; (D→E, rapid traverse speed 200mm/min in X direction, 0mm/min in Z direction)

G3 W-100 R20; (EFG circular interpolation, E point: instantaneous speed 200mm/min in X direction, 0mm/min in Z direction
F point: instantaneous speed 0mm/min in X direction, 100mm/min in Z direction)

W-10; (G→H, rapid traverse speed 0mm/min in X direction , 100mm/min in Z direction)

M30;

The system supplies 16 steps for spindle override (0%～150%, increment of 10%) .PLC ladder defines tune ways of spindle override and whether the actual feedrate override steps is reserved or not after the system is switched off, which is referred to *User Manual* from machine manufacturer when using the system. Refer to the following functions of GSK980TD standard PLC ladder.

The cutting feedrate can be tuned real time by the feedrate override key on the operator panel or the external override switch, and the actual cutting feedrate is tuned at 16 steps in 0～150% (increment of 10%) but it is invalid for thread cutting to tune the feedrate override.

Refer to *Ⅲ* ***Operation*** *about* cutting feedrate override**.**

Parameters：

System parameter NO.027: the upper limit value of cutting feedrate(they are the same in X, Z direction，diameter/min in X direction);

System parameter NO.029: exponential function for time constant of acceleration/deceleration when cutting feed and manual feed;

System parameter NO.030: initial (ultimate) speed of acceleration/deceleration in exponential function when cutting feed and manual feed.

2.4.2 Thread Cutting

Thread cutting: The system specifies a pitch to execute thread cutting along with spindle rotating. The tool moves a pitch when the spindle rotates one rev. Feedrate is relevant to the specified pitch, actual spindle speed. The system must be employed with spindle encoder which transmits the spindle actual speed to CNC in thread cutting. The thread cutting is not relevant to feedrate override and rapid override.

$$F = f \times S$$

F：Thread cutting feedrate（mm/min）；

f：Specified pitch（mm）；

S：Spindle actual speed（r/min）.

Parameters:

Data parameter NO.026: Acceleration/deceleration time constant of short axis in thread run-out

Data parameter NO.028: Feedrate lower limit in thread cutting;

Data parameter NO.029: Exponential acceleration/deceleration time constant in cutting feed and manual feed;

Data parameter NO.070: Set spindle encoder lines: 100～5000;

Data parameter NO.106: Max. absolute value of spindle speed fluctuation in thread cutting

Data parameter NO.107: Thread run-out speed in thread cutting

Data parameter NO.111: Set encoder teeth

Data parameter NO.110: Set spindle teeth

Data parameter NO.175 Bit4(THDACC): Set exponential or linear acceleration/deceleration when starting to thread cutting.

2.4.3 Manual Feed

Manual feed: the tool traverses in X or Z direction at the current manual feedrate in Manual but does not traverse in X, Z direction at the same time.

The system supplies 16 steps for manual feedrate（0%～150%, increment of 10%）. Actual feedrate override and its tune ways are defined by PLC ladder, which is referred to instructions from machine manufacturer when using the system. Refer to the following functions of GSK980TD standard PLC ladder.

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 is in diameter per minute in X direction; the feedrate override defined by GSK980TD PLC ladder is not reserved when the system is switch off.

Parameters：

System parameter NO.029: Exponential acceleration/ deceleration time constant in manual feed;

System parameter NO.041: the initial (terminate) speed (diameter per minute in X direction) of acceleration/ deceleration in manual feed.

2.4.4 Handwheel/Step Feed

Handwheel feed: the tool traverses in X or Z positive/negative direction at the current increment in "*Handwheel*" mode but does not traverse in X, Z direction at the same time.

Step feed: the tool traverses in X or Z positive/negative direction at the current increment in "*Step*" mode but does not traverse in X, Z direction at the same time.

One of "*Handwheel*" mode and "*Step*" mode is valid, which is defined by NO.001 Bit3.

The system supplies 4 steps(0.001mm, 0.01mm, 0.1mm, 1mm) for Handwheel mode and Step increment. PLC ladder defines actual handwheel/step increment steps, increment selection and current valid axis selection, which are referred to instructions from machine manufacturer when using the system.

Parameters:

System parameter NO.029: Exponential acceleration/ deceleration time constant in cutting feed and manual feed;

System parameter NO.041:the initial (terminate) speed (diameter per minute in X direction) of acceleration/ deceleration in manual feed.

2.4.5 Automatic Acceleration/Deceleration

When the axis begins to move and before it stops, the system can automatically accelerate/ decelerate contributed to smooth speed to reduce impinge of run starting and stopping. The system is employed with accelerations/decelerations as follows:

Rapid traverse: S acceleration/deceleration

Rapid traverse: Exponential acceleration/deceleration

Thread cutting: Exponential/linear acceleration/deceleration

Manual feed: Exponential acceleration/deceleration

Handwheel feed: Exponential acceleration/deceleration

Step feed: Exponential acceleration/deceleration

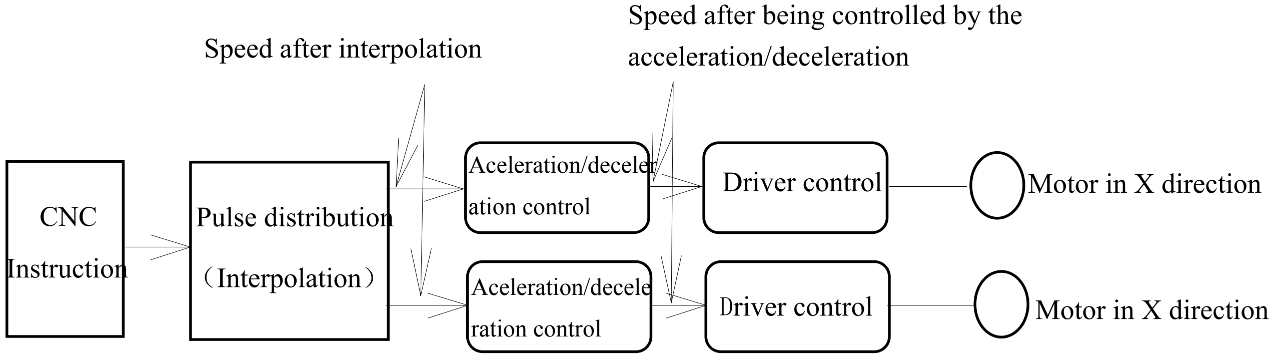
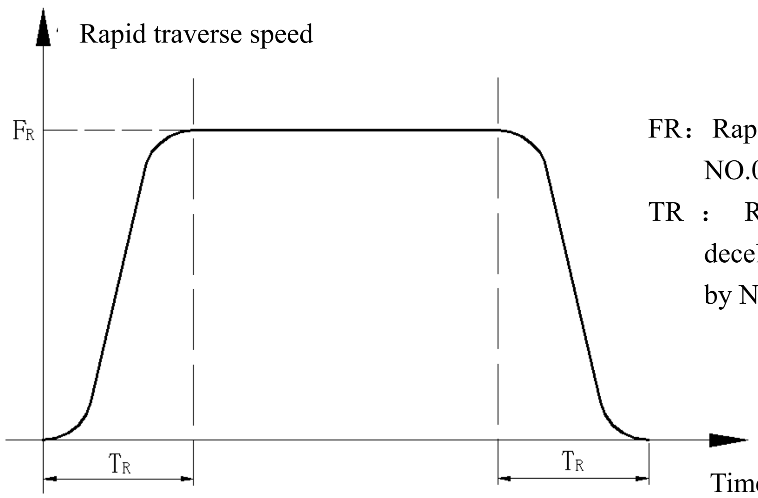


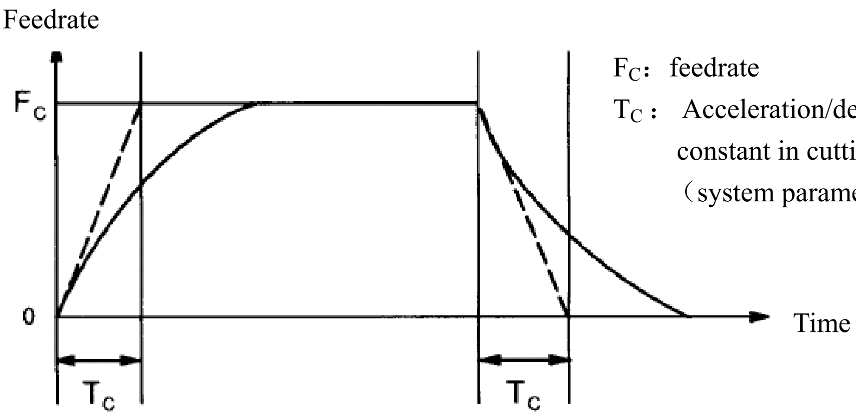
Fig. 2-9



FR: Rapid traverse speed defined by NO.022, NO.023

TR : Rapid traverse acceleration/ deceleration time constant defined by NO.022, O.023

Fig. 2-10 Curve of rapid traverse speed



F_C: feedrate

T_C : Acceleration/deceleration time constant in cutting feed (system parameter NO.029)

Fig. 2-11 Curve of cutting feedrate and manual feerate

The system is employed with exponential acceleration/deceleration, a transitive arc, which is not positioned exactly at the intersection of two path and there is contour error between actual path and programming path, is formed at a path intersection of neighboring two blocks in cutting feed caused by acceleration/ deceleration when the system parameter NO.007 Bit3 is 0. To avoid the contour error, execute G04 in two blocks or set NO.007 Bit3 to 1. At this moment, the previous block runs and positions exactly to its end point with zero mm/min and then the system starts to execute the next block, which increases program's running time and reduces machining efficiency.

The system executes the transition of program as Table 2-3 between neighboring blocks.

Table 2-3

Previous block / Next block	Rapid positioning	Cutting feed	No traversing
Rapid positioning	X	X	X
Cutting feed	X	O	X
No traversing	X	X	X

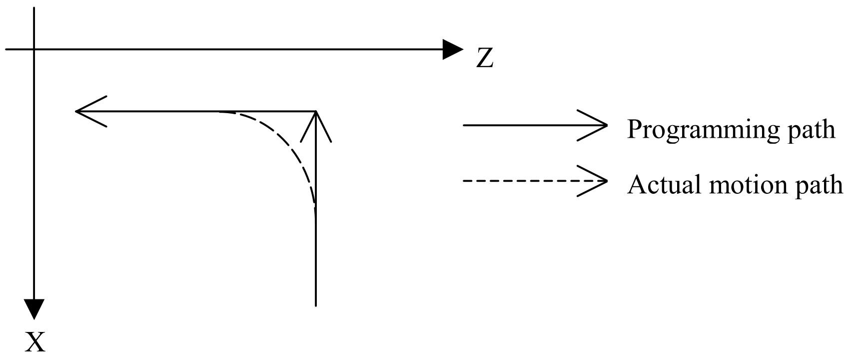
Note: X: The next block can be executed after the previous block is exactly positioned on its end point.

O: Acceleration/deceleration is employed for each axis between the neighboring blocks and there is a transitive arc (it is not exactly positioned)in the intersection path.

Example：(NO.007 Bit3=0):

G01 U-100；（Traverse in X negative direction）

W-200；　　（Traverse in Z negative direction）



Fi.g 2-12

Chapter 3 G INSTRCUTIONS

3.1 INTRODUCTION

G instruction consists of instruction address G and its following 1～2 bits instruction value, used for defining the motion mode of tool relative to the workpiece, defining the coordinates and so on. Refer to G instructions as Fig. 3-1

G □□

Instruction value（00～99，the leading zero can be omitted）

Instruction address G

G words are divided into 5 groups (00, 01, 02, 03). G words in the different groups can be input to the same block and the last one is valid when two or more G words in the same group are input. The words in the different groups without the same parameter (word) can be in the same block and their functions are valid without sequence at the same time. The system alarms when G words do not belong to Table 3-1 or they are optional functions without being supplied.

Table 3-1 G function list

Word	Group	Function	Remark
		Rapid traverse movement	Initial mode G
G01		Linear interpolation	
G02		Circular interpolation（CW）	
G03	01	Circular interpolation（CCW）	Modal G instructions
G32		Thread cutting	
G90		Axial cutting cycle	
G92		Thread cutting cycle	
G94		Radial cutting cycle	
G04		Dwell time preset	
G28		Machine reference point automatic return	
G50		Setting workpiece coordinates system	
G65		Macro instruction	
G70		Finishing cycle	
G71	00	Axial roughing cycle	Non-modal G instructions
G72		Radial roughing cycle	
G73		Closed cutting cycle	
G74		Axial grooving cycle	
G75		Radial grooving cycle	
G76		Multiple thread cutting cycle	
G96	02	Constant surface speed ON	Modal G
G97		Constant surface speed OFF	Initial mode G
G98	03	Feed per minute	Initial mode G
G99		Feed per rev	Modal G
G40		Deselect cutter radius compensation	Initial mode G
G41	04	Tool nose radius compensation to left of contour (option)	Modal G instruction
G42		Tool nose radius compensation to right of contour （optional）	

3.1.1 MODAL, NON-MODAL and INITIAL MODE

G instructions are divided into 5 groups (00, 01, 02, 03, 04). The instructions in the 00 group are non-modal and ones in other groups are modal, and G00, G97, G98, G40 are initial mode.

After G instructions are executed, their defined functions and states are valid until they are changed by others in the same group, which instructions are called modal G instructions. After the modal G words are executed, and before their defined functions and states are changed, the G instruction cannot be input again when they are executed by the following block.

The defined function and state are valid one time after G instruction is executed, and the G word must be input again when it is executed every time, which instruction is called non-modal G instruction.

After the system is switched on, the valid modal G instructions which are not executed their functions or states are called initial mode G instruction. Take it as the initial mode G instruction to be executed when it is not be input after the system is switched on. The initial words of the system include G00, G40, G97, G98.

3.1.2 OMIT a WORD

To simplify the programming, their instruction values are reserved after executing words in Table 3-2. If the words are contained in the previous blocks, they cannot be input when using the words with the same values and definitions in the following blocks.

Table 3-2

Instruction address	Function	Initial value when power on
U	Cutting depth in G71	NO.51 parameter value
U	Move distance of tool retraction in X direction in G73	NO.53 parameter value
W	Cutting depth in G72	NO.51 parameter value
W	Move distance of tool retraction in Z direction in G73	NO.54 parameter value
R	Move distance of tool retraction in G71, G72 cycle	NO.52 parameter value
R	Cycle times of stock removal in turning in G73	NO.55 parameter value
R	Move distance of tool retraction after cutting in G74,G75	NO.56 parameter value
R	Allowance of finishing in G76	NO.60 parameter value
R	Taper in G90,G92,G94,G96	0
（G98）F	Feed rate per minute（G98）	NO.030 parameter value
（G99）F	Feedrate per rev （G99）	0
F	Metric pitch（G32,G92,G76）	0
I	Inch pitch（G32,G92,）	0
S	Spindle speed specified（G97）	0
S	Spindle surface speed specified（G96）	0
S	Spindle speed switching value output	0

Instruction address	Function	Initial value when power on
P	Finishing times of thread cutting in G76； Tool retraction width of thread cutting in G76; Angle of tool nose of thread cutting in G76；	NO.57 parameter value NO.19 parameter value NO.58 parameter value
Q	Min. cutting value in G76	NO.59 parameter value

Note 1: For the instruction addresses with functions (such as F, used for feedrate per minute, feedrate per rev and metric pitch and so on), they can be omitted not to input when executing the same function to definite words after the words are executed. For example, after executing G98 F_ without executing the thread instruction, the pitch must be input with F word when machining metric thread;

Note 2:They can be omitted not to input when the address characters X（U）, Z（W）are the coordinates of end point of block and the system defaults the current absolute coordinates in X or Z direction to the coordinate value of end point of block;

Note 3: The corresponding words must be input when the instruction addresses which are not in Table 3-2 are used.

Example 1:

```
O0001；
G0 X100 Z100;      （rapid traverse to X100 Z100; the modal G0 is valid）
X20 Z30;           （rapid traverse to X20 Z30; the modal G0 is not input）
G1 X50 Z50 F300;   （linear interpolation to X50 Z50, feedrate 300mm/min; the modal G1 is valid）
X100;              （linear interpolation to X100 Z50, feedrate 300mm/min; Z coordinates is not input and
                     is the current coordinates Z50; F300 is kept, G1 is modal and is not input）
G0 X0 Z0;          （rapid traverse to X0 Z0 and the modal G0 is valid）
M30;
```

Example 2:

```
O0002;
G0 X50 Z5;         （rapid traverse to X50 Z5）
G04 X4;            （dwell 4 seconds）
G04 X5;            （dwell 5 seconds again, G04 is non-modal and is needed to input again）
M30;
```

Example 3（the first run after power on）:

```
O0003;
G98 F500 G01 X100 Z100;    （Feedrate per minute 500mm/min in G98）
G92 X50 W-20 F2 ;          （F value is a pitch and must be input in thread cutting）
G99 G01 U10 F0.01          （Feedrate per rev in G99 must be input again）
G00 X80 Z50 M30;
```

3.1.3 RELATED DEFINITIONS

In the user manual, the definitions of Word are as follows except for the especial explanations:

Starting point: position before the current block runs;
End point: position after the current block ends;
X: absolute coordinates of end point in X direction;
U: different value of absolute coordinates between starting point and end point;
Z: absolute coordinates of end point in Z direction;
W: different value of absolute coordinates between starting point and end point;
F: cutting feedrate.

3.2 RAPID TRAVERSE MOVEMENT G00

Instruction format: : G00 X（U）__ Z（W）__;

Instruction function: X, Z axis rapidly traverses at the respective traverse speed to the end points from their starting point. G00 is initial instruction as Fig.3-1.

X, Z axis traverses at the respective traverse speed, the short axis arrives the end point and the length axis continuously moves to the end point and the compound path maybe be not linear.

Instruction specification: G00 is initial mode;

Range of X, U, Z, W: -9999.999～+9999.999mm;

Can omit one or all instruction addresses X(U), Z(W). The coordinate values of starting point and end point are the same when omitting one instruction address; the end point and the starting point are in the same position when all are omitted. X, Z are valid, and U, W are invalid when X, U, Z and W are in the same one block.

Instruction path:

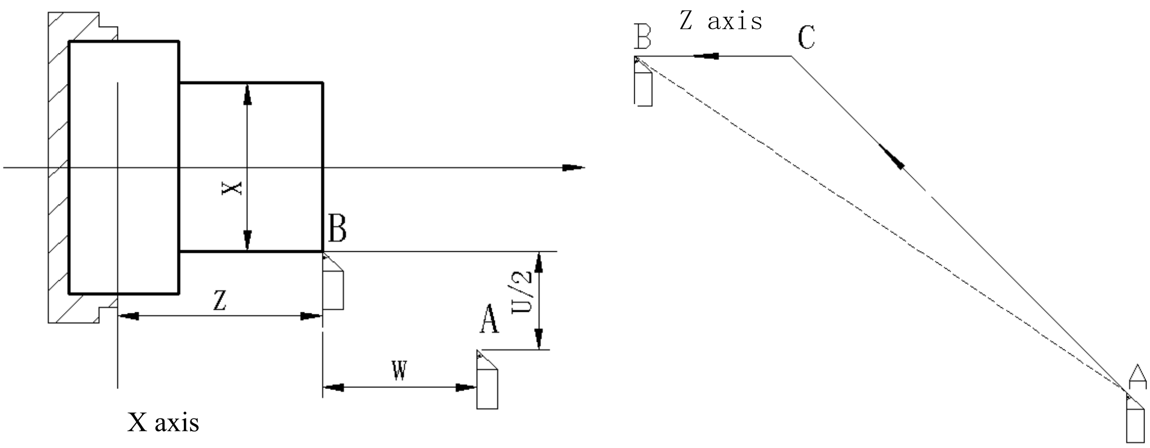


Fig. 3-1

The respective rapid traverse speed of X,Z axis is defined by the system parameter NO.022, NO.023, and their traverse speed can changed by rapid override key on the machine control panel.

Example: The tool rapidly traverses to B from A as Fig. 3-2.

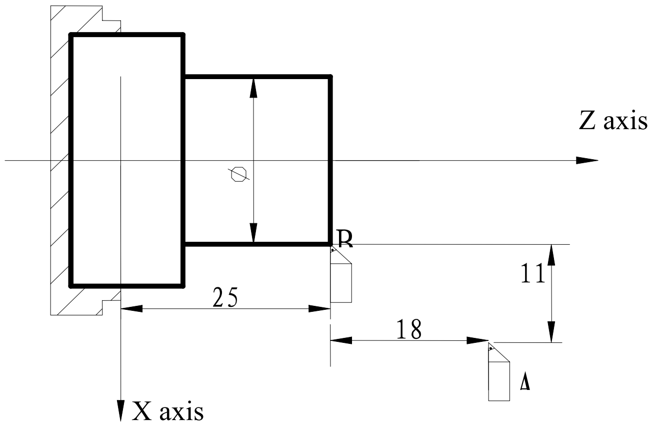


Fig. 3-2

G0 X20 Z25； （absolute programming）
G0 U-22 W-18； （incremental programming）
G0 X20 W-18； （compound programming）
G0 U-22 Z25； （compound programming）

3.3 LINEAR INTERPOLATION G01

Instruction format：G01 X（U）_ Z（W）_ F_;

Instruction function: the movement path is a straight line from starting point to end point as Fig.3-3.

Instruction specification: G01 is modal.

Range of X, U, Z, W: -9999.999～+9999.999mm；

Can omit one or all instruction addresses X(U), Z(W). The coordinate values of starting point and end point are the same when omitting one instruction address; the end point and the starting point are in the same position when all are omitted.

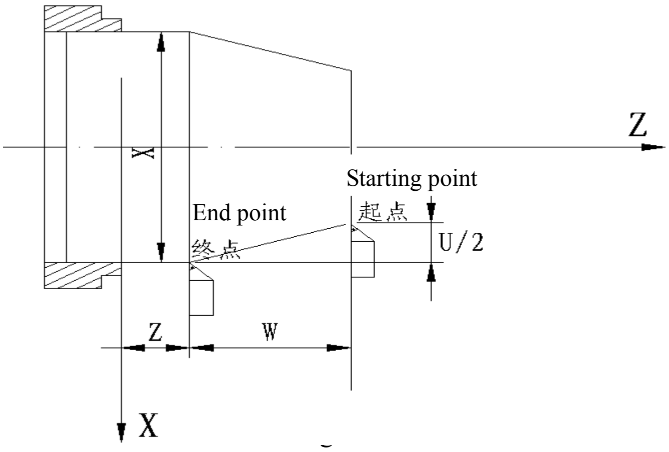
F instruction value is the compound speed of vector of instantaneous speed in X and Z direction and the actual cutting feedrate is the product between the feedrate override and F instruction value.

After F instruction value is executed, it has been reserved unless the new one is executed. Do not repeat it when the following G instructions adopt functions of F word.

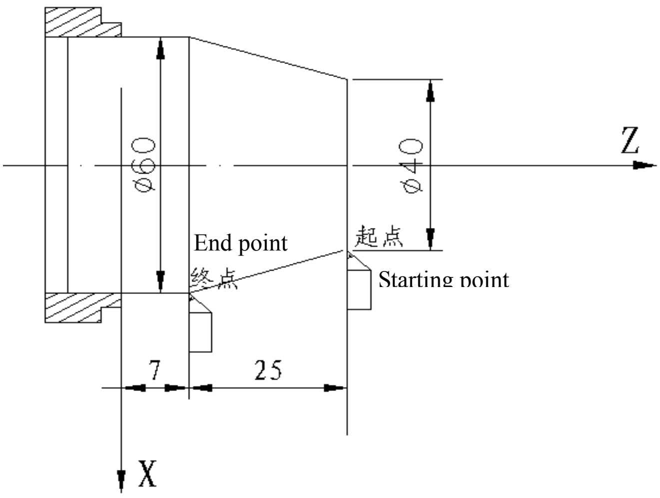
Range of value as follows:

Instruction function	G98（mm/min）	G99（mm/rev）
Range	1~8000	0.001~500

Instruction path：



Example: Cutting path from Φ40 toΦ60 as follows:



Fi.g 3-4

Program：

G01 X60 Z7 F500； （Absolute programming）
G01 U20 W-25； （Incremental programming）
G01 X60 W-25； （Compound programming）
G01 U20 Z7； （Compound programming）

3.4 CIRCULAR INTERPOLATION G02, G03

Instruction format ： { G02 / G03 } X（U）__ Z（W）__ { R__ / I__ K__ }

Instruction function: G02 movement path is clockwise(rear toolpost coordinate system/counterclockwise (front toolpost coordinates system) arc from starting point to end point as Fig.3-5.

G03 movement path is clockwise(rear toolpost coordinate system/ counterclockwise (front toolpost coordinates system) arc from starting point to end point as Fig. 3.6.

Instruction path ：

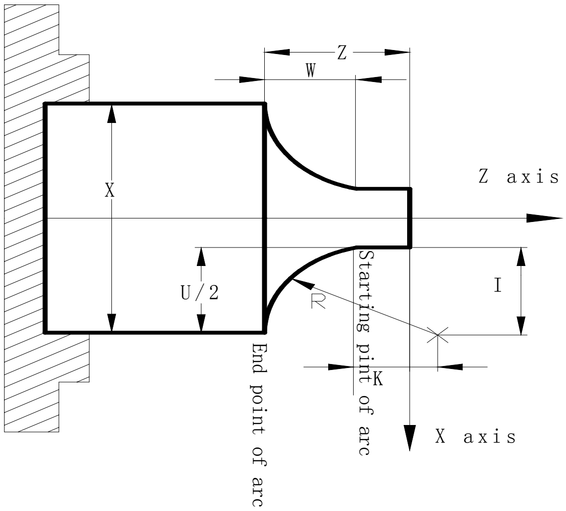


Fig.3-5 G02 path

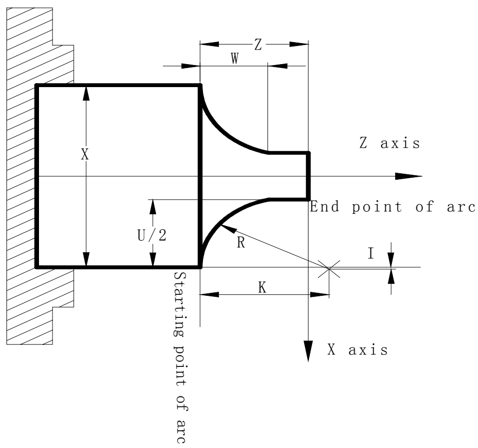


Fig.3-6 G03 path

Instruction specification: G02, G03 are modal;

R：arc radius（0～9999.999mm）;

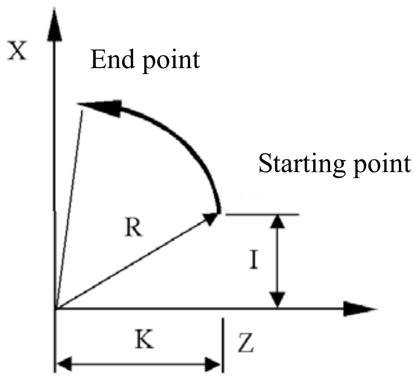
I：difference value between circle center and starting point of arc in radius in X direction（-9999.999～9999.999mm）;

K：difference value between circle center and starting point of arc in Z direction（-9999.999～9999.999mm）;

Center point of arc is specified by address I, K which separately corresponds to amount(increment) of vector from starting point to center point of arc in X, Z direction as Fig. 3-6-1.

I=Coordinates of center point-that of starting point in X direction; K= Coordinates of center point-that of starting point in Z direction;

I, K are with sign symbol, directions of I, K are the same those of X, Z axis.



Arc direction: G02/G03 direction (clockwise/counterclockwise)is opposite on the front toolpost coordinate system and the rear one as Fig.3-7:

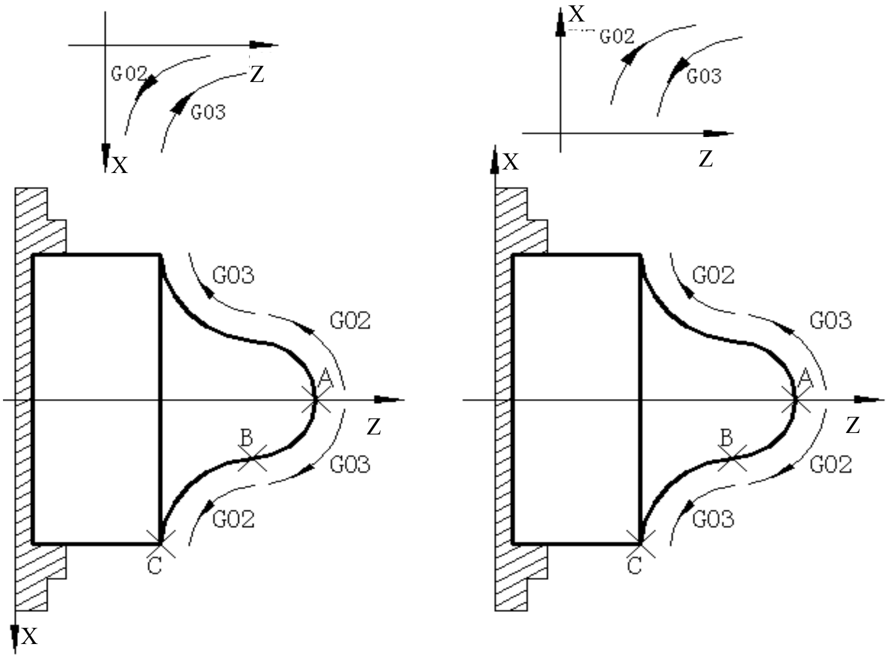


Fig. 3-7

Cautions:

- When I = 0 or K = 0, they can be omitted; one of I, K or R must be input, otherwise the system alarms.
- R is valid and I, K are invalid when they are input at the same time;
- R value must be equal to or more than half distance from starting point to end point, and the system alarms if the end point is not on the arc defined by R instruction;
- Omit all or one of X(U), Z(W); coordinates of starting point and end point of this axis are the same when omitting ones, the path is a full circle（360°） in G02/G03 when center point are specified by I,K; the path is 0（0°） when center point is specified by R.
- R should be used for programming. The system executes in R=$\sqrt{I^2+K^2}$ to ensure starting point and end point of arc path are the specified ones in I,K programming.
- When the distance from center point to end point is not equal to R(R＝$\sqrt{I^2+K^2}$) in I,K programming, the system automatically adjusts position of center point to ensure starting point and end point of arc path are the specified ones; when the distance from center point to end point is more than 2R, and the system alarms.
- Arc path can be more than and less than 180° when R is commanded, the arc is more than180° when R is negative, and it is less than or equal to 180° when R is positive.

Example: Arc cutting path fromΦ45.25 toΦ63.06 as Fig. 3-8.

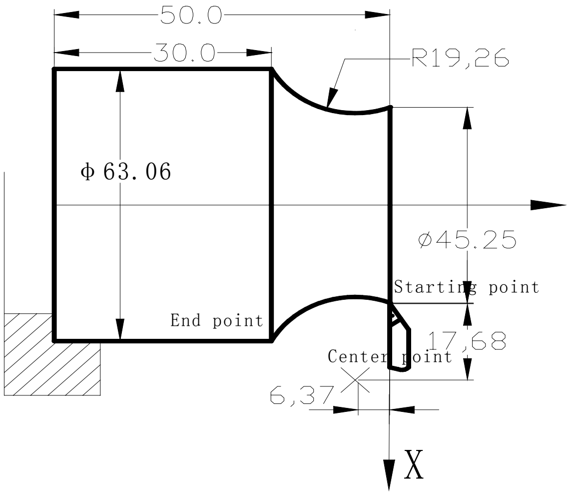


Fig.3-8

Program：
G02 X63.06 Z-20.0 R19.26 F300 ； or
G02 U17.81 W-20.0 R19.26 F300 ； or
G02 X63.06 Z-20.0 I17.68 K-6.37 ； or
G02 U17.81 W-20.0 I17.68 K-6.37 F300

Compound programming in G02/G03：

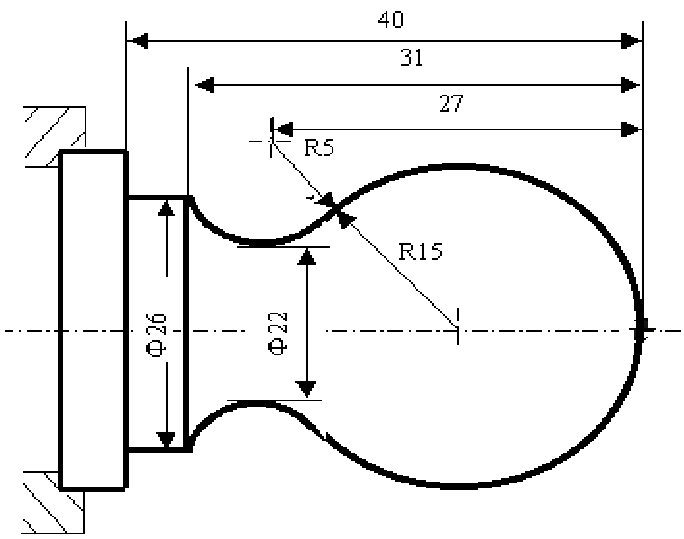


Fig. 3-9 Arc programming

- Program：O0001
  N001 G0 X40 Z5； （Rapid position）
  N002 M03 S200； （Start spindle）
  N003 G01 X0 Z0 F900； (Approach workpiece)
  N005 G03 U24 W-24 R15； （Cut R15 arc）
  N006 G02 X26 Z-31 R5； （Cut R5 arc）
  N007 G01 Z-40； (Cut Φ26)
  N008 X40 Z5； (Return to starting point)
  N009 M30； （End of program）

3.5 DWELL G04

Instruction format：G04 P__ ； or

G04　X__ ；or

G04　U__ ；or

G04;

Instruction function: X and Z axis stop, the modal of G instructions and the reserved data, state are not changed, and execute the next block after dwelling the defined time.

Instruction specification: G04 is non-modal.

The dwell time is defined by the word P__,X__ or U__.

Range of P, X, U: 0.001～99999.999 seconds.

Time of P__, X__ or U__ is as follows:

Table 3-3

Instruction address	P	U	X
Unit	0.001 second	Second	Second

Cautions:

- The system exactly stop a block when P, X, U are not input or P, X , U specify negative values.
- P is valid when P, X, U are in the same block; X is valid when X, U are in the same block.
- When the system executes the feed hold in G04, dwell can be executed after the current delay time.

3.6　MACHINE REFERNCE POINT RETURN G28

Instruction format：G28　X（U）__　Z（W）_ ；

Instruction function: the tool rapid traverses to the middle point defined by X(U), Z(W) from starting point and then return to reference point of machine.

Instruction specifications:

G28 is no-modal.

X：absolute coordinates of middle point in X direction；

U：Difference value of absolute coordinates between middle point and starting point in X direction；

Z：absolute coordinates of middle point in Z direction；

W：Difference value of absolute coordinates between middle point and starting point in Z direction.

Can omit all or one of X(U), Z(W) as follows:

Table 3-4

Instruction	**Function**
G28　X（U）__	Return to machine reference point in X direction and remain in previous position in Z direction
G28　Z（W）__	Return to machine reference point in Z direction and remain in previous position in X direction

Instruction	Function
G28	X, Z axis are in the previous positions and continuously execute nest block
G28　X（U）__　Z（W）__	Return to machine reference point in X, Z direction

Running path（as Fig. 3-10）：

（1）Rapid traverse to middle point of specified axis from current position（A point→B point）；

（2）Rapid traverse to reference point from the middle point（B point→R point）；

(3) If the machine is not locked, LED is ON when the machine reference point return is completed.

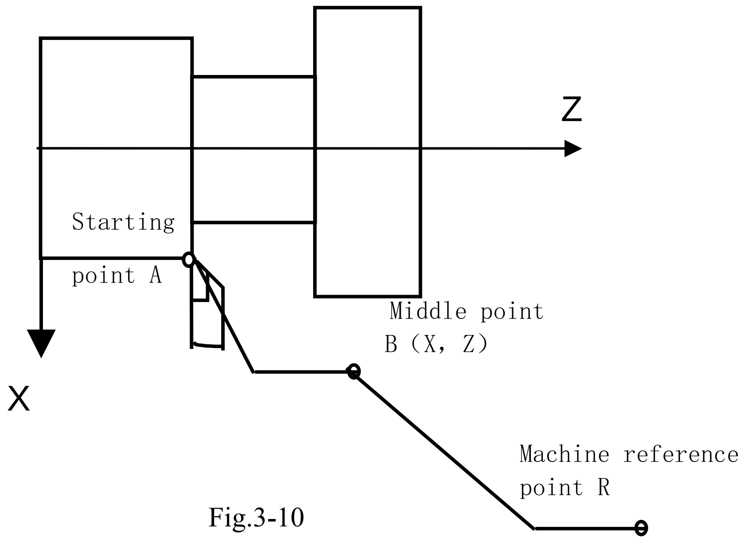


Fig.3-10

Note 1: Machine reference point returns in Jog mode and in G28 are the same and their deceleration signals and signals per rev must be detected;

Note 2: X and Z axis move at the respectively rapid traverse speed from A to B and from B to R, and so the path is not always a straight line.

Note 3: The system cancels the tool length compensation after executing G28 to perform the reference point return of machine;

Note 4: Do not execute G28 and machine reference point return without the reference point signal on the machine.

3.7　WORKPIECE COORDINATE SYSTEM G50

Instruction format：G50　X（U）___　Z（W）___；

Instruction function: define the absolute coordinates of current position and create the workpiece coordinates system(called floating coordinates system) by setting the absolute coordinates of current position in the system. After executing G50, the system takes the current position as reference point of program (program reference point) and returns to the reference point after executing the reference point return of program. After the workpiece coordinate system is created, input the coordinate values in the coordinate system until the next workpiece coordinate system is created again when executing the absolute coordinates programming.

Instruction specifications:

G50 is non-modal;

X: New absolute coordinates of current position in X direction;

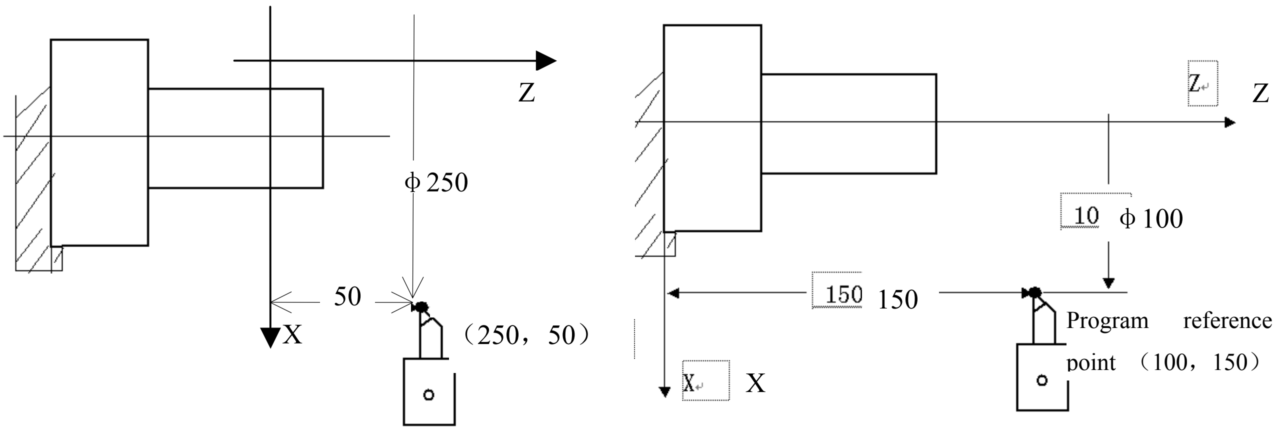
U: Different value between the new absolute coordinates of current position in X direction and the absolute coordinates before executing instructions;

Z: New absolute coordinates of current position in Z direction;

W: Different value between the new absolute coordinates of current position in X direction and the absolute coordinates before executing instructions;

In G50, X（U）or Z（W）are not input, the system does not change current coordinates position as program reference point; when X（U）and Z（W）are not input, the system takes the previous setting position as program reference point.

Example：



Before setting coordinates system with G50

After setting coordinates system with G50

Fig.3-11

As Fig.3-11, create the above-mentioned workpiece coordinate system and set (X100 Z150) to the reference point of program after executing “G50 X100 Z150”.

Note: When NO.003 Bit4 is 1(executing tool compensation by coordinates offset), T function is executed, motion instruction is not executed and the system creates workpiece coordinate system with G50, the displayed coordinate value are ones which are defined by G50 adding or subtracting tool compensation value which is not executed.

Current tool compensation state	Executing motion instruction	Coordinate value after executing G50 X20 Z20	No. 01 tool compensation value
T0100 or T0101	G0 X_ Z_	X：20 Z：20	X：12 Z：23
	No executing motion instruction	Coordinate value after executing G50 X20 Z20	

	※※※	X：8　　Z：-3 or X：32　　Z：43	

3.8　FIXED CYCLE INSTRUCTIONS

To simplify programming, the system defines G instruction of single machining cycle with one block to complete the rapid traverse to position, linear/thread cutting and rapid traverse to return to the starting point:

G90: axial cutting cycle;

G92: thread cutting cycle;

G94: radial cutting cycle;

G92 will be introduced in Section ***Thread Function***

3.8.1　AXIAL CUTTING CYCLE G90

Instruction format：G90 X（U）__ Z（W）__ F__；　　（cylinder cutting）

G90 X（U）__ Z（W）__ R__ F__；（taper cutting）

Instruction function: From starting point, the cutting cycle of cylindrical surface or taper surface is completed by radial feeding(X axis) and axial(Z axis or X and Z) cutting.

Instruction specifications:

G90 is modal;

Starting point of cutting：starting position of linear interpolation(cutting feed)

End point of cutting：　　end position of linear interpolation(cutting feed)

X：absolute coordinates of cutting end point in X direction

U：different value of absolute coordinates between end point and starting point of cutting in X direction

Z：different value of absolute coordinates between end point and starting point of cutting in Z direction

W：different value of absolute coordinates between end point and starting point of cutting in Z direction

R：different value (radius value) of absolute coordinates between end point and start point of cutting in X direction. When the signs of R is not the same that of U, R｜≤｜U/2｜

Cycle process:

① The tool rapidly traverses from starting point to cutting starting point in X direction;

② Cutting feed(linear interpolation) from the cutting starting point to cutting end point;

③ Retract the tool at feedrate in X direction (opposite direction to the above-mentioned ①), and return to the position which the absolute coordinates and the starting point are the same;

④ The tool rapidly traverses to return to the starting point and the cycle is completed.

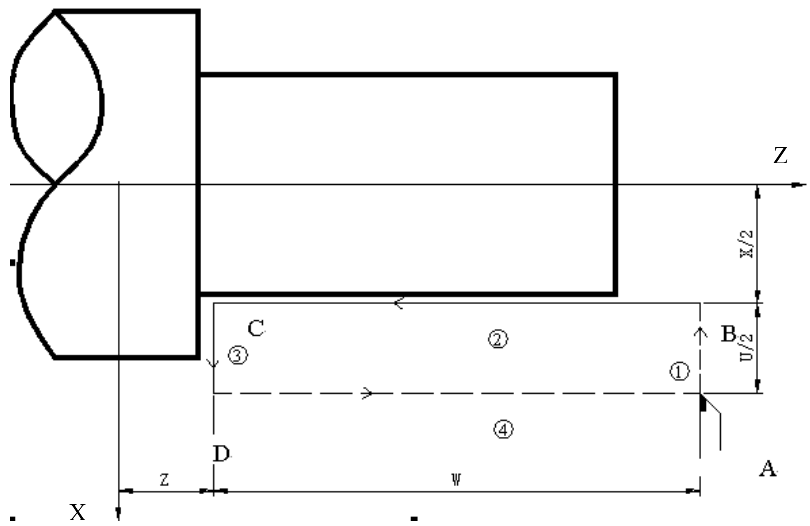


Fig.3-12

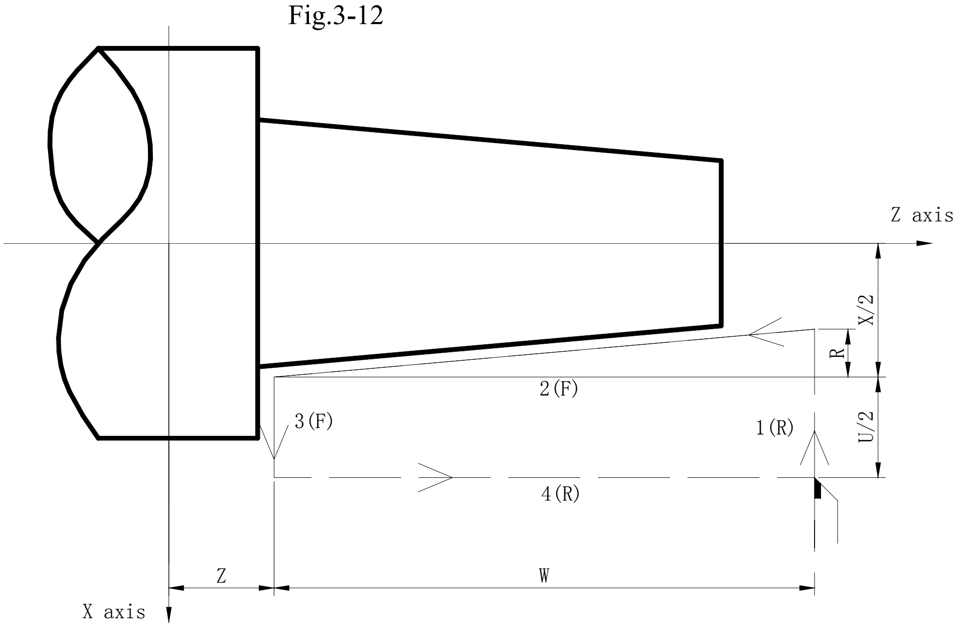


Fig.3-13

Cutting path: Relative position between cutting end point and starting point with U,W is as Fig.3-14:

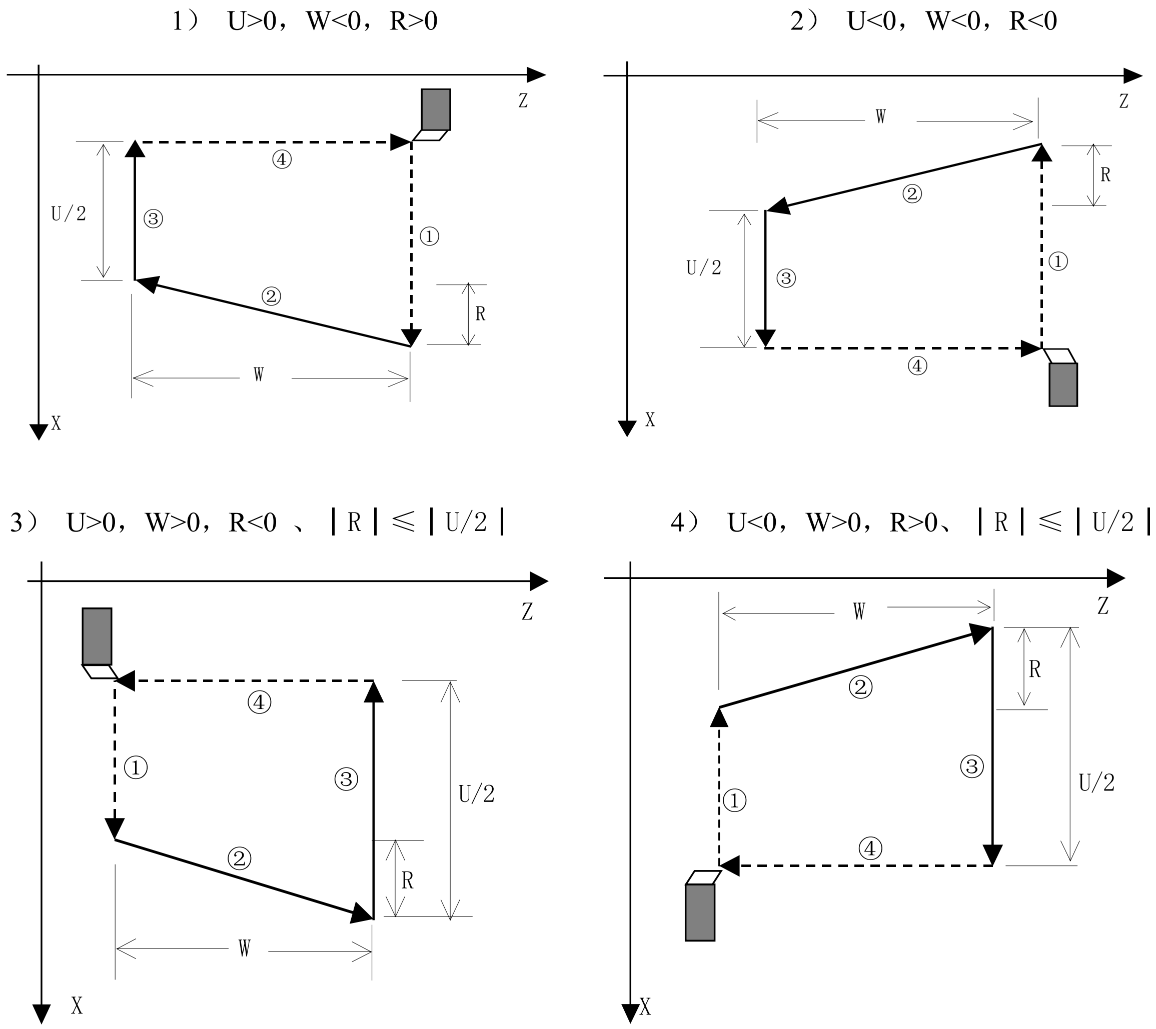


Fig. 3-14

Example: Fig. 3-15，rod Φ125×110

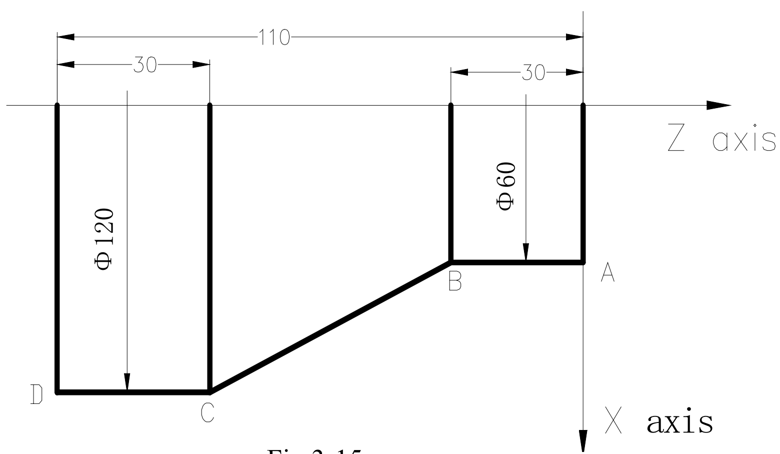


Fig.3-15

```
Program ：O0002；
M3 S300 G0 X130 Z3；
G90 X120 Z-110 F200；          （A→D，cut Φ120）
X110 Z-30；
X100；
X90；                          （A → B，6 times cutting cycle
X80；                             Φ60,increment of 10mm）
X70；
X60；
G0 X120 Z-30；
G90 X120 Z-44 R-7.5 F150；
Z-56 R-15                      （B→C，4 times taper cutting）
Z-68 R-22.5
Z-80 R-30
M30；
```

3.8.2 RADIAL CUTTING CYCLE G94

Instruction format：G94 X（U）__ Z（W）__ F__；   （face cutting）

G94 X（U）__ Z（W）__ R__ F__；（taper face cutting）

Instruction function: From starting point, the cutting cycle of cylindrical surface or taper surface is completed by radial feeding(X axis) and axial(Z axis or X and Z) cutting.

Instruction specifications:

G94 is modal;

Starting point of cutting: starting position of linear interpolation(cutting feed).Unit:mm;

End point of cutting: end position of linear interpolation(cutting feed) Unit:mm;

X：absolute coordinates of end point of cutting in X direction Unit:mm;

U：different value of absolute coordinates from end point to starting point of cutting in X direction, Unit:mm;

Z：absolute coordinates of end point of cutting in Z direction, Unit:mm;

W：different value of absolute coordinates from end point to starting point of cutting in X direction, Unit:mm;

R：different value(R value) of absolute coordinates from end point to starting point of cutting in X direction. When the sign of R is not the same that of U, R, ｜R｜≤｜W｜.

Radial linear cutting is as Fig. 3-16, radial taper cutting is as Fig. 3-17, Unit:mm

Cycle process:

① The tool rapidly traverses from starting point to cutting starting point in Z direction;

② Cutting feed (linear interpolation) from the cutting starting point to cutting end point;

③ Retract the tool at the cutting feedrate in Z direction (opposite direction to the above-mentioned ①), and return to the position which the absolute coordinates and the starting point are the same;

④ The tool rapidly traverses to return to the starting point and the cycle is completed.

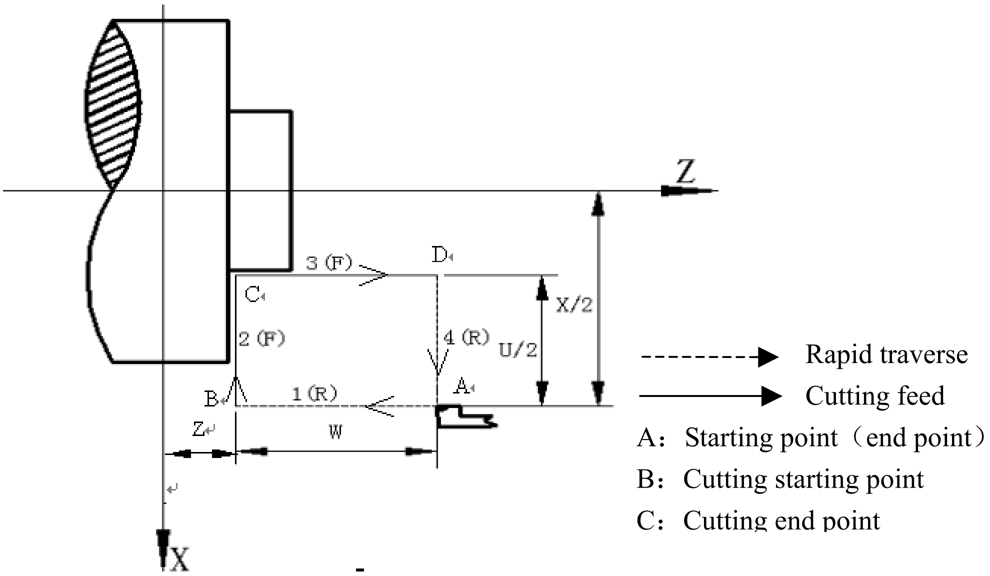


Fig.3-16

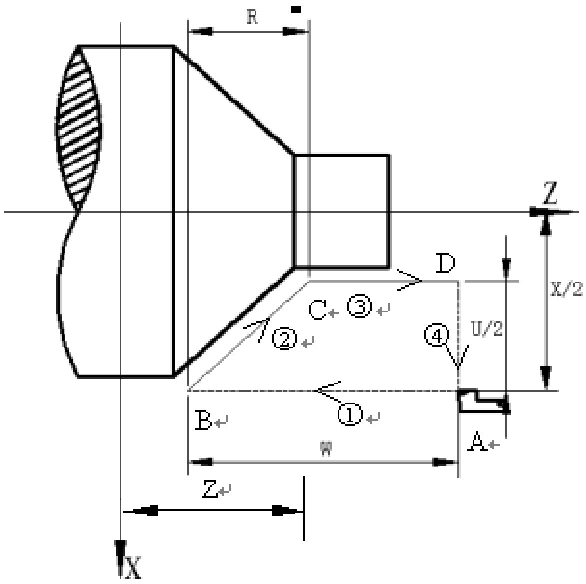


Fig.3-17

Cutting path: Relative position between cutting end point and starting point with U,W is as Fig.3-18:

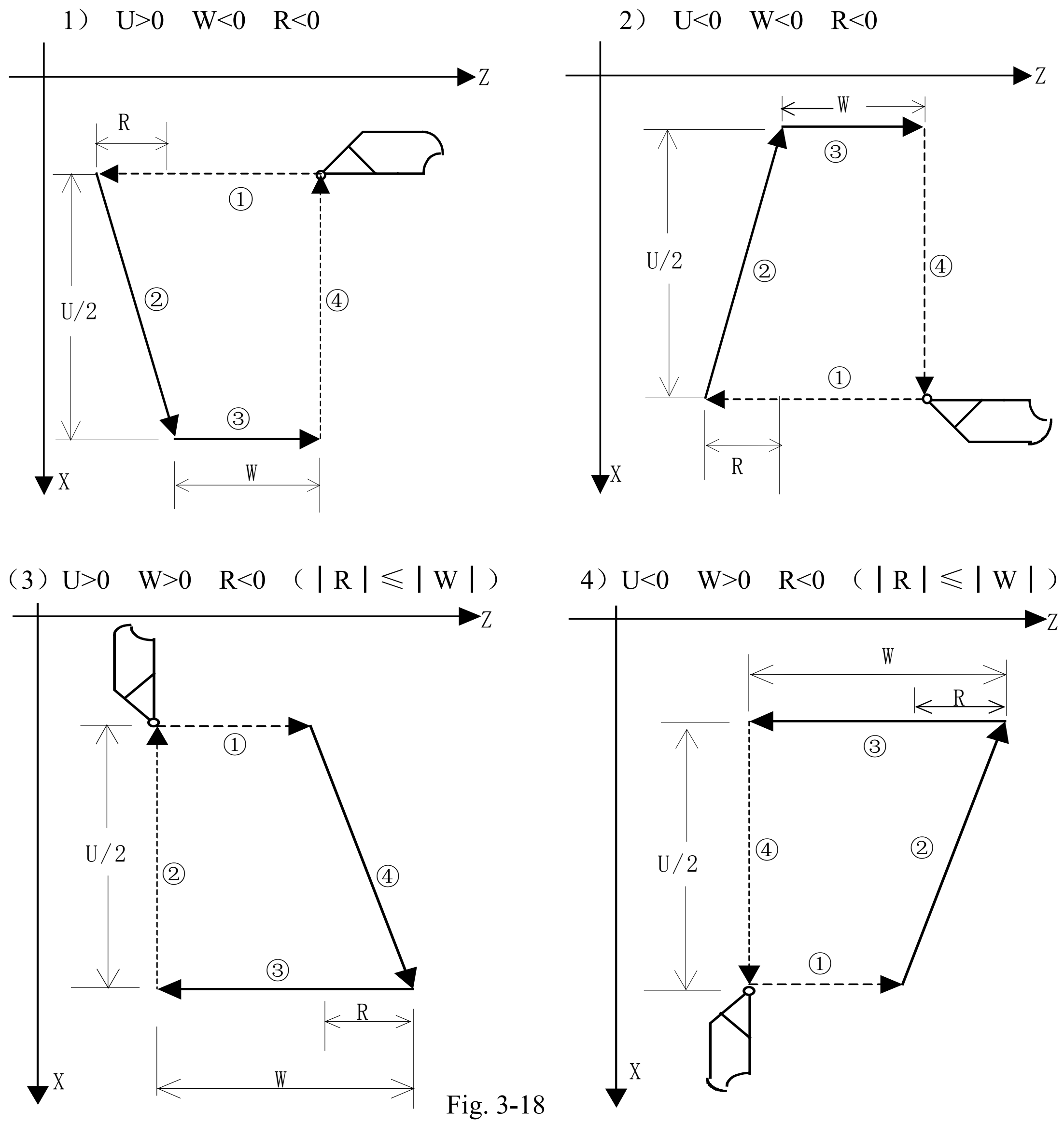


Fig. 3-18

Example：Fig. 3-19，rod Φ125×112

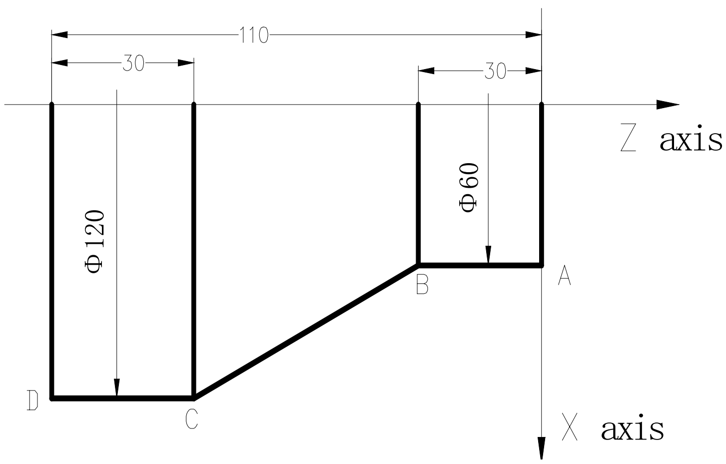


Fig.3-19

Program：O0003；
G00 X130 Z5 M3 S1；
G94 X0 Z0 F200 　　Face cutting
X120 Z-110 F300；　（Outer cutting Φ120）
G00 X120 Z0
G94 X108 Z-30 R-10
X96 R-20
X84 R-30 　　（C→B→A，cutting Φ60）
X72 R-40
X60 R-50；
M30；

3.8.3 CAUTIONS OF FIXED CYCLE INSTRUCTIONS

1）After X（U）, Z（W）, R are executed in the canned cycle instruction, their instruction values are value if X（U），Z（W），R are not redefined by executing a new canned cycle instructions. The instruction values of X（U），Z（W），R are cleared if non-modal G instruction（00 Group） except for G04 or G00, G01,G02, G03, G32 is executed.

2）In MDI mode, the previous canned cycle can be executed by pressing the cycle start key after the canned cycle is completed.

3）One cycle cannot be executed repetitively in G90～G94 when the next block of G90～G94 is M, S, T instruction; the previous cycle is executed repetitively in G90～G94 when the next block is ended(EOB;).

Example： …
N010 G90 X20.0 Z10.0 F400；
N011 ； （executed G90 one time repetitively ）
…

4）Pause or single block is executed in G90, G94, the single block stops after the tool moves end point of current path.

3.9 MULTIPLE CYCLE INSTRUCTIONS

Multiple cycle instructions of the system includes axial roughing cycle G71, radial roughing cycle G72, closed cutting cycle G73, finishing cycle G70, axial grooving multiple cycle G74, axial grooving multiple cycle G75 and multiple thread cutting cycle G76. When the system executes these instructions, it automatically counts the cutting times and the cutting path according to the programmed path, travels of tool infeed and tool retraction, executes multiple machining cycle（tool infeed →cutting→retract tool→tool infeed ）, automatically completes the roughing, finishing workpiece and the starting point and the end point of instruction are the same one.

3.9.1 AXIAL ROUGHING CYCLE G71

Instruction format：G71 U（Δd） R（e） F__ S__ T__； (1)
G71 P（ns） Q（nf） U（Δu） W（Δw）； (2)

N（ns） . . . . . ;
. . . . ;
. . . . F;
. . . . S;
. . . .
.

N（nf） . . . . . ;   (3)

Instruction function: G71 is divided into three parts:

(1)：1st blocks for defining the travels of tool infeed and retract tool, the cutting feedrate, the spindle speed and the tool function when roughing;

(2)：2nd blocks for defining the block interval, finishing allowance;

(3)：3rd blocks for some continuous finishing path, counting the roughing path without being executed actually when executing G71.

According to the finishing path, the finishing allowance, the path of tool infeed and tool retract, the system automatically counts the path of roughing，the tool cuts the workpiece in paralleling with Z axis, and the roughing is completed by multiple executing the cutting cycle tool infeed→cutting→tool retraction. The starting point and the end point are the same one. The instruction is applied to the formed roughing of non-formed rod.

Relevant definitions:

Finishing path: the above-mentioned Part 3 of G71**(**ns～nf block)defines the finishing path, and the starting point of finishing path (starting point of ns block)is the same these of starting point and end point of G71, called A point; the first block of finishing path(ns block)is used for X rapid traversing or tool infeed, and the end point of finishing path is called to B point; the end point of finishing path(end point of nf block)is called to C point. The finishing path is A→B→C.

Roughing path: The finishing path is the one after offsetting the finishing allowance（Δu,Δw）and is the path contour formed by executing G71. A, B, C point of finishing path after offset corresponds separately to A',B',C'point of roughing path, and the final continuous cutting path of G71 is B'→C'point.

Δd：it is each travel（unit：mm，radius value）of tool infeed without in X direction when roughing, and the direction of tool infeed is defined by move direction of ns block. The instruction value Δd is reserved after executing U（Δd）and the value of system parameter NO.051 is rewritten to Δd×1000（unit：0.001 mm）. The value of system parameter NO.051 is regarded as the travel of tool infeed when U（Δd）is not input.

e: it is each travel（unit：mm，radius value）of tool infeed without in X direction when roughing, and the direction of tool retraction is opposite to that of tool infeed, the instruction value e is reserved and the value of system parameter NO.052 is rewritten to e×1000（unit：0.001 mm）after R（e）is executed. The value of system parameter NO.052 is regarded as the travel of tool retraction when R（e）is not input.

ns: Block number of the first block of finishing path.

nf: Block number of the last block of finishing path.

Δu：finishing allowance in X direction is -99.999~99.999 (unit: mm in diameter). the coordinates offset in X direction of roughing path compared to finishing path, i.e. the different value of absolute coordinates in X direction between A'and A. The system defaults Δu=0 when U（Δu） is not input, i.e. there is no finishing allowance in X direction for roughing cycle.

Δw：finishing allowance in Z direction is -99.999~99.999 with sign symbol (unit: mm in diameter). the coordinates offset in Z direction of roughing path compared to finishing path, i.e. the different value of absolute coordinates in Z direction between A'and A. The system defaults Δw=0 when W（Δw） is not input, i.e. there is no finishing allowance in Z direction for roughing cycle.

F: Feedrate; S: Spindle speed; T: Tool number, tool offset number.

M, S, T, F: They can be specified in the first G71 or the second ones or program ns～nf. M, S, T, F functions of M, S, T, F blocks are invalid in G71, and they are valid in G70 finishing blocks.

Execution process:(Fig. 3-20)

① Rapid traverse to A' from A point, the travel in X direction is Δu, and the travel in Z direction is Δw

② The travel in X direction from A'is Δd(tool infeed), ns block is for tool infeed at rapid traverse speed with G0, is for tool infeed at feedrate F with G71, and its direction of tool infeed is that of A→B point;

③ Cutting feeds to the roughing path in Z direction, and its direction is the same that of coordinates in Z direction A→B point;

④ The travel of tool retraction is e (45° straight line)at feedrate in X, Z direction, the directions of tool retraction is opposite to that of too infeed;

⑤ Rapid retract at rapid traverse speed in Z direction to the position which is the same that of the coordinates in Z direction;

⑥ After executing the tool infeed (Δd+e)again in X direction, the end point of traversing tool is still on the middle point of straight line between A'and B'(the tool does not reach or exceed B'), and after executing the tool infeed (Δd+e)again, execute ③；after executing the tool infeed (Δd+e)again, the end point of tool traversing reaches B'point or exceeds the straight line between A'→B'point and execute the tool infeed to B'point in X direction and the execute the next step;

⑦ Cutting feed from B'to C'point along the roughing path;

⑧ Rapid traverse to A from C'point and the program jumps to the next clock following nf block after G71 cycle is ended.

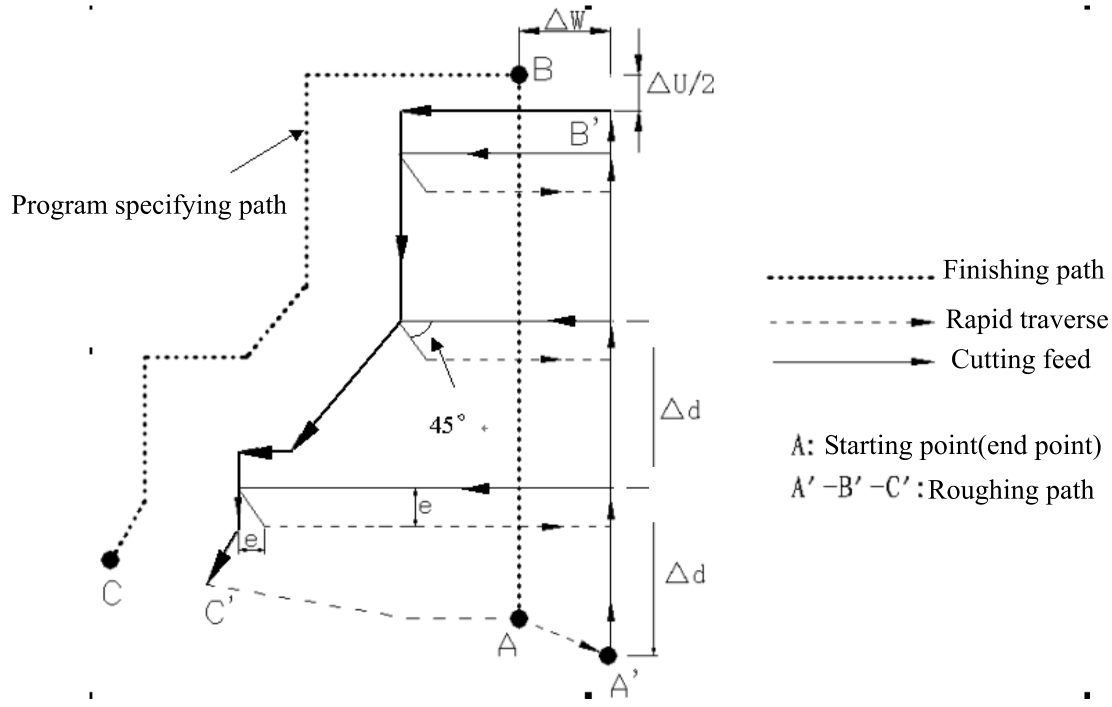


Fig. 3-20 G71 path

Instruction specifications:

- ns～nf blocks in programming must be followed G71 blocks. If they are in the front of G71 blocks, the system automatically searches and executes ns～nf blocks, and then executes the next program following nf block after they are executed, which causes the system executes ns～nf blocks repetitively;
- ns～nf blocks are used for counting the roughing path and the blocks are not executed when G71 is executed. F, S, T instructions of ns～nf blocks are invalid when G71 is executed, at the moment, F, S, T instructions of G71 blocks are valid. F, S, T of ns～nf blocks are valid when executing ns～nf to command G70 finishing cycle;
- There are G00,G01 without the word Z(W) in ns block, otherwise the system alarms ;
- The dimensions in X, Z direction must be changed monotonously (always increasing or reducing) for the finishing path;
- In ns～nf blocks, there are only G instructions: G01, G02, G03, G04, G96, G97, G98, G99, G40, G41,G42 and the system cannot call subprograms(M98/M99);
- G96, G97, G98, G99, G40, G41, G42 are invalid in G71 and valid in G70.　G96,G97,G98,
- When G71 is executed, the system can stop the automatic run and manual traverse, but return to the position before manual traversing when G71 is executed again, otherwise, the following path will be wrong;
- When the system is executing the feed hold or single block, the program pauses after the system has executed end point of current path;
- △d，△u are specified by the same U and different with or without being specified P,Q instructions;
- G71 cannot be executed in MDI, otherwise, the system alarms;
- There are no the same block number in ns~nf when compound cycle instructions are executed repetitively in one program.

Coordinate offset direction with finishing allowance:

Δu,Δw define the coordinates offset and its direction of finishing, and their sign symbols are as follows Fig. 3-21: B→C for finishing path, B'→C'for roughing path and A is the starting point.

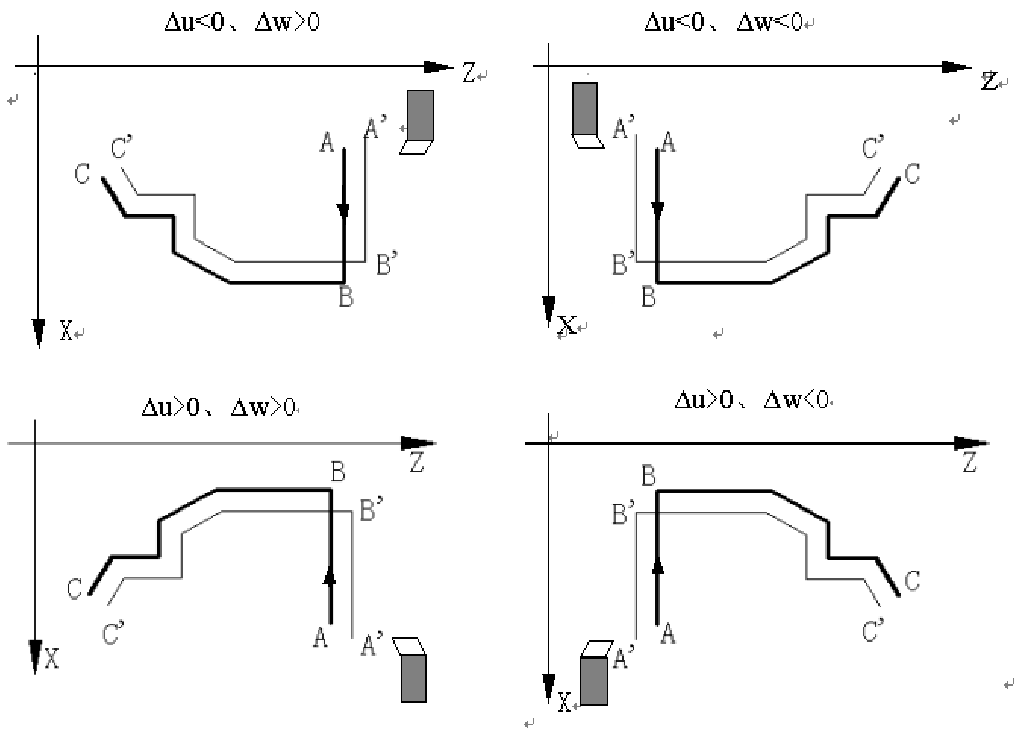


Fig.3-21

Example: Fig.3-22

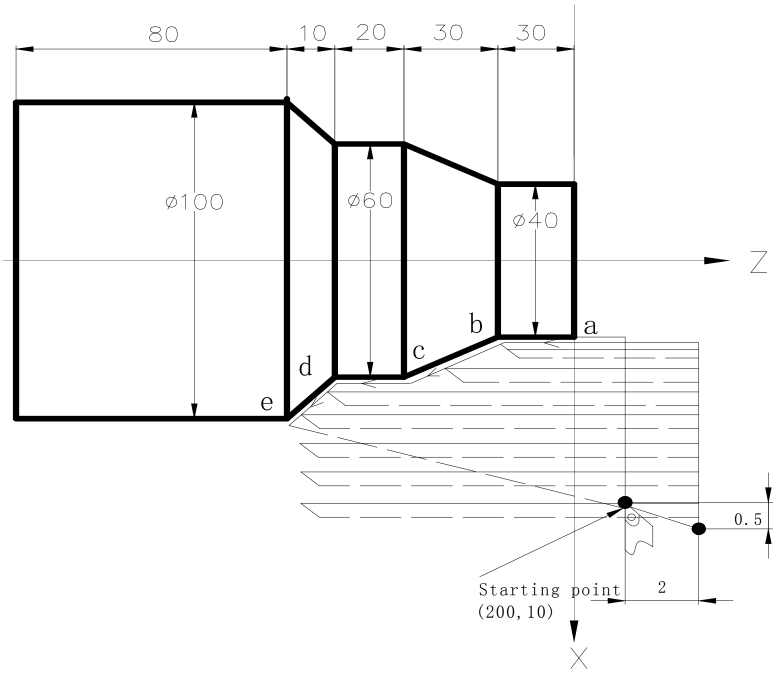


Fig.3-22

Program：O0004；

G00 X200 Z10 M3 S800； （Spindle clockwise with 800 rev/min）

G71 U2 R1 F200； （Cutting depth each time 4mm，tool retraction [in diameter]）

G71 P80 Q120 U0.5 W0.2； （roughing a---e，machining allowance 0.5mm in X direction, 0.2mm in Z direction）

N80 G00 X40 S1200;　　（Positioning）
G01 Z-30 F100 ;　　（a→b）
X60 W-30;　　（b→c）　　a→b→c→d→e blocks for finishing path
W-20;　　（c→d）
N120 X100 W-10;　　（d→e）
G70 P80 Q120;　　（a---e blocks for finishing path）
M30;　　（End of block）

3.9.2 RADIAL ROUGHTING CYCLE G72

Instruction format： G72 W（Δd） R（e） F__ S__ T__;　　(1)

G72 P（ns） Q（nf） U（Δu） W（Δw）;　　(2)

N__（ns）. . . . . . ;
. . . . ;
. . . . F;
. . . . S;
. . . . ;　　(3)
.
N__（nf）. . . . . . ;

Instruction function： G72 is divided into three parts:

(1) 1st blocks for defining the travels of tool infeed and tool retraction, the cutting speed, the spindle speed and the tool function when roughing;

(2) 2nd blocks for defining the block interval, finishing allowance;

(3) 3rd blocks for some continuous finishing path, counting the roughing path without being executed actually when executing G72.

According to the finishing path, the finishing allowance, the path of tool infeed and retract tool, the system automatically counts the path of roughing, the tool cuts the workpiece in paralleling with Z axis, and the roughing is completed by multiple executing the cutting cycle tool infeed→cutting feed→tool retraction. The starting point and the end point of G72 are the same one. The instruction is applied to the formed roughing of non-formed rod.

Relevant definitions:

Finishing path: the above-mentioned Part(3) of G71(ns～nf block)defines the finishing path, and the starting point of finishing path (i.e. starting point of ns block)is the same these of starting point and end point of G72, called A point; the first block of finishing path(ns block)is used for Z rapid traversing or cutting feed, and the end point of finishing path is called to B point; the end point of finishing path(end point of nf block)is called to C point. The finishing path is A→B→C.

Roughing path: The finishing path is the one after offsetting the finishing allowance（Δu,Δw）and is the path contour formed by executing G72. A, B, C point of finishing path after offset corresponds separately to A’,B’,C’point of roughing path, and the final continuous cutting path of G72 is B’→C’point.

Δd: it is each travel(unit: mm,)of tool infeed without in Z direction when roughing, and the direction of tool infeed is defined by move direction of ns block. The instruction value Δd is reserved after executing U（Δd） and the value of system parameter NO.051 is rewritten to Δd×1000(unit: 0.001 mm). The value of system parameter NO.051 is regarded as the travel of tool infeed when U（Δd） is not input.

e: it is each travel（unit: mm） of tool infeed without in Z direction when roughing, and the direction of tool retraction is opposite to that of tool infeed, the instruction value e is reserved and the value of system parameter NO.052 is rewritten to e×1000(unit: 0.001 mm)after R(e)is executed. The value of system parameter NO.052 is regarded as the travel of tool retraction when R（e） is not input.

ns: Block number of the first block of finishing path.

nf: Block number of the last block of finishing path.

Δu: finishing allowance in X direction when roughing(the coordinates offset in X direction of roughing path compared to finishing path, i.e. the different value of absolute coordinates in X direction between A’and A, unit: mm).

Δw: finishing allowance in Z direction when roughing(the coordinates offset in Z direction of roughing path compared to finishing path, i.e. the different value of absolute coordinates in X direction between A’and A, unit: mm).

F: Cutting feedrate; S: Spindle speed; T: Tool number, tool offset number.

M, S, T, F: They can be specified in the first G72 or the second ones or program ns～nf. M, S, T, F functions of M, S, T, F blocks are invalid in G72, and they are valid in G70 finishing blocks.

Execution process ： Fig. 3-23

① Rapid traverse to A’from A point, the travel in X direction is Δu, and the travel in Z direction is Δw;

② The travel in Z direction from A’is Δd(tool infeed), ns block is for tool infeed at rapid traverse speed with G0, is for tool infeed at G72feedrate F in G1, and its direction of tool infeed is that of A→B point;

③ Cutting feeds to the roughing path in X direction, and its direction is the same that of coordinates in X direction B→C point;

④ The travel of tool retraction is e (45° straight line)at feedrate in X, Z direction, the directions of tool retraction is opposite to that of tool infeed ;

⑤ Rapidly retract at rapid traverse speed in X direction to the position which is the same that of the coordinates in Z direction ;

⑥ After executing the tool infeed (Δd+e)again in Z direction, the end point of traversing tool is still on the middle point of straight line between A’and B’(the tool does not reach or exceed B’), and after executing the tool infeed (Δd+e)again, execute ③; after executing the tool infeed (Δd+e)again, the end point of tool traversing reaches B’point or exceeds the straight line between A’→B’point and execute the tool infeed to B’point in Z direction and the execute the next step;

⑦ Cutting feed from B’to C’point along the roughing path;

⑧ Rapidly traverse to A from C'point and the program jumps to the next clock following nf block after G71 cycle is completed.

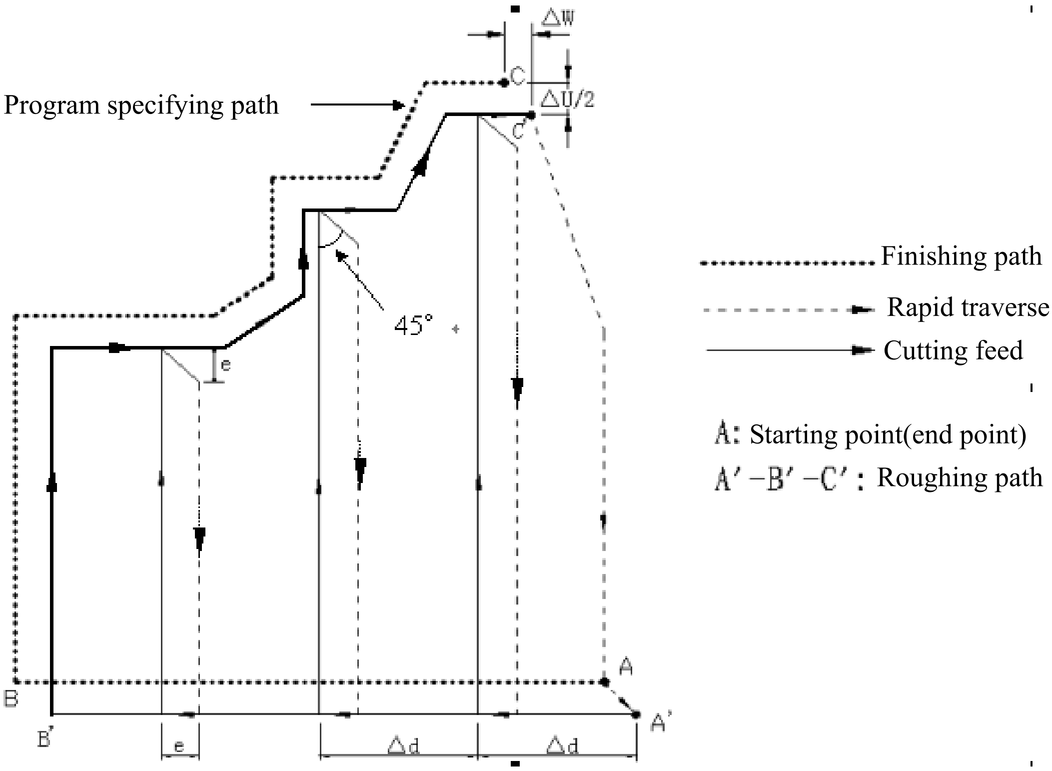


Fig. 3-23

Instruction specifications:

- ns～nf blocks in programming must be followed G72 blocks. If they are in the front of G72 blocks, the system automatically searches and executes ns～nf blocks, and then executes the next program following nf block after they are executed, which causes the system executes ns～nf blocks repetitively.
- ns～nf blocks are used for counting the roughing path and the blocks are not executed when G72 is executed. F, S, T instructions of ns～nf blocks are invalid when G72 is executed, at the moment, F, S, T instructions of G72 blocks are valid. F, S, T of ns～nf blocks are valid when executing ns～nf to command G70 finishing cycle;
- There are G00,G01 without the word X(U) in ns block, otherwise the system alarms.
- The dimensions in X, Z direction must be changed monotonously (always increasing or reducing) for the finishing path;
- In ns～nf blocks, there are only G instructions: G01, G02, G03, G04, G96, G97, G98, G99, G40, G41,G42 and the system cannot call subprograms(M98/M99);
- G96, G97, G98, G99, G40, G41, G42 are invalid in G71 and valid in G70;G96,G97,G98,
- When G72 is executed, the system can stop the automatic run and manual traverse, but return to the position before manual traversing when G72 is executed again, otherwise, the following path will be wrong;
- When the system is executing the feed hold or single block, the program pauses after the system has executed end point of current path;
- △d，△u are specified by the same U and different with or without being specified P,Q instructions;
- There are no the same block number in ns~nf when compound cycle instructions are executed repetitively in one program;

● G72 cannot be executed in MDI, otherwise, the system alarms.

Coordinate offset direction with finishing allowance:

Coordinate offset direction with finishing allowance:

Δu,Δw define the coordinates offset and its direction of finishing, and their sign symbols are as follows Fig. 3-24: B→C for finishing path, B'→C'for roughing path and A is the starting point.

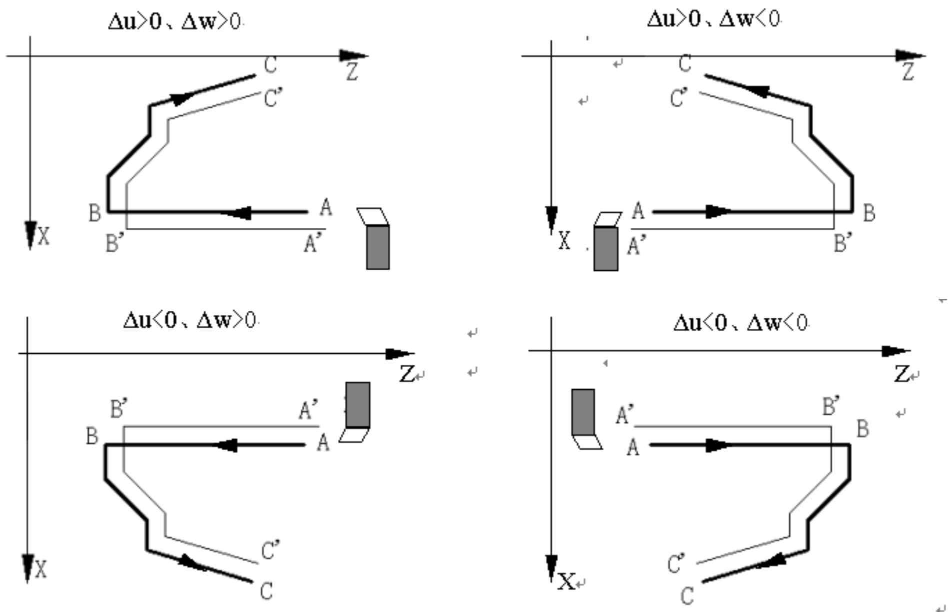


图 3-24

Example: Fig.3-25

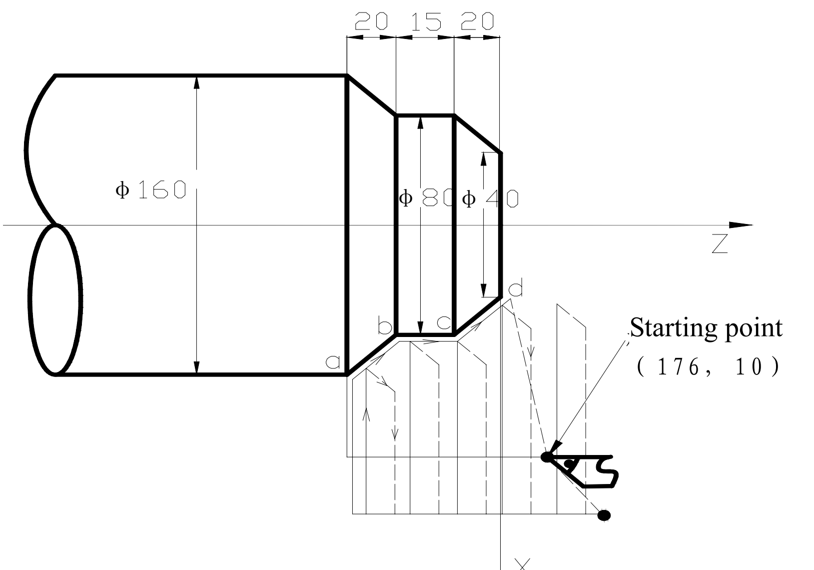


Fig. 3-25

Program：O0005；

G00 X176 Z10 M03 S500	(Change No.2 tool and execute its compensation, spindle rotation with 500 rev/min)
G72 W2.0 R0.5 F300；	(Tool infeed 2mm, tool retraction 2mm)
G72 P10 Q20 U0.2 W0.1；	(Roughing a--d，roughing allowance 0.2mm in X direction and 0.1mm in Z direction)

N10 G00 Z-55 S800 ; (Rapid traverse)
G01 X160 F120; (Infeed to a point)
X80 W20; (Machining a—b)
W15; (Machining b—c)
N20 X40 W20 ; (Machining c—d)
Blocks for finishing path
G70 P050 Q090 M30; (Finishing a—d)

3.9.3 CLOSED CUTTING CYCLE G73

Instruction format: G73 U(Δi) W (Δk) R (d) F__ S__ T__; (1)
G73 P(ns) Q(nf) U(Δu) W(Δw); (2)
N__ (ns) . . . . . ;
. . . . ;
. . . . F;
. . . . S;
. . . . ; (3)
.
N__ (nf) . . . . . ;

Instruction functions: G73 is divided into three parts:

(1) blocks for defining the travels of tool infeed and tool retraction, the cutting speed, the spindle speed and the tool function when roughing;

(2) blocks for defining the block interval, finishing allowance;

(3) blocks for some continuous finishing path, counting the roughing path without being executed actually when executing G73.

According to the finishing allowance, the travel of tool retraction and the cutting times, the system automatically counts the travel of roughing offset, the travel of each tool infeed and the path of roughing, the path of each cutting is the offset travel of finishing path, the cutting path approaches gradually the finishing one, and last cutting path is the finishing one according to the finishing allowance. The starting point and end point of G73 are the same one, and G73 is applied to roughing for the formed rod. G73 is non-modal and its path is as Fig.3-26.

Relevant definitions:

Finishing path: the above-mentioned Part 3 of G73(ns～nf block)defines the finishing path, and the starting point of finishing path (start point of ns block)is the same these of starting point and end point of G73, called A point; the end point of the first block of finishing path(ns block)is called B point; the end point of finishing path(end point of nf block)is called C point. The finishing path is A→B→C.

Roughing path: It is one group of offset path of finishing one, and the roughing path times are the same that of cutting. After the coordinates offset, A,B,C of finishing path separately corresponds to A_n,B_n,C_n of roughing path(n is the cutting times, the first cutting path is A_1,B_1,C_1 and the last one is A_d,B_d,C_d). The coordinates offset value of the first cutting compared to finishing path is （Δi×2+Δu，Δw+Δk）（diameter programming）, the coordinates offset value of the last cutting compared to finishing path is（Δu，Δw）, the coordinates offset value of each cutting compared to the previous one is as follows:

$$(-\frac{\Delta i\times 2}{1000\times d-1},-\frac{\Delta k}{1000\times d-1})$$

Δi：Travel of tool retraction for roughing is -9999.999~9999.999（unit：mm，radius value with sign symbol）in X direction , Δi is equal to the coordinates offset value （radius value）of A_1 point in X direction compared to A_d point. The total cutting travel(radius value)in X direction is equal to |Δi|when roughing, and the cutting direction in X direction is opposite to the sign of Δi: Δi>0, cut in X negative direction when roughing. It is reserved after Δi instruction value is executed and the value of system parameter NO.053 is rewritten to Δi×1000（unit：0.001 mm）. The value of system parameter NO.053 is regarded as the travel of tool retraction of roughing in X direction when U（Δi）is not input.

Δk：It is travel of tool retraction for roughing -9999.999~9999.999 （unit：mm，radius value with sign symbol）in Z direction , Δk is equal to the coordinates offset value （radius value）of A_1 point in Z direction compared to A_d point. The total cutting travel(radius value)in Z direction is equal to |Δk|when roughing, and the cutting direction in Z direction is opposite to the sign of Δk: Δk>0, cut in Z negative direction when roughing. It is reserved after Δk instruction value is executed and the value of system parameter NO.054 is rewritten to Δk×1000（unit：0.001 mm）. The value of system parameter NO.054 is regarded as the travel of tool retraction of roughing in Z direction when W（Δk）is not input.

d: It is the cutting times 1~9999 (unit: times). R5 means the closed cutting cycle is completed by 5 times cutting. R （d）is reserved after it is executed and the value of system parameter NO.055 is rewritten to d (unit: times). The value of system parameter NO.055 is regarded as the cutting times when R （d）is not input.

ns: Block number of the first block of finishing path.

nf: Block number of the last block of finishing path.

Δu：It is the finishing allowance -99.999~99.999 (unit: mm, diameter value with sign symbol)in X direction and is the coordinates offset in X direction of roughing contour compared to finishing path, i.e. the different value of absolute coordinates of A_1 compared to A in X direction. Δu>0，it is the offset of the last roughing path compared to finishing path in X positive direction The system defaults Δu=0 when U（Δu） is not input, i.e. there is no finishing allowance in X direction for roughing cycle.

Δw：It is the finishing allowance -99.999~99.999 (unit: mm)in Z direction and is the coordinates offset in Z direction of roughing contour compared to finishing path, i.e. the different value of absolute coordinates of A_1 compared to A in Z direction. Δw＞0，it is the offset of the last roughing path compared to finishing path in Z positive direction The system defaults Δw=0 when U（Δw） is not input, i.e. there is no finishing allowance in Z direction for roughing cycle.

F: Feedrate; S: Spindle speed; T: Tool number, tool offset number.

M, S, T, F: They can be specified in the first G73 or the second ones or program ns～nf. M, S, T, F functions of M, S, T, F blocks are invalid in G73, and they are valid in G70 finishing blocks.

Execution process:(Fig. 3-26)

① $A \rightarrow A_1$：Rapid traverse；

② First roughing $A_1 \rightarrow B_1 \rightarrow C_1$ ：

$A_1 \rightarrow B_1$：Rapid traverse speed in ns block in G0, cutting feedrate specified by G73 in ns block in G1；

$B_1 \rightarrow C_1$：Cutting feed.

③ $C_1 \rightarrow A_2$：Rapid traverse；

④ Second roughing $A_2 \rightarrow B_2 \rightarrow C_2$ ：

$A_2 \rightarrow B_2$：Rapid traverse speed in ns block in G0, cutting feedrate specified by G73 in ns block in G1;

$B_2 \rightarrow C_2$：Cutting feed.

⑤ $C_2 \rightarrow A_3$：rapid traverse；

…………

No. n times roughing，$A_n \rightarrow B_n \rightarrow C_n$ ：

$A_n \rightarrow B_n$：ns Rapid traverse speed in ns block in G0, cutting feedrate specified by G73 in ns block in G1;

$B_n \rightarrow C_n$：Cutting feed.

$C_n \rightarrow A_{n+1}$：Rapid traverse;

…………

Last roughing，$A_d \rightarrow B_d \rightarrow C_d$ ：

$A_d \rightarrow B_d$: Rapid traverse speed in ns block in G0, cutting feedrate specified by G73 in ns block in G1;

$B_d \rightarrow C_d$：Cutting fee.

$C_d \rightarrow A$：Rapid traverse to starting point；

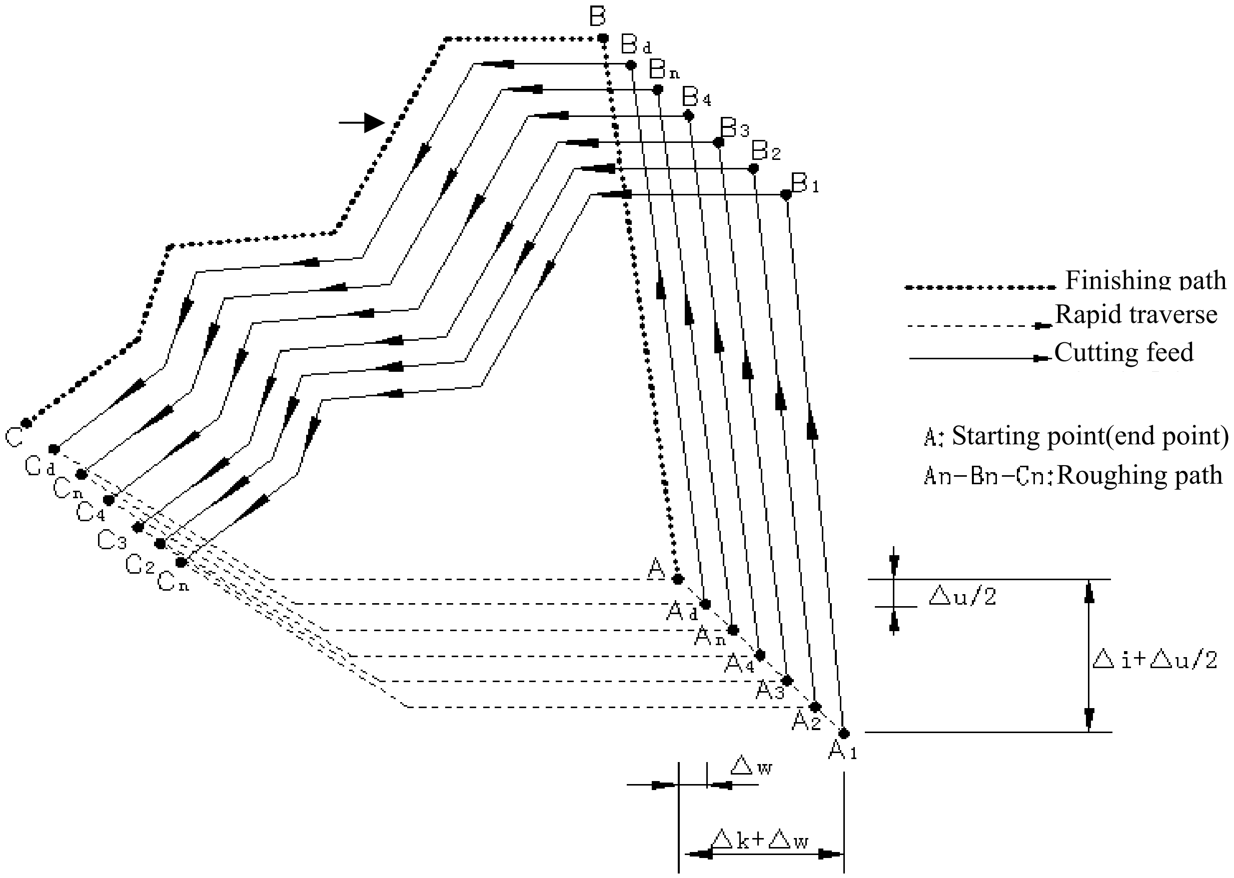


Fig. 3-26 G73 path

Instruction specifications：

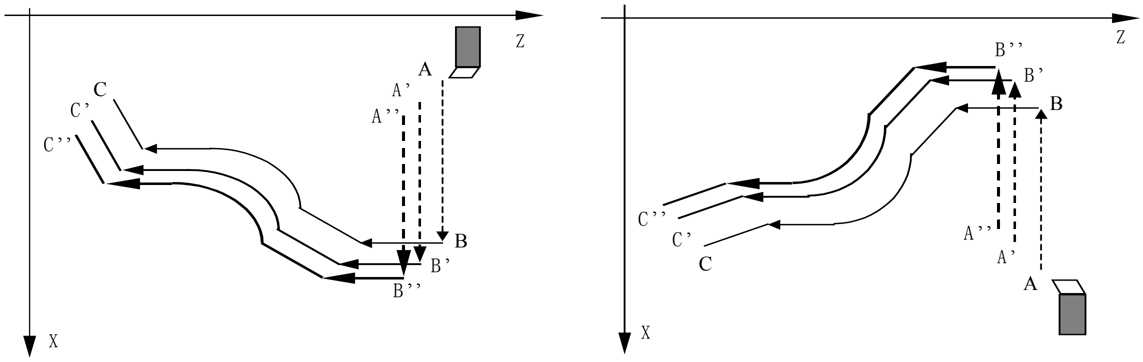
- ns～nf blocks in programming must be followed G73 blocks. If they are in the front of G73 blocks, the system automatically searches and executes ns～nf blocks, and then executes the next program following nf block after they are executed, which causes the system executes ns～nf blocks repetitively.
- ns～nf blocks are used for counting the roughing path and the blocks are not executed when G73 is executed. F, S, T instructions of ns～nf blocks are invalid when G71 is executed, at the moment, F, S, T instructions of G73 blocks are valid. F, S, T of ns～nf blocks are valid when executing ns～nf to command G70 finishing cycle.
- There are only G00, G01, G02, G03 in ns block.
- In ns～nf blocks, there are only G instructions:G00, G01, G02, G03, G04, G96, G97, G98, G99, G40, G41,G42 and the system cannot call subprograms(M98/M99)
- G96, G97, G98, G99, G40, G41, G42 are invalid in G73 and valid in G70.
- When G73 is executed, the system can stop the automatic run and manual traverse, but return to the position before manual traversing when G73 is executed again, otherwise, the following path will be wrong.
- When the system is executing the feed hold or single block, the program pauses after the system has executed end point of current path.
- △i，△u are specified by the same U and Δk，Δw are specified by the same U, and they are different with or without being specified P,Q instructions.
- G73 cannot be executed in MDI, otherwise, the system alarms.
- There are no the same block number in ns~nf when compound cycle instructions are executed repetitively in one program.

Coordinate offset direction with finishing allowance:

Δi,Δk define the coordinates offset and its direction of roughing;

Δu,Δw define the coordinates offset and its direction of finishing, and their sign symbols are as follows Fig. 3-27: B→C for workpiece contour, B'→C'for roughing contour and B''→C'' for finishing path.

1) Δi<0 Δk>0, △u<0 Δw>0; 2) Δi>0 Δk>0, △u>0 Δw>0;



3) Δi<0 Δk<0, △u<0 Δw<0; 4) Δi>0 Δk<0, △u>0 Δw<0;

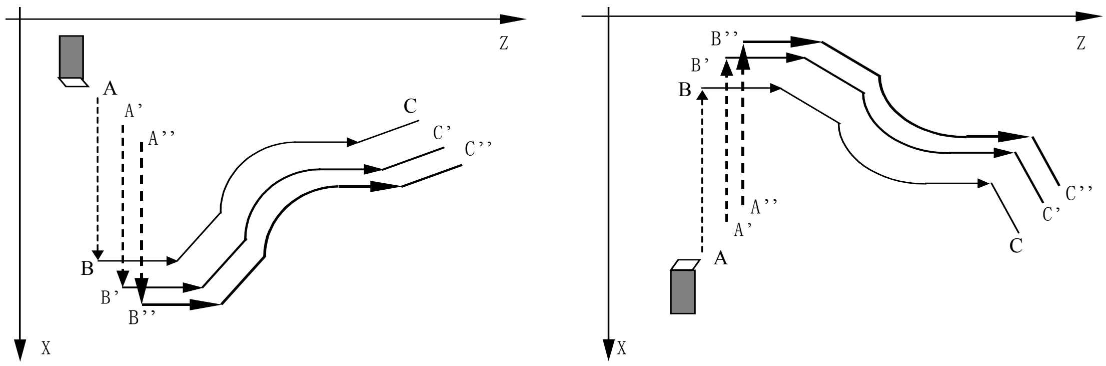


Fig.3-27

Example: Fig.3-28

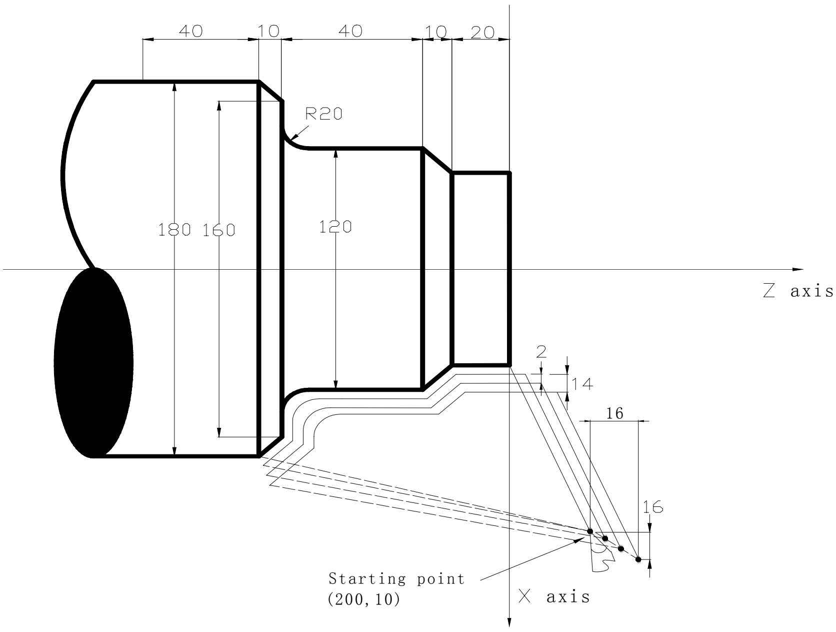


Fig. 3-28

Program：O0006；

G99 G00 X200 Z10 M03 S500;	（Specify feedrate per rev and position starting point and start spindle）
G73 U1.0 W1.0 R3 ;	（Tool retraction with 2mm in X direction, 1mm in Z direction）
G73 P14 Q19 U0.5 W0.3 F0.3 ;	（Roughing with 0.5 allowance in X direction and 0.mm in Z direction）
N14 G00 X80 W-40 ;	Blocks for finishing
G01 W-20 F0.15 S600 ;	
X120 W-10 ;	
W-20 ;	
G02 X160 W-20 R20 ;	
N19 G01 X180 W-10 ;	
G70 P14 Q19 M30;	（Finishing）

3.9.4　FINISHING CYCLE G70

Instruction format：G70 P（ns） Q（nf）；

Instruction function：The tool executes the finishing of workpiece from starting point along with the finishing path defined by ns～nf blocks. After executing G71, G72 or G73 to roughing, execute G70 to finishing and single cutting of finishing allowance is completed. The tool returns to starting point and execute the next block following G70 block after G70 cycle is completed.

ns：Block number of the first block of finishing path

nf: Block number of the last block of finishing path.

G70 path is defined by programmed one of ns～nf blocks. Relationships of relative position of ns,nf block in G70～G73 blocks are as follows:

```
．．．．．．．．
G71/G72/G73 ……；
N__（ns）．．．．．．   ┐
．．．．．．．．          │
      • F                 │
      • S                 ├ Blocks for finishing path
      •                   │
      •                   │
N__（nf）……             ┘
     ．．．
G70 P（ns） Q（nf）；
     ．．．
```

Instruction specifications:

● ns～nf blocks in programming must be followed G70 blocks. If they are in the front of G71 blocks, the system automatically searches and executes ns～nf blocks, and then executes the next program following nf block after they are executed, which causes the system executes ns～nf blocks repetitively.

- F, S, T in ns～nf blocks are valid when executing ns～nf to command G70 finishing cycle.
- G96, G97, G98, G99, G40, G41, G42 are valid in G70;
- When G70 is executed, the system can stop the automatic run and manual traverse, but return to the position before manual traversing when G70 is executed again, otherwise, the following path will be wrong.
- When the system is executing the feed hold or single block, the program pauses after the system has executed end point of current path.
- G70 cannot be executed in MDI, otherwise, the system alarms.
- There are no the same block number in ns~nf when compound cycle instructions are executed repetitively in one program.

3.9.5 AXIAL GROOVING MULTIPLE CYCLE G74

Instruction format：G74 R（e）；

G74 X（U）__ Z（W）__ P（Δi） Q（Δk） R（Δd） F__；

Instruction function: Axial (X) tool infeed cycle compounds radial discontinuous cutting cycle: Tool infeeds from starting point in radial direction(Z), retracts, infeeds again, and again and again, and last tool retracts in axial direction, and retracts to position in Z direction in radial direction, which is called one radial cutting cycle; tool infeeds in axial direction and execute the next radial cutting cycle; cut to end point of cutting, and then return to starting point (starting point and end point are the same one in G74), which is called one radial grooving compound cycle. Directions of axial tool infeed and radial tool infeed are defined by relative position between end point X（U）Z（W） and starting point of cutting. G75 is used for machining radial loop groove or column surface by radial discontinuously cutting, breaking stock and stock removal.

Relevant definitions：

Starting point of axial cutting cycle: starting position of axial tool infeed for each axial cutting cycle, defining with A_n(n=1,2,3……), the coordinates of A_n in Z direction is the same that of starting point A, the different value of coordinates between A_n and A_{n-1} in X direction is Δi. The starting point A_1 of the first axial cutting cycle is the same as the starting point A, and the starting point (A_f) of the last axial cutting cycle in X direction is the same that of cutting end point.

End point of axial tool infeed: starting position of axial tool infeed for each axial cutting cycle, defining with B_n(n=1,2,3……), the coordinates of B_n in Z direction is the same that of cutting end point, the coordinates of B_n in X direction is the same that of A_n , and the end point (B_f) of the last axial tool infeed is the same that of cutting end point.

End point of radius tool retraction: end position of radius tool infeed(travel of tool infeed is Δd) after each axial cutting cycle reaches the end point of axial tool infeed, defining with

C_n(n=1,2,3……), the coordinates of C_n in Z direction is the same that of cutting end point, and the different value of coordinates between C_n and A_n in X direction is Δd;

End point of axial cutting cycle: end position of axial tool retraction from the end point of radius tool retraction, defining with D_n(n=1,2,3……), the coordinates of D_n in Z direction is the same that of starting point, the coordinates of D_n in X direction is the same that of C_n (the different value of coordinates in X direction between it and A_n is Δd);

Cutting end point: it is defined by X（U）__ Z（W）___ ,and is defined with B_f of the last axial tool infeed.

R（e）: it is the travel（0~99.999，unit:mm） of tool retraction after each axial(Z axis) tool infeed without signs. The instruction value is reserved after executing R（e）and the value of system parameter NO.056 is rewritten to e×1000（unit：0.001 mm）. The value of system parameter NO.056 is regarded as the travel of tool retraction when R（e）is not input.

X: Absolute coordinate value of cutting end point B_f in X direction (unit:mm)

U: Different value of absolute coordinates in X direction between cutting end point B_f and starting point.

Z: Absolute coordinate value of cutting end point B_f in Z direction (unit:mm).

W: Different value of absolute coordinates in Z direction between cutting end point B_f and starting point.

P（Δi）：travel of radial(X axis) cutting for each axial cutting cycle without signs.

Q（Δi）：travel of discontinuous tool infeed in Z direction without signs when axial(Z axis) cutting.

R（Δd）: travel (unit: mm, radius value)of radial (X axis) tool retraction after cutting to end point of axial cutting. The radial tool retraction is 0 when the system defaults the axial cutting end point. The system defaults the tool retraction is executed in positive direction when X（U）and P（Δi） are omitted.

Execution process:(Fig. 3-29)

① Axial (Z axis) cutting feed △k from the starting point of axial cutting cycle, feed in Z negative direction when the coordinates of cutting end point is less than that of starting point in Z direction, otherwise, feed in Z positive direction;

②Axial(Z axis) rapid tool retraction e and its direction is opposite to the feed direction of ①;

③ Cutting feed(Δk+e) again in Z direction, the end point of cutting feed is still in it between starting point A_n of axial cutting cycle and end point of axial tool infeed, cutting feed (Δk+e)again in Z direction and execute ②; after cutting feed (Δk+e)again in Z direction, the end point of cutting feed is on B_n or is not on it between A_n and B_n cutting feed to B_n in Z direction and then execute ④;

④ Radial(X axis) rapid tool retraction △d（radius value）to C_n , when the coordinates of B_f (cutting end point) is less than that of A (starting point) in X direction, retract tool in X positive, otherwise, retract tool in X negative direction;

⑤ Axial(Z axial) rapid retract tool to Dn, No. n axial cutting cycle is completed. If the current axial cutting cycle is not the last one, execute ⑥ ; if it is the previous one before the last axial cutting cycle, execute ⑦;

⑥ Radial(X axial)rapid tool infeed, and it direction is opposite to ④ retract tool. If the end point of tool infeed is still on it between A and A_f (starting point of last axial cutting cycle) after tool infeed

(△d+△i) (radius value) in X direction, i.e. Dn→A_{n+1} and then execute ① (start the next axial cutting cycle); if the end point of tool infeed is not on it between D_n and A_f after tool infeed (△d+△i) (radius value) in X direction, rapidly traverse to A_f and execute ① to start the first axial cutting cycle;

⑦ Rapidly traverse to return to A in X direction, and G74 is completed.

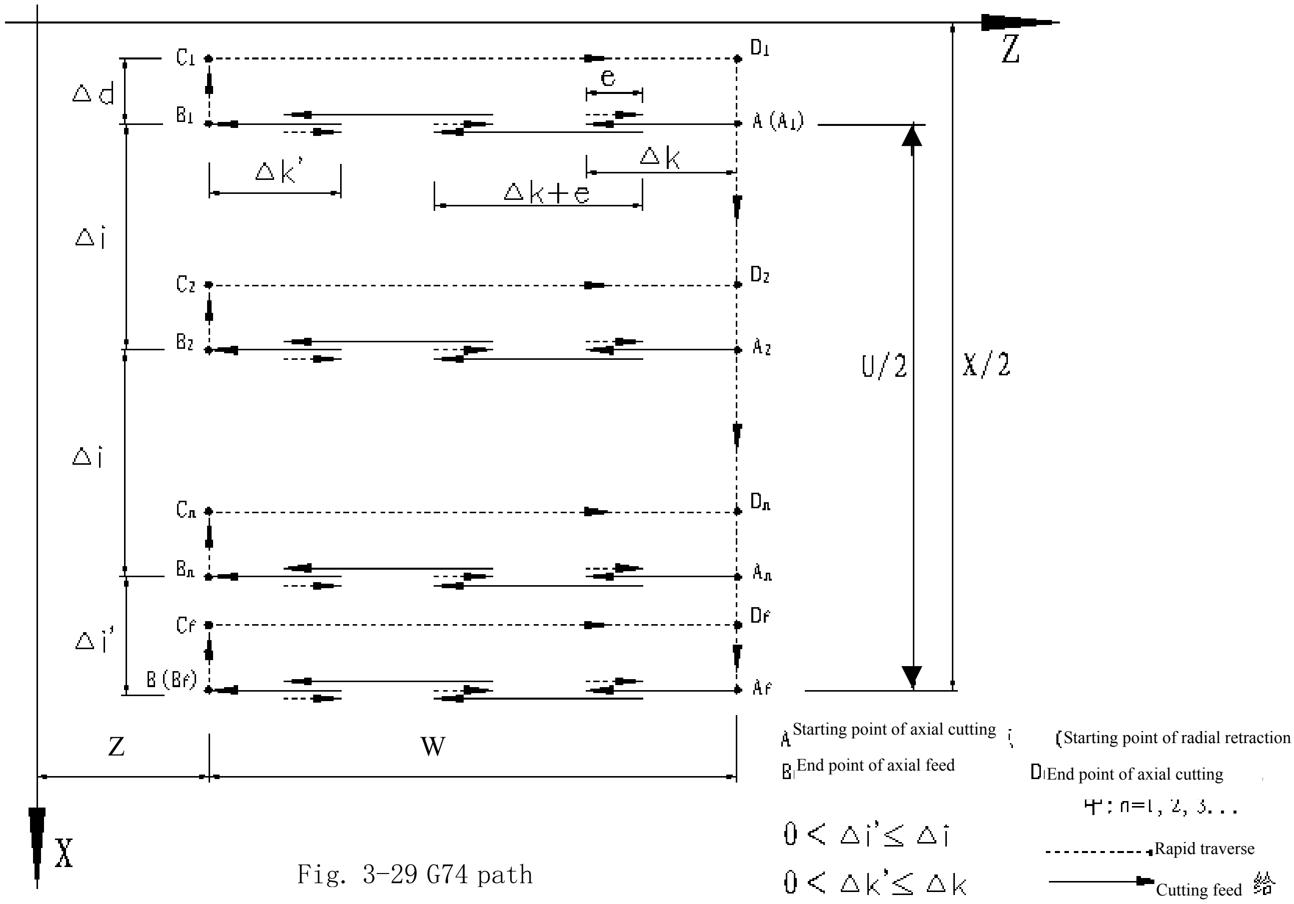


Fig. 3-29 G74 path

Instruction specifications:

- The cycle movement is executed by Z（W）and P（Δk）blocks of G74, and the movement is not executed if only "G74 R（e）;" block is executed;
- Δd and e are specified by the same address and whether there are Z（W）and P（Δk）word or not in blocks to distinguish them;
- The tool can stop in Auto mode and traverse in Manual mode when G74 is executed, but the tool must return to the position before executing in Manual mode when G74 is executed in G74 again, otherwise the following path will be wrong.
- When the single block is running, programs dwell after each axial cutting cycle is completed.
- R（Δd）must be omitted in blind hole cutting .and so there is no distance of tool retraction when the tool cuts to axial end point of cutting.

Example:

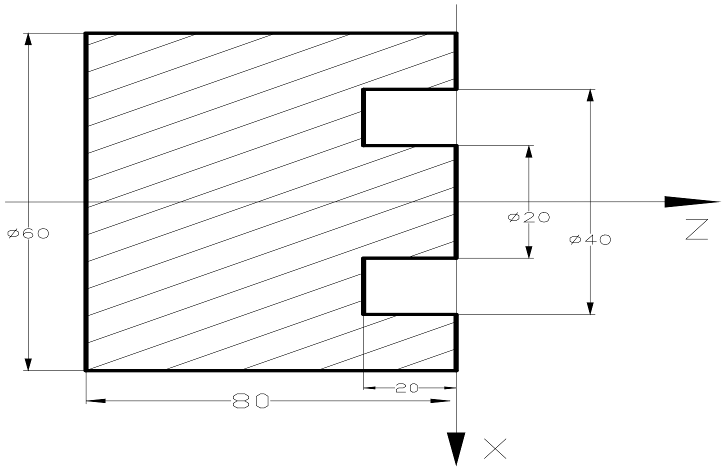


Fig. 3-30

Program：O0007；

G0 X40 Z5 M3 S500；　　(Start spindle and position to starting point of machining)

G74 R0.5 ；　　(Machining cycle)

G74 X20 Z60 P3000 Q5000 F50；　　(Tool infeed 5mm in Z axis and tool retraction 0.5mm each time; rapid return to starting point(Z5) after cutting feed to end point(Z60), tool infeed 3mm in X direction and cycle the above-mentioned steps)

M30；　　(End of program)

3.9.6 RADIAL GROOVING MULTIPLE CYCLE G75

Instruction format：G75 R（e）；

G75 X（U）__ Z（W）__ P（Δi） Q（Δk） R（Δd） F__；

Instruction function: Axial (Z axis) tool infeed cycle compounds radial discontinuous cutting cycle: Tool infeeds from starting point in radial direction, retracts, infeeds again, and again and again, and last tool retracts in axial direction, and retracts to position in X direction in radial direction, which is called one radial cutting cycle; tool infeeds in axial direction and execute the next radial cutting cycle; cut to end point of cutting, and then return to starting point (starting point and end point are the same one in G75), which is called one radial grooving compound cycle. Directions of axial tool infeed and radial tool infeed are defined by relative position between end point X（U）Z（W） and starting point of cutting. G75 is used for machining radial loop groove or column surface by radial discontinuously cutting, breaking stock and stock removal.

Relevant definitions：

Starting point of radial cutting cycle: starting position of axial tool infeed for each radial cutting cycle, defined by A_n(n=1,2,3……), the coordinates of A_n in X direction is the same that of starting point A, the different value of coordinates between A_n and A_{n-1} in X direction is Δk. The starting point A_1 of the first radial cutting cycle is the same as the starting point A, and the starting point (A_f) of the last axial cutting cycle in Z direction is the same that of cutting end point.

End point of radial tool infeed: starting position of radial tool infeed for each radial cutting cycle, defined by B_n(n=1,2,3……), the coordinates of B_n in X direction is the same that of cutting end point, the coordinates of B_n in Z direction is the same that of A_n, and the end point (B_f) of the last radial tool infeed is the same that of cutting end point.

End point of axial tool retraction: end position of axial tool infeed(travel of tool infeed is Δd) after each axial cutting cycle reaches the end point of axial tool infeed, defining with C_n(n=1,2,3……), the coordinates of C_n in X direction is the same that of cutting end point, and the different value of coordinates between C_n and A_n in Z direction is Δd;

End point of radial cutting cycle: end position of radial tool retraction from the end point of axial tool retraction, defined by D_n(n=1,2,3……), the coordinates of D_n in X direction is the same that of starting point, the coordinates of D_n in Z direction is the same that of C_n (the different value of coordinates in Z direction between it and A_n is Δd);

Cutting end point: it is defined by X（U）__ Z（W）___ ,and is defined with B_f of the last radial tool infeed.

R（e）: it is the travel(unit: mm) of tool retraction after each radial(X axis) tool infeed without signs. The instruction value is reserved after executing R（e）and the value of system parameter NO.056 is rewritten to e×1000（unit：0.001 mm）. The value of system parameter NO.056 is regarded as the travel of tool retraction when R（e）is not input.

X: Absolute coordinate value of cutting end point B_f in X direction (unit:mm)

U: Different value of absolute coordinates in X direction between cutting end point B_f and starting point.

Z: Absolute coordinate value of cutting end point B_f in Z direction (unit:mm).

W: Different value of absolute coordinates in Z direction between cutting end point B_f and starting point.

P（Δi）：travel(0~9999999) of radial(X axis) discontinuous tool infeed for each axial cutting cycle without signs.

Q（Δk）：travel of discontinuous tool infeed in Z direction without signs when axial(Z axis) cutting.

R（Δd）：travel (unit: mm, radius value)of axial (Z axis) tool retraction after cutting to end point of radial cutting. The system defaults the tool retraction is executed in positive direction when Z（W）and Q（Δk） are omitted.

Travel of axial(Z axis) tool retraction is 0 after the system defaults radial cutting end point when R（Δd） is omitted.

The system defaults the tool retraction is executed in positive direction when Z（W）and Q（Δk） are omitted.

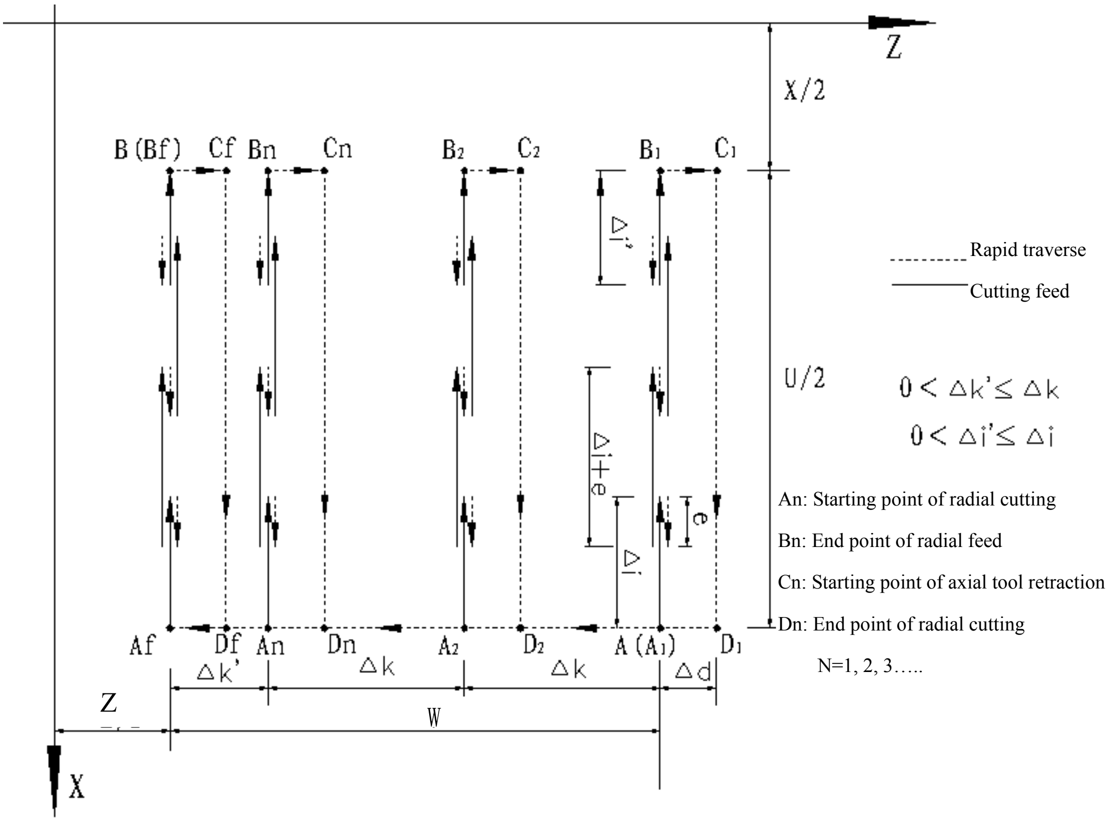


Fig. 3-31 G75 path

Execution process:(Fig. 3-31)

① Radial (X axis) cutting feed △i from the starting point of radial cutting cycle, feed in X negative direction when the coordinates of cutting end point is less than that of starting point in X direction, otherwise, feed in X positive direction;

② Radial(X axis) rapid tool retraction e and its direction is opposite to the feed direction of ①;

③ Cutting feed(Δk+e) again in X direction, the end point of cutting feed is still in it between starting point A_n of radial cutting cycle and end point of radial tool infeed, cutting feed (Δi+e)again in X direction and execute ②; after cutting feed (Δi+e)again in X direction, the end point of cutting feed is on B_n or is not on it between A_n and B_n cutting feed to B_n in X direction and then execute④;

④ Axial(Z axis) rapid tool retraction △d（radius value）to C_n , when the coordinates of B_f (cutting end point) is less than that of A (starting point) in Z direction, retract tool in Z positive, otherwise, retract tool in Z negative direction;

⑤ Radial(Z axis) rapid retract tool to Dn, No. n radial cutting cycle is completed. The current radial cutting cycle is not the last one, execute ⑥ ; if it is the previous one before the last radial cutting cycle, execute ⑦;

⑥ Axial(X axis)rapid tool infeed, and it direction is opposite to ④ retract tool. If the end point of tool infeed is still on it between A and A_f (starting point of last radial cutting cycle) after tool infeed (△d+△k) (radius value) in Z direction, i.e. Dn→A_{n+1} and then execute ① (start the next radial cutting cycle); if the end point of tool infeed is not on it between D_n and A_f after tool infeed (△d+△k) in Z direction, rapidly traverse to A_f and execute ① to start the first radial cutting cycle;

⑦ Rapidly traverse to return to A in Z direction, and G75 is completed.

Explanation：

- The cycle movement is executed by X（W）and P（Δi）blocks of G75, and the movement is not executed if only “G75 R（e)；” block is executed;
- Δd and e are specified by the same address R and whether there are X（U）and P（Δi）words or not in blocks to distinguish them;
- The tool can stop in Auto mode and traverse in Manual mode when G75 is executed, but the tool must return to the position before executing in Manual mode when G75 is executed again, otherwise the following path will be wrong;
- When the system is executing the feed hold or single block, the program pauses after the system has executed end point of current path;
- R（Δd）must be omitted in grooving, and so there is no travel of tool retraction when the tool cuts to radial cutting end point.

Example: Fig. 3-32

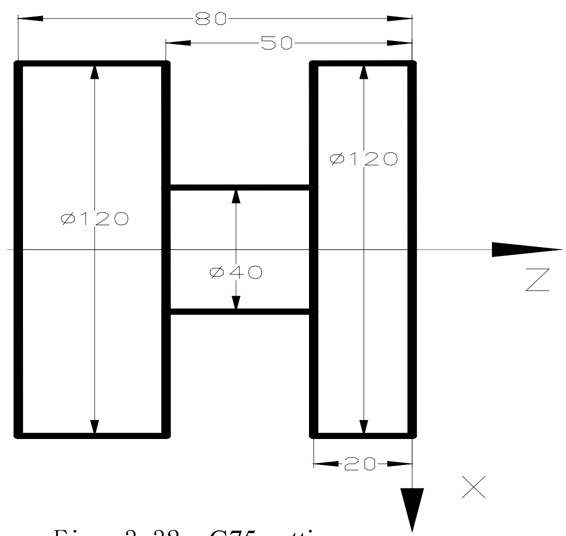


Fig. 3-32 G75 cutting

Program：O0008；	
G00 X150 Z50 M3 S500；	（Start spindle with 500 rev/min）
G0 X125 Z-20；	（Position to starting point of machining）
G75 R0.5 F150；	（Machining cycle）
G75 X40 Z-50 P6000 Q3000；	（Tool infeed 6mm every time in X direction, tool retraction 0.5mm, rapid returning to starting point (X125) after infeeding to end point (X40), tool infeed 3mm in Z direction and cycle the above-mentioned steps to continuously run programs）
G0 X150 Z50；	（Return to starting point of machining）
M30；	（End of program）

3.10 THREAD CUTTING

GSK980TD CNC system can machine many kinds of thread cutting, including metric/inch single, multi threads, thread with variable lead and tapping cycle. Length and angle of thread run-out can be changed, multiple cycle thread is machined by single sided to protect tool and improve smooth finish of its surface. Thread cutting includes: continuous thread cutting G32, thread cutting with variable lead G34, Thread cutting in Z direction G33, Thread cutting cycle G92, Multiple thread cutting cycle G76

The machine used for thread cutting must be installed with spindle encoder which lines are set by NO.070m. Drive ratio between spindle and encoder is set by NO.110 and NO.111. X or Z axis traverses to start machine after the system receives spindle signal per rev in thread cutting, and so one thread is machined by multiple roughing, finishing without changing spindle speed.

The system can machine many kinds of thread cutting, such as thread cutting without tool retraction groove. There is a big error in the thread pitch because there are the acceleration and the deceleration at the starting and ending of thread cutting in X, Z direction, and so there is length of thread lead-in and distance of tool retraction at the actual starting and ending of thread cutting。

The traverse speed of tool in X, Z direction is defined by spindle speed instead of cutting feedrate override in thread cutting when the pitch is defined. The spindle override control is valid in thread cutting. When the spindle speed is changed, there is error in pitch caused by acceleration/deceleration in X, Z direction, and so the spindle speed cannot be changed and the spindle cannot be stopped in thread cutting, which will cause tool and workpiece to be damaged.

3.10.1 THREAD CUTTING WITH CONSTANT LEAD G32

Instruction format：G32 X(U)_ Z(W)_ F(I)_ J_ K_ Q_

Instruction function: The path of tool traversing is a straight line from starting point to end point as Fig.3-33; the longer moving distance from starting point to end point(radius value in X direction) is called as the long axis and another is called as the short axis. In course of motion, the long axis traverses one lead when the spindle rotates one rev, and the short axis and the long axis execute the linear interpolation. Form one spiral grooving with variable lead on the surface of workpiece to realize thread cutting with constant lead. Metric pitch and inch pitch are defined respectively by F, I. Metric or inch straight, taper, end face thread and continuous multi-section thread can by machined in G32:

Instruction specification:

G32 is modal;

Pitch is defined to moving distance when the spindle rotates one rev(X axis in radius);

Cutting straight thread when starting point and end point in X direction are the same one(not to input X or U);

Cutting end face thread when starting point and end point in X direction are the same one(not to input Z or W);

Cutting taper thread when starting point and end point in X,Z direction are not the same one;

F: Metric pitch is moving distance of long axis when the spindle rotates one rev: 0.001～500 mm. After F is executed, it is valid until F with specified pitch is executed again.

I: Teeth per inch. It is ones per inch（25.4 mm） in long axis, and also is circles of spindle rotation when the long axis traverses one inch（25.4 mm）:0.06～25400 tooth/inch. After I is executed, it is valid until I with specified pitch is executed again.

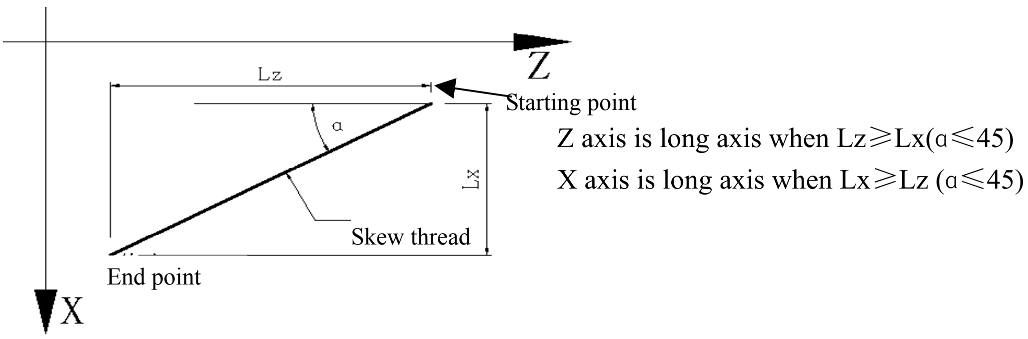
K: Length in long axis when thread run-out: 0～9999.999(unit：mm). If the long axis is X, its value is in radius without direction; K is modal parameter.

Q: Initial angle(offset angle)between spindle rotation one rev and starting point of thread cutting: 0～360000(unit：0.001 degree). Q is non-modal parameter, must be defined every time, otherwise it is 0^0.

Q rules:

1. Its initial angle is 0^0 if Q is not specified;
2. For continuous thread cutting, Q specified by its following thread cutting block except for the first block is invalid, namely Q is omitted even if it is specified;
3. Multi threads formed by initial angle is not more than 65535;
4. Unit : 0.001^0 . Q180000 is input in program if it offsets 180^0 with spindle one rev; if Q180 or Q180.0, it is 0.18^0.

Difference between long axis and short axis is as Fig. 3-33.



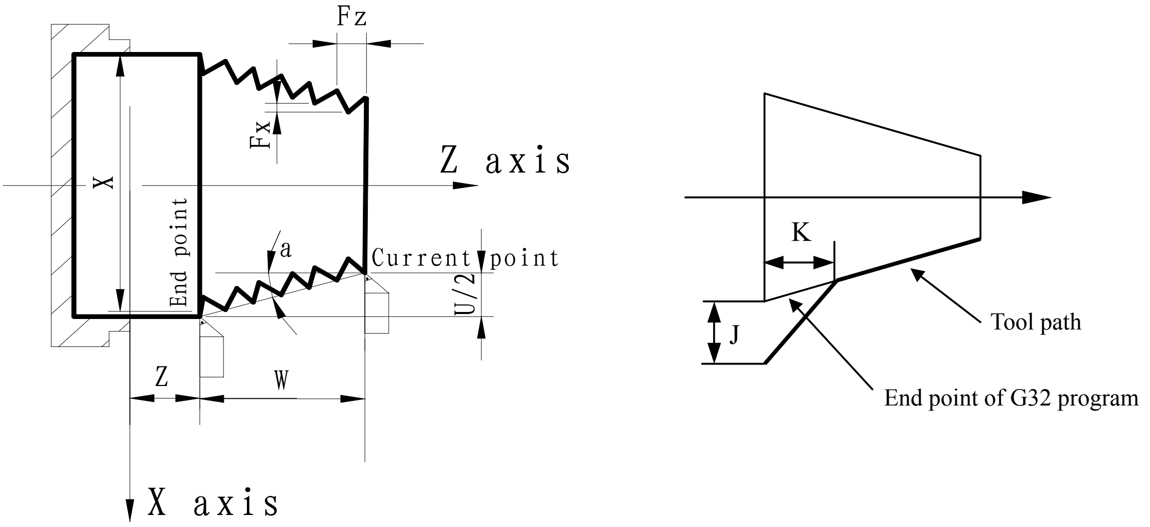


Fig. 3-33 G32 path

Cautions：

- J, K are modal. The thread run-out is previous J, K value when they are omitted in the next block in continuous thread cutting. Their mode are cancelled when no thread cutting are executed;
- There is no thread run-out when J, or J, K are omitted; K=J is the thread run-out value when K is omitted;
- There is no thread run-out when J=0 or J=0,K=0;

- The thread run-out value J=K when J≠0，K=0;
- There is no thread run-out when J=0 or K≠0;
- If the current block is for thread and the next block is the same, the system does not test the spindle encoder signal per rev at starting the next block to execute the direct thread cutting, which function is called as continuous thread machining.
- After the feed hold is executed, the system displays “Pause” and the thread cutting continuously executes not to stop until the current block is executed completely; if the continuous thread cutting is executed, the program run pauses after thread cutting blocks are executed completely.
- In Single block, the program stops run after the current block is executed. The program stops run after all blocks for thread cutting are executed.
- The thread cutting decelerates to stop when the system resets, emergently stop or its driver alarms.

Example: Pitch: 2mm。 δ 1 = 3mm， δ 2 = 2mm，total cutting depth 2mm with two times cut-in.

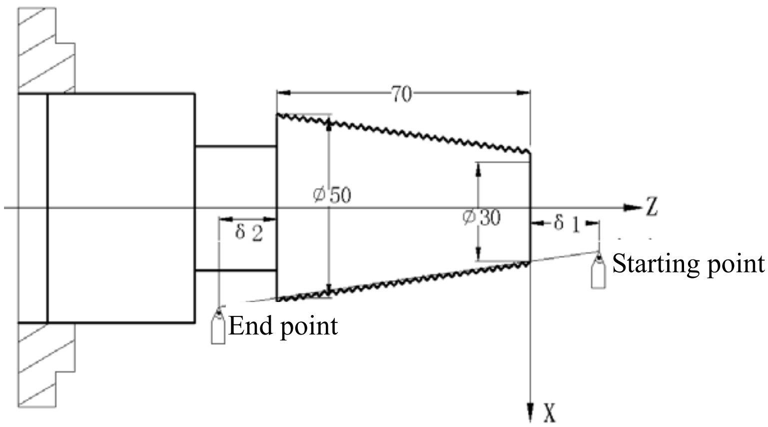


Fig. 3-34

Program: O0009;

```
G00 X28 Z3；          ( First cut-in 1mm )
G32 X51 W-75 F2.0；   ( First taper cutting )
G00 X55；             ( Tool retraction )
W75；                 ( Return to starting point in Z direction )
X27；                 ( Second tool infeed 0.5mm )
G32 X50 W-75 F2.0；   ( Second taper thread cutting  )
G00 X55；             ( Tool retraction )
W75 ；                ( Return to starting point in Z direction )
M30；
```

3.10.2 THREAD CUTTING WITH VARIABLE LEAD G34

Instruction format: G34 X（U）__ Z（W）__ F（I）__ J__ K__ R__ ；

Instruction function: The path of tool traversing is a straight line from starting point to end point in X, Z direction, the longer moving distance from starting point to end point(radius value in X direction) is called as the long axis and another is called as the short axis. In course of motion, the long axis traverses one lead when the spindle rotates one rev, the pitch increases or decreases a specified value per rev and one spiral grooving with variable lead on the surface of workpiece to realize thread cutting with variable lead. Tool retraction can be set in thread cutting.

F, I are specified separately to metric, inch pitch. Machine metric or inch straight, taper, end face thread with variable pitch.

Instruction specifications:

G34 is modal;

Functions of X（U）, Z（W）,J, K are the same that of G32;

F: Metric thread of first pitch from starting point: 0.001～500 mm；

I: Inch thread of first pitch from starting point: 0.06～25400 tooth/inch;

R: Increment or decrement of pitch per rev, R=F1-F2, with direction; F1>F2, pitch decreases when R is negative; F1<F2, pitch increases when R is positive (as Fig. 3-35);

R: ±0.001~±500.000 mm/pitch (metric thread);

±0.060~±25400 tooth/inch (inch thread).

The system alarms when R exceeds the above-mentioned range or the pitch exceeds permissive value or is negative owing to R increases or decreases.

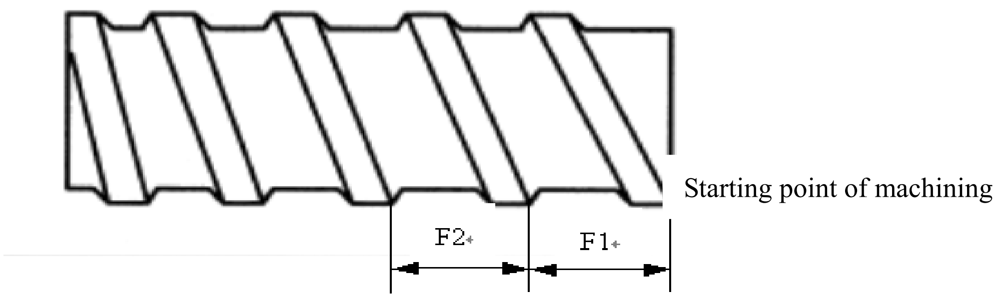


Fig. 3-35 Variable pitch lead machining

Caution：

- It is the same that of G32.

Example：First pitch of starting point :4mm，increment 0.2mm per rev of spindle.

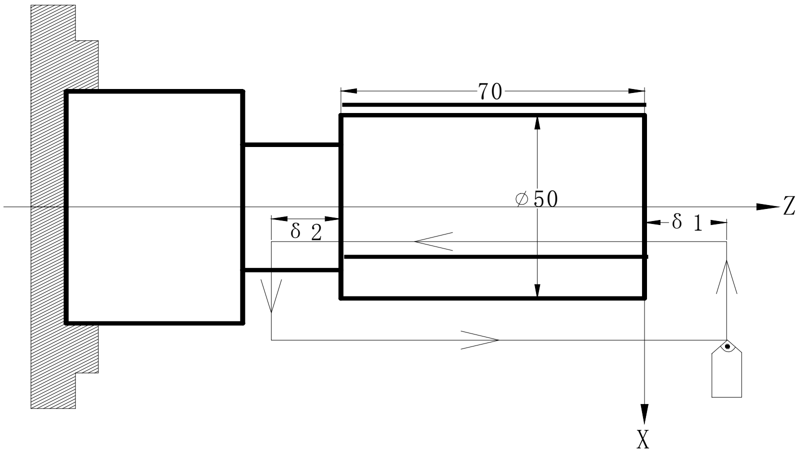


Fig. 3-36 Variable thread machining

When G34 is used many times, use macro variables to simplify programming. δ 1 = 4mm, δ 2 = 4mm, total cutting depth 4mm, total cutting cycle 15 times; first tool infeed 0.8mm, gradual decreasing cutting every time 0.2mm, min. infeed 0.2mm.

Program: O0010;

G00 X60 Z4 M03 S500;	
G65 H01 P#202 Q800;	First tool infeed:: evaluation #202=0.8mm
G65 H01 P#203 Q0;	Cycle count: evaluation #203=0
N10 G65 H02 P#204 Q#203 R1;	Cycle count starting: #204=#203+1
G65 H01 P#203 Q#204;	#203=#204
G65 H81 P30 Q#204 R15;	Total cutting cycle times: #204=15, jump to block N30
G00 U-10;	Tool infeed to Φ50
G65 H01 P#200 Q#202;	Cutting infeed: #200=#202
G00 U-#200;	Tool infeed
G34 W-78 F3.8 J5 K2 R0.2;	Variable pitch cutting
G00 U10;	Tool retraction
Z4;	Return to starting point in Z direction
G65 H03 P#201 Q#200 R200;	Decreasing of cutting feed again: #201=#200—0.2
G65 H01 P#202 Q#201;	Evaluation again #202=#201
G65 H86 P20 Q#202 R200;	Infeed: Jump to block N20 when #202≦0.2mm
G65 H80 P10;	Unconditionally jump to block N10
N20 G65 H01 P#202 R200;	Min. infeed: #202=0.2
G65 H80 P10;	Unconditionally jump to block N10
N30 M30;	

3.10.3 THRED CUTTING in Z DIRECTION G33

Instruction format: G33 Z（W）__ F（I）__ L__ ;

Instruction function: Tool path is from starting point to end point and then from end point to starting point. The tool traverses one pitch when the spindle rotates one rev, the pitch is consistent with pitch of tool and there is spiral grooving in internal hole of workpiece and the internal machining can be completed one time.

Instruction specification: G33 is modal instruction;

Z(W): starting point and end point in Z direction are the same one not to execute the thread cutting when Z or W is not input;

F: metric thread pitch 0.001～500 mm；

I: teeth per inch thread 0.06～25400 teeth/inch;

L: multi threads 1～99. It is single thread when L is omitted.

Cycle process：

① Tool infeed in Z direction (start spindle before G33 is executed)；

② M05 signal outputs after the tool reaches the specified end point in Z direction in progamming；

③ Test spindle after completely stopping；

④ Spindle rotation (CCW) signal outputs；

⑤ The tool retracts to starting point in Z direction；

⑥ M05 signal outputs and the spindle stops；

⑦ Repeat the steps ①～⑤ if multi threads are machined.

Example：Fig.3-37，thread M10×1.5

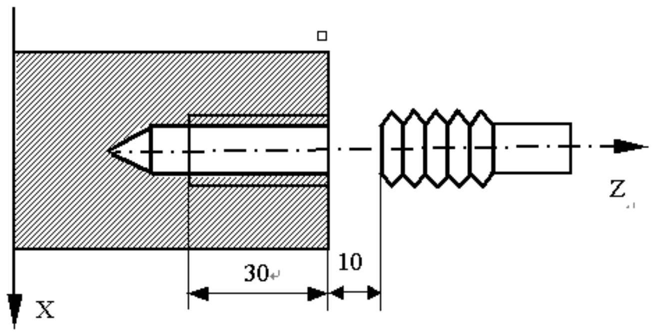


Fig. 3-37

Program：O0011；

G00 Z90 X0 M03；	Start spindle
G33 Z50 F1.5；	Tap cycle
M03	Start spindle again
G00 X60 Z100；	Machine continuously
M30	

Note 1: Before tapping, define rotation direction of spindle according to tool rotating. The spindle stops rotation after the tapping is completed and the spindle is started again when machining thread continuously.

Note 2: G33 is for rigid tapping. The spindle decelerates to stop after its stop signal is valid, at the moment, the tool continuously infeeds in Z direction along with the spindle rotating, and so the actual cutting bottom hole is deeper than requirement and the length is defined by the spindle speed and its brake in tapping.

Note 3: Rapid traverse speed in tapping in Z direction is defined by spindle speed and pitch is not related to cutting feedrate override.

Note 4: In Single block to feed hold, the tapping cycle continuously executes not to stop until the tool returns to starting point when the system displays "Pause".

Note5: The thread cutting decelerates to stop when the system resets, emergently stop or its driver alarms.

3.10.4 THREAD CUTTING CYCLE G92

Instruction format: G92 X（U）_ Z（W）_ F_ J_ K_ L_;（Metric straight thread cutting cycle）

G92 X（U）_ Z（W）_ I_ J_ K_ L_; （Inch straight thread cutting cycle）

G92 X（U）_ Z（W）_ R_ F_ J_ K_ L_; （Metric taper thread cutting cycle）

G92 X（U）_ Z（W）_ R_ I_ J_ K_ L_; （Metric taper thread cutting cycle）

Instruction function: Tool infeeds in radial(X axis) direction and cuts in axial(Z axis or X, Z axis) direction from starting point of cutting to realize straight thread, taper thread cutting cycle with constant thread pitch. Thread run-out in G92: at the fixed distance from end point of thread cutting, the tool executes thread interpolation in Z direction and retracts with exponential or linear acceleration in X direction, and retracts at rapidly traverse speed in X direction after it reaches to end point of cutting in Z direction as Fig. 3-41.

Instruction specifications:

G92 is modal;

Starting point of cutting: starting position of thread interpolation;

End point of cutting: end position of thread interpolation;

X: absolute coordinate of end point of cutting in X direction, unit:mm;

U: different value of absolute coordinate from end point to starting point of cutting in X direction, unit:mm;

Z: absolute coordinate of end point of cutting in Z direction, unit:mm;

W: different value of absolute coordinate from end point to starting point of cutting in X direction, unit:mm;

R: different value(R value) of absolute coordinate from end point to starting point of cutting in X direction. When the sign of R is not the same that of U, R｜≤｜U/2｜, unit:mm.

F=0.001～500 mm, metric thread pitch. After F value is executed, it is reserved and can be omitted;

I=0.06～25400 tooth/inch, metric thread teeth per inch, After F value is executed, it is not reserved and can be not omitted;

J: Moving distance in the short axis in thread run-out is 0~9999.999 (unit: mm) without direction (automatically define its direction according to starting position of program), and it is modal parameter. If the short axis is X, its value is specified by radius;

K: Moving distance in the long axis in thread run-out is 0~9999.999 (unit: mm) without direction (automatically define its direction according to starting position of program), and it is modal parameter. If the long axis is X, its value is specified by radius;

L: Multi threads: 1～99 and it is modal parameter. (the system defaults it is single thread when L is omitted)

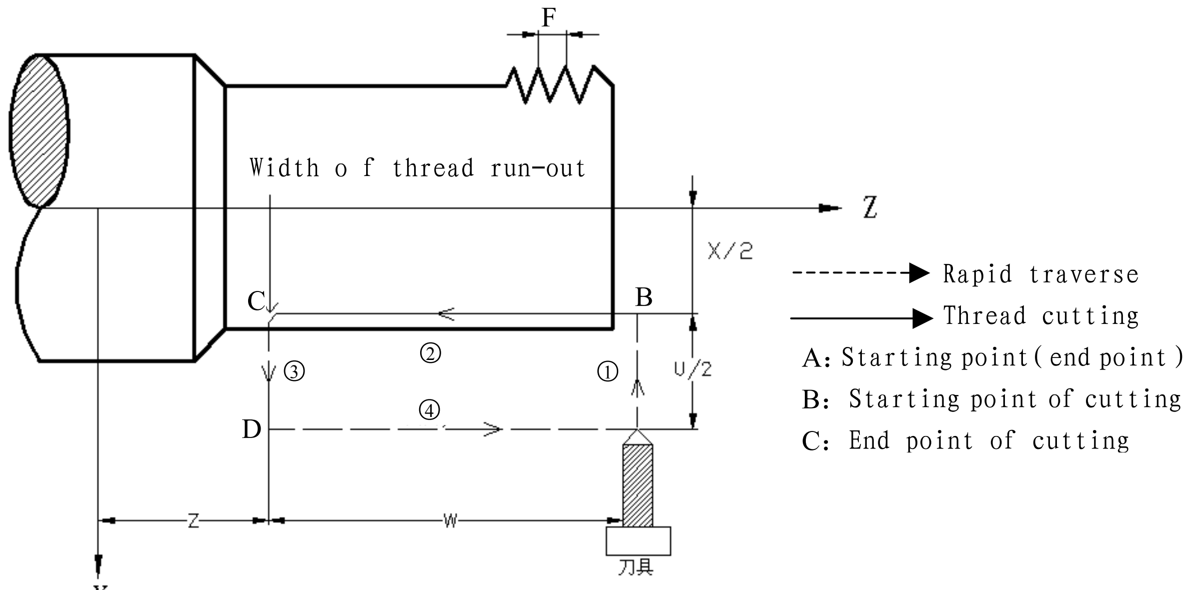


Fig. 3-39

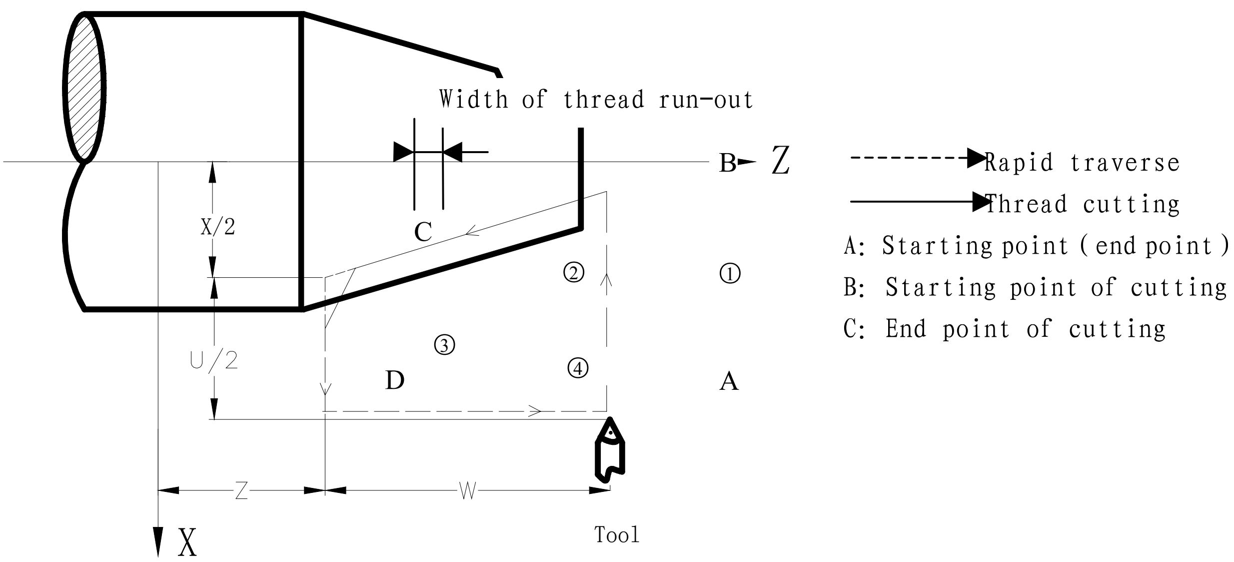


Fig. 3-40

The system can machine one thread with many tool infeeds in G92, but cannot do continuous two thread and end face thread. Definition of thread pitch in G92 is the same that of G32, and a pitch is defined that it is a moving distance of long axis(it is in radius in X direction) when the spindle rotates one rev.

Pitch of taper thread is defined that it is a moving distance of long axis(it is in radius in X direction). When absolute value of coordinate difference between B point and C point in Z direction is more than that of X (in radius), Z axis is long axis; and vice versa.

Cycle process: straight thread as Fig.3-39 and taper thread as Fig.3-40.

① The tool rapidly traverses from starting point to cutting starting point in X direction;

② Thread interpolates(linear interpolation) from the cutting starting point to cutting end point;

③ Retract the tool at the cutting feedrate in X direction (opposite direction to the above-mentioned ①), and return to the position which the absolute coordinate in X direction and the starting point are the same;

④ The tool rapidly traverses to return to the starting point in Z direction and the cycle is completed.

Cautions:

- Length of thread run-out is specified by NO.019 when J, K are omitted;
- Length of thread run-out is K in the long direction and is specified by NO.019 when J is omitted;
- Length of thread run-out is J=K when K is omitted;
- There is no thread run-out when J=0 or J=0, K=0;
- Length of thread run-out is J=K when J≠0，K=0;
- There is no thread run-out when J=0，K≠0;
- After executing the feed hold in thread cutting, the system does not stop cutting until the thread cutting is completed with ***Pause*** on screen;
- After executing single block in thread cutting, the program run stops after the system returns to starting point(one thread cutting cycle is completed).
- Thread cutting decelerates to stop when the system resets, emergently stop or its driver alarms.

Instruction path: relative position between thread cutting end point and starting point with U, W, R and tool path and thread run-out direction with different U, W, R signs as Fig. 3-41:

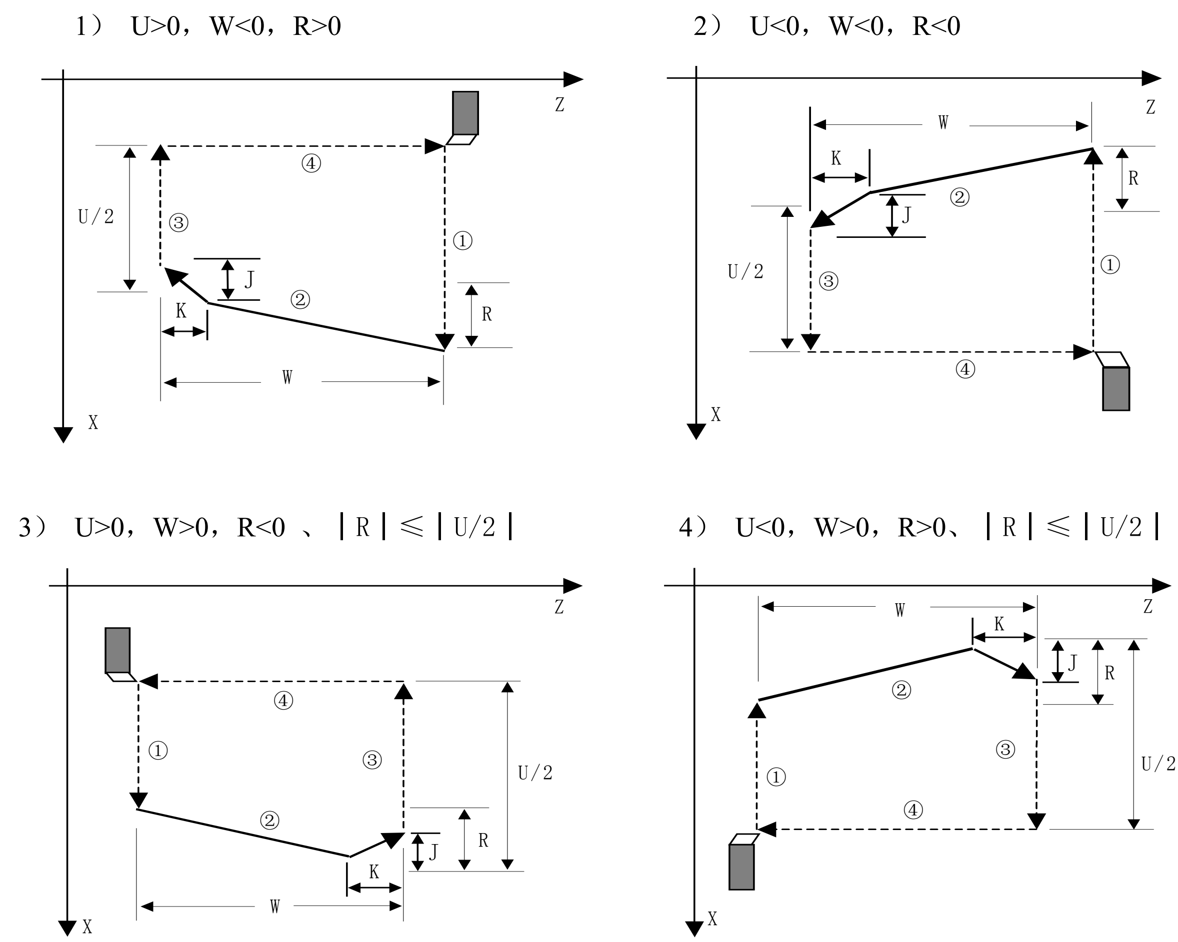


Fig. 3-41

Example: Fig.3-42

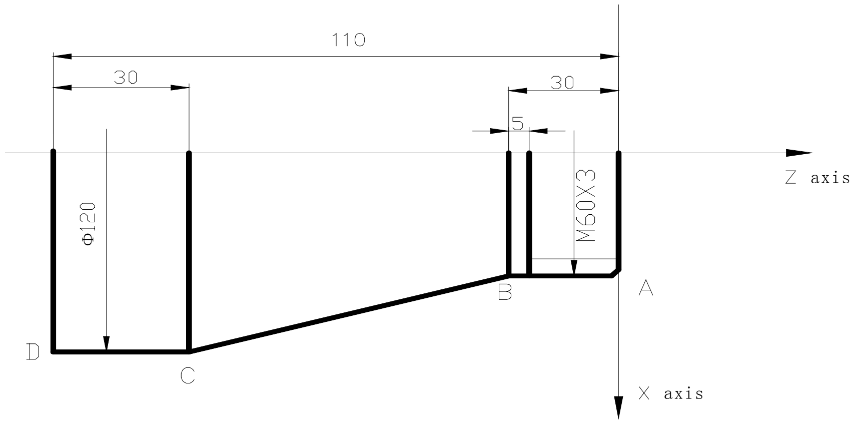


Fig. 3-42

Program：O0012；
M3 S300 G0 X150 Z50 T0101； （Thread tool）
G0 X65 Z5； (Rapid positioning)
G92 X58.7 Z-28 F3 J3 K1； （Machine thread with 4 times cutting, the first tool infeed 1.3mm）
X57.7 ； （The second tool infeed 1mm）
X57； （The third tool infeed 0.7mm）
X56.9； （The fourth tool infeed 0.1mm）
M30；

3.10.5 MULTIPLE THREAD CUTTING CYCLE G76

Instruction format：G76 P（m）（r）（a） Q（△dmin） R（d）；

G76 X（U）__ Z（W）__ R（i） P（k） Q（△d） F（I）__；

Instruction function：Machining thread with specified depth of thread (total cutting depth)is completed by multiple roughing and finishing, if the defined angle of thread is not 0°, thread run-in path of roughing is from its top to bottom, and angle of neighboring thread teeth is the defined angle of thread. G76 can be used for machining the straight and taper thread with thread run-out path, which is contributed to thread cutting with single tool edge to reduce the wear of tool and to improve the precision of machining thread. But G76 cannot be used for machining the face thread. machining path is as Fig. 3-43(a):

Relevant definitions：

Starting point(end point): position before block runs and behind blocks run, defined by A point;

End point of thread(D point): end point of thread cutting defined by X（U）__ Z（W）__,.

The tool will not reach the point in cutting if there is the thread run-out path;

Starting point of thread(C point): its absolute coordinates is the same that of A point and the different value of absolute coordinates between C and D in X direction is i(thread taper with radius value). The tool cannot reach C point in cutting when the defined angle of thread is not 0°;

Reference point of thread cutting depth（B point）: its absolute coordinates is the same that of A point and the different value of absolute coordinates between B and C in X direction is k(thread taper with radius value).The cutting depth of thread at B point is 0 which is the reference point used for counting each thread cutting depth by the system;

Thread cutting depth: it is the cutting depth for each thread cutting cycle. It is the different value (radius value, without signs) of absolute coordinates in X direction between B and intersection of reversal extension line for each thread cutting path and straight line BC. The cutting depth for each roughing is $\sqrt{n}\times\triangle d$, n is the current roughing cycle times, △d is the thread cutting depth of first roughing;

Travel of thread cutting: different value between the current thread current depth and the previous one: ：$(\sqrt{n}-\sqrt{n-1})\times\triangle d$；

End point of tool retraction: it is the end position of radial (X axis) tool retraction after the thread cutting in each thread roughing, finishing cycle is completed, defining with E point;

Run-in path of thread:

X: Absolute coordinates (unit: mm) of thread end point in X direction;

U: Different value (unit: mm) of absolute coordinates between thread end point and starting point in X direction;

Z: Absolute coordinates (unit: mm) of thread end point in Z direction;

W: Different value (unit: mm) of absolute coordinates between thread end point and starting point in Z direction;

P(m)：Times of thread finishing: 00～99 (unit：times) with 2-digit digital. It is valid after m instruction value is executed, and the value of system parameter NO.057 is rewritten to m. The value of system parameter No.057 is regarded as finishing times when m is not input. The thread is finished according to the programmed thread path, the first finishing cutting travel is d and the following one is 0,

P(r)：Width of thread run-out 00～99(unit: 0.1×L，L is the thread pitch) with 2-digit digital. It is valid after r instruction value is executed and the value of system parameter NO.019 is rewritten to r. The value of system parameter NO.019 is the width of thread run-out when r is not input. The thread run-out function can be applied to thread machining without tool retraction groove and the width of thread run-out defined by system parameter NO.019 is valid for G92;

P(a)：Angles at taper of neighboring two tooth are 00, 29, 30, 55, 60, 80, unit: degree（°），with 2-digit digital. It is valid after a instruction value is executed and the value of system parameter NO.058 is rewritten to a. The value of system parameter NO.058 is regarded as angle of thread tooth. The actual angle of thread in defined by tool ones and so a should be the same as the tool angle;

△ Q(△dmin)：Minimum cutting travel of thread roughing(unit: 0.001mm, radius value without signs). When $(\sqrt{n}-\sqrt{n-1})\times\triangle d<\triangle dmin$, △dmin is regarded as the cutting travel of current roughing, i.e. depth of current thread cutting is $(\sqrt{n-1}\times\triangle d+\triangle dmin)$.

△dmin is applied because the cutting travel of roughing is undersize and the times of roughing is excessive, which is caused the cutting travel of thread roughing gradually decreases. After Q（△dmin） is executed, the instruction value △dmin is value and the value of system parameter NO.059 is rewritten to minimum cutting travel;

R(d): It is the cutting travel of thread finishing, and is the different value(unit:mm, radius value without signs) of absolute coordinates in X direction between cut-in point B_e of thread finishing and B_f of thread roughing. After R（d）is executed, the instruction value d is value and the value of system parameter NO.060 is rewritten to d×1000（unit：0.001 mm）. The value of system parameter NO.060 is regarded as the cutting travel of thread finishing when R（d）is not input.

R(i)：It is thread taper and is the different value of absolute coordinates between thread starting point and end point in X direction(unit:mm, radius value). The system defaults i=0(straight thread) when i is not input;

P(k)：It is the depth of thread tooth and is also the total cutting depth of thread(unit: 001mm, radius value without signs);

Q(△d)：It is the first depth of thread cutting (unit: 0.001mm, radius value without signs).The system alarms when △d is not input;

F: 0.001～500 mm，metric thread pitch.

I: 0.06～25400 tooth/inch, thread teeth per inch for inch thread.

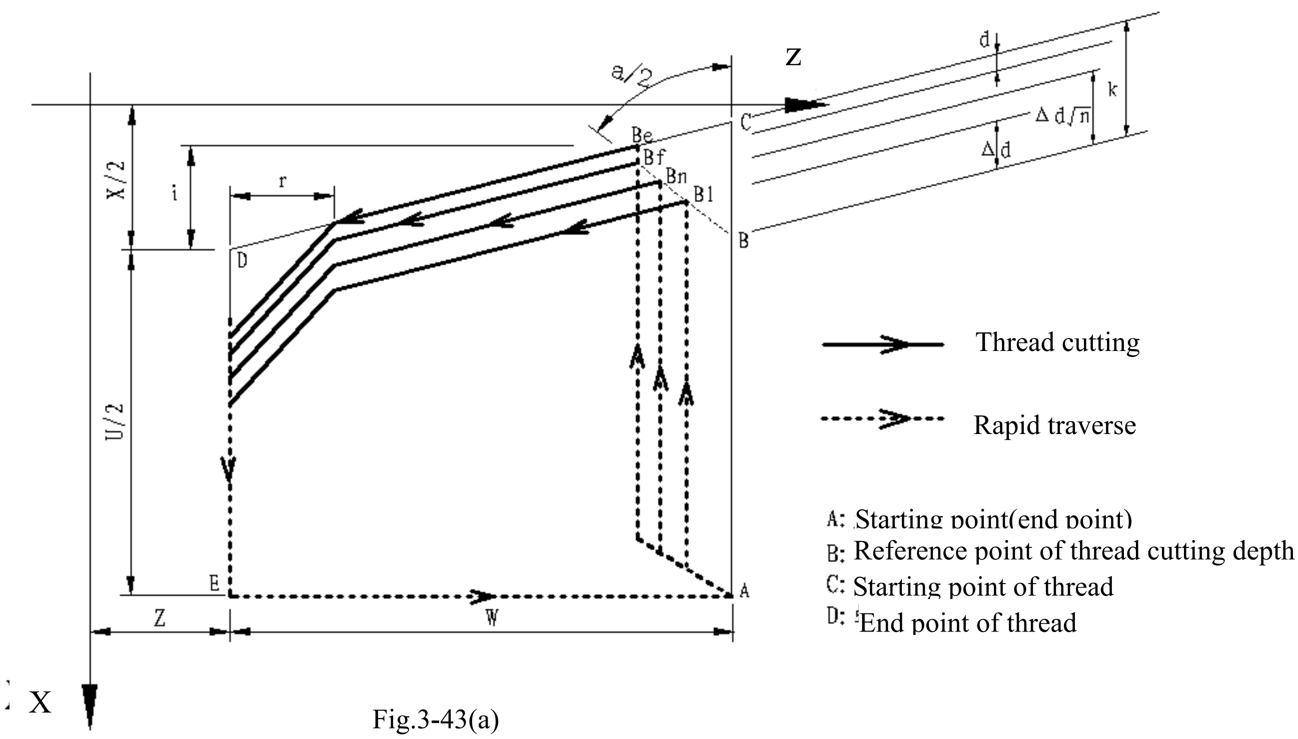


Fig.3-43(a)

Cut-in method as follows:3-43(b)

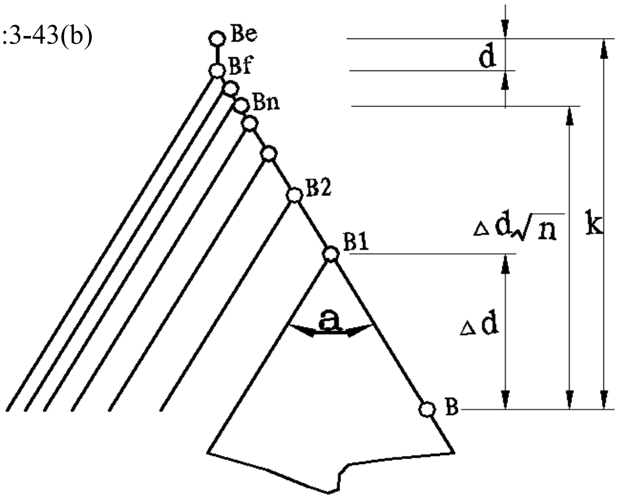


Fig. 3-43(b)

Pitch is defined to moving distance (radius value in X direction) of long axis when the spindle rotates one rev. Z axis is long when absolute value of coordinate difference between C point and D point in Z direction is more than that of X direction (radius value, be equal to absolute value of i); and vice versa.

Execution process:

① The tool rapidly traverses to B_1 , and the thread cutting depth is △d. The tool only traverses in X direction when a=0; the tool traverses in X and Z direction and its direction is the same that of A→D when a≠0;

② The tool cuts threads paralleling with C→D to the intersection of D→E (r≠0: thread run-out);

③ The tool rapidly traverses to E point in X direction;

④ The tool rapidly traverses to A point in Z direction and the single roughing cycle is completed;

⑤ The tool rapidly traverses again to tool infeed to B_n (is the roughing times), the cutting depth is the bigger value of ($\sqrt{n}$ ×△d), ($\sqrt{n-1}$ ×△d+△dmin), and execute ② if the cutting depth is less than (k-d); if the cutting depth is more than or equal to (k-d), the tool infeeds (k-d) to B_f ,and then, execute ⑥ to complete the last thread roughing;

⑥ The tool cuts threads paralleling with C→D to the intersection of D→E (r≠0: thread run-out);

⑦ The tool rapidly traverses to E point in X direction;

⑧ The tool rapidly traverses to A point in Z direction and the thread roughing cycle is completed to execute the finishing;

⑨ After the tool rapidly traverses to B(the cutting depth is k and the cutting travel is d), execute the thread finishing, at last the tool returns to A point and so the thread finishing cycle is completed;

⑩ If the finishing cycle times is less than m, execute ⑨ to perform the finishing cycle, the thread cutting depth is k and the cutting travel is 0; if the finishing cycle times is equal to m, G76 compound thread machining cycle is completed.

Cautions:

- In thread cutting, execute the feed hold, the system displays ***Pause*** after the thread cutting is executed completely, and then the program run pauses;
- Execute single block in thread cutting, the program run stops after returning to starting point(one thread cutting cycle is completed);

- The thread cutting decelerates to stop when the system resets and emergently stop or the driver alarms;
- Omit all or some of G76 P（m）（r）（a） Q（△dmin） R（d）. The omitted address runs according to setting value of parameters;
- m, r, a used for one instruction address P are input one time. Program runs according to setting value of NO.57, 19, 58 when m, r, a are all omitted; Setting value is a when address P is input with 1 or 2-bit digits; setting values are r, a when address P is input with 3 or 4-bit digits;
- The direction of A→C→D→E is defined by signs of U,W , and the direction of C→D is defined by the sign of R（i）. There are four kinds of sign composition of U, W corresponding to four kinds of machining path as Fig. 3-44.

Example: Fig. 3-44，thread M68 × 6。

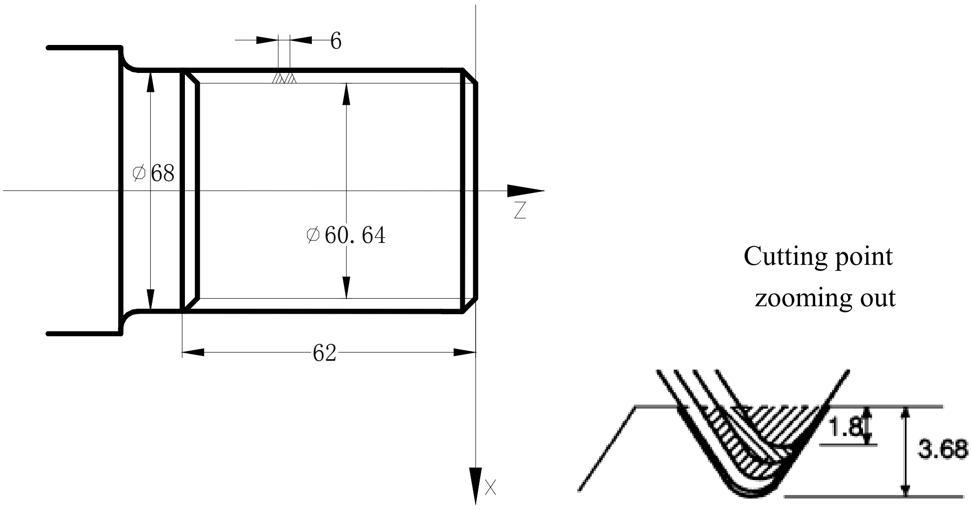


Fig. 3-44

Program：O0013；	
G50 X100 Z50 M3 S300；	（Set workpiece coordinate system, start spindle and specify spindle speed）
G00 X80 Z10；	（Rapid traverse to starting point of machining）
G76 P020560 Q150 R0.1；	（Finishing 2 times, chamfering width 0.5mm, tool angle 60°, min. cutting depth 0.15, finishing allowance 0.1）
G76 X60.64 Z-62 P3680 Q1800 F6；	(Tooth height 3.68, the first cutting depth1.8)
G00 X100 Z50 ；	（Return to starting point of program）
M30；	（End of program）

3.11 CONSTANT SURFACE SPEED CONTROL G96, CONSTANT ROTATIONAL SPEED CONTROL G97

Instruction format：G96 Sxxxx；（S0000～S9999，）

Instruction function：the constant surface speed control is valid, the cutting surface speed is defined (m/min) and the constant rotational speed control is cancelled. G96 is modal G instruction. If the current modal is G96, G96 can not be input.

Instruction format：G97 Sxxxx；（S0000～S9999，the leading zero can be omitted.）

Instruction function：the constant surface speed control is cancelled, the constant rotational speed control is valid and the spindle speed is defined(rev/min). G96 is modal G instruction. If the current modal is G97, G97 cannot be input.

Instruction format：G50 Sxxxx；（S0000～S9999，the leading zero can be omitted.）

Instruction function: define max. spindle speed limit (rev/min) in the constant surface speed control and take the current position as the program reference point.

G96, G97 are the modal word in the same group but one of them is valid. G97 is the initial word and the system defaults G97 is valid when the system turns on.

When the machine tool cuts it, the workpiece rotates based on the axes of spindle as the center line, the cutting point of tool cutting workpiece is a circle motion around the axes of spindle, and the instantaneous speed in the circle tangent direction is called the cutting surface(for short surface speed). There are different surface speed for the different workpiece and tool with different material.

When the spindle speed controlled by the analog voltage is valid, the constant surface control is valid. The spindle speed is changed along with the absolute value of X absolute coordinates of programming path in the constant speed control. If the absolute value of X absolute coordinates adds, the spindle speed reduces, and vice verse, which make the cutting surface speed as S instruction value. The constant speed control to cut the workpiece makes sure all smooth finish on the surface of workpiece with diameter changing.

Surface speed=spindle speed× |X| × π÷1000 (m/min)

Spindle speed: rev/min

|X|：absolute value of X absolute coordinate value (diameter value), mm

π≈3.14

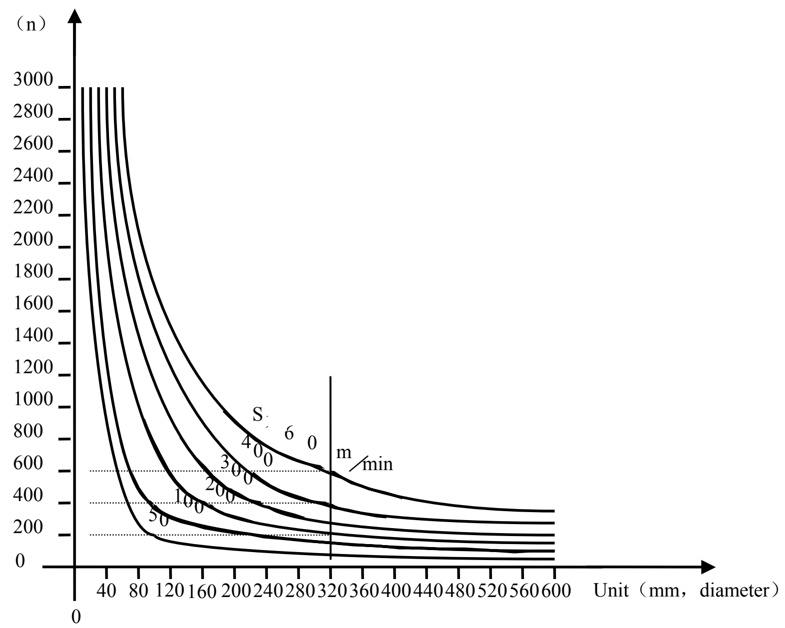


Fig. 2-4

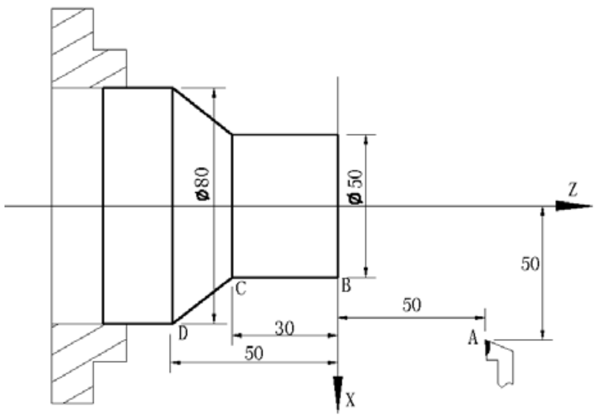
In G96, the spindle speed is changed along with the absolute value of programming path X absolute coordinate value in the course of cutting feed (interpolation), but it is not changed in G00 because there is no actual cutting and is counted based on the surface speed of end point in the program block.

In G96, Z coordinates axis of workpiece system must consist with the axes of spindle (rotary axis of workpiece), otherwise, there is different between the actual surface speed and the defined one.

In G96, G50 S_ can limit max. spindle speed (rev/min). The actual spindle speed is the limit value of max. speed when the spindle speed counted by the surface speed and X coordinate value is more than the max. spindle speed set by G50 S_. After the system powers on, max. spindle speed limit value is not defined and its function is invalid. Max. spindle speed limit value defined by G50 S_ is reserved before it is defined again and its function is valid in G96. Max. spindle speed defined by G50 S_ is invalid in G97 but its limit value is reserved.

Note: In G96, the spindle speed is limited to 99 rev/min (set by NO.043) if G50, S0 are executed.
When the constant surface speed is controlled by the system parameter NO.043, the spindle speed is lower limit, which is higher than one counted by the surface speed and X axis coordinate value

Example：



Program：O0014；

M3 G96 S300；（Spindle rotates clockwise, the constant surface speed control is valid and the surface speed is 300m/min）
G0 X100 Z100；（Rapid traverse to A point with spindle speed 955 rev/min）
G0 X50 Z0；（Rapid traverse to B point with spindle speed 1910 rev/min）
G1 W-30 F200；（Cut from B to C with spindle speed 1910 rev/min）
X80 W-20 F150；（Cut from C to D with spindle speed 1910 rev/min and surface speed 1194 rev/min）
G0 X100 Z100；（Rapid retract to A point with spindle speed 955 rev/min）
M30；（End of program, spindle stop and coolant OFF）

Note 1: In G96, S value commanded is reserved in g97. Its value is resumed after it returns to G96. Example:

G96 S50；（Cutting surface speed 50m/min）
G97 S1000；（Spindle speed 1000 rev/min
G96 G01 X200；（Cutting surface speed 50m/min）

Note 2: The constant surface speed control is valid when the machine tool is locked (X, Z axis do not move when their motion instruction are executed);

Note 3: To gain the precise thread machining, it should not be adopted with the constant surface speed control but the constant rotational speed (G97) in the course of thread cutting;

Note 4: From G96 to G97, if none of S instruction (rev/min) is commanded in the program block in G97, the last spindle speed in G96 is taken as S instruction in G97, namely, the spindle speed is not changed at this time;

Note 5: In G96, when the spindle speed counted by the cutting surface speed is more than max. speed of current spindle gear (system parameter NO.037～NO.040), at this time, the spindle speed is limited to max. one of current spindle gear.

3.12 FEEDRATE per MINUTE G98, FEEDRATE per REV G99

Instruction format:G98 Fxxxx；（F0001～F8000，the leading zero can be omitted, feed rate per minute is specified, mm/min）

Instruction function: cutting feed rate is specified as mm/min, G98 is the modal G instruction. G98 cannot be input if the current instruction is G98 modal.

Instruction format:G99 Fxxxx； （F0.0001～F500，the leading zero can be omitted）

Instruction function: cutting feed rate is specified as mm/min, G99 is the modal G instruction. G99 cannot be input if the current instruction is G98 modal.

The cutting feed per rev specified by G99 F_ is contributed to the equable cutting line on the surface of workpiece. In G99, the machine tool must be adopted with the spindle encoder to machine the workpiece on the machine tool

G98, G99 are the modal G instruction in the same group and only one is valid. G98 is the initial state G instruction and the system defaults G98 is valid when the system turns on.

Reduction formula of feed between per rev and per min:

$$F_m = F_r \times S$$

F_m：feed per min （mm/min）；

F_r：feed per rev（mm/r）；

S：spindle speed （r/min）.

After the system turns on, the feedrate is ones set by NO.030 and F value is reserved after F is commanded. The feed rate is 0 after F0 is executed. F value is reserved when the system resets and emergently stops.

Parameters：

System parameter NO.027: the upper limit value of cutting feed rate(they are the same in X, Z axis，diameter/min in X axis);

System parameter NO.029: exponential function for time constant of acceleration/ deceleration when cutting feed and manual feed;

System parameter NO.030: initial (ultimate) speed of acceleration/deceleration in exponential function when cutting feed and manual feed.

Note: In G99 modal, there is the uneven cutting feed rate when the spindle speed is lower than 1 rev/min; there *is the follow error in the actual cutting feed rate when there is the swing in the spindle speed. To gain the high machining quality, it is recommended that the selected spindle speed should be not lower than min. speed of spindle servo or inverter*

3.13 MACRO INSTRUCTIONS

The system provides the macro instruction which similar to the high language, and can realize the variable evaluation, add and subtract operation, logic decision and conditional jump by user macro instruction, contributed to compiling part program for special workpiece, reduce the fussy counting and simplify the user program

3.13.1 MACRO VARIABLES

（1）Usage of macro variables

Macro variables can command the address values in program, or evaluate the variable or set directly variable by keyboard. Many macro variables can be used in program and they can distinguish with macro variables number.

- **Presentation of macro variables**

  Present with“#”＋ macro variables number.；

  Format：# **i**（i=200，202，203，……）;

  Example：#205，#209，#225。

- **Macro variables reference**

1,Macro variables can replace instruction values

Format： < Address > + “# i”或 < Address > + “－# I”. It shows the system takes variable valueor negative value of variable value as address value.

Example: F#203…when #203=15，its function is the same as F15;

Z-#210…when #210=250，its function is the same as Z-250;

G#230…when #230=3，its function is the same as G3.

2. Macro variables can replace macro variables values.

Format: “＃”+“9”+macro variables number

Example: if #200 = 205，#205 = 500,

The instruction function of X#9200 is the same as X500;

The instruction function of X-#9200 is the same as X-500

Note 1: The address 0 and N cannot refer macro variables;

Note 2: If macro variables values exceed the maximum rang of instruction values, they cannot be used;

Example: M#230 exceeds max. instruction value when #230 = 120.

（**2**）Variety of macro variables

According to macro variable numbers, macro variables are divided into common macro variables and system macro variables.

- Common macro variables

  Common macro variables(#200～#231,#500～#515) are common in all user programs, i.e. Macro variables defined in the program 1 can be applied to the program 2 or program 3.

  The values of common variables(#200～#231,#500～#515) are reserved after power off

- System macro variables

  Use of system macro variables are fixed in the system with interface input signals #1000～#1015 and interface output signals #1100～#1107,

  Interface input/output signal of system variables and other function interface signals share one interface which is valid set by parameters, and the interface input signal of system variables is valid when the corresponding interface signal is valid.

  The system judges and executes other operations including jumping after it reads value of interface input signal #1000～1015（values of #1005—#1015corresponds to 0/1）

  Interface signals of system variables #1000～#1015 are defined as follows:

Bit No.：	7	6	5	4	3	2	1	0
Diagnostic No.00	*TCP	DIQP	*DECX	BDT	T04	T03	T02	T01
				DITW				
Macro variable No.	#1007	#1006	#1005	#1004	#1003	#1002	#1001	#1000
Socket pin No.	XS6:49	XS6:47	XS40:1	XS40:2	XS40:3	XS40:4	XS40:5	XS40:6

Diagnostic No. 001	*SP	*ST	*DECZ	*ESP				
Macro variable No.	#1015	#1014	#1013	#1012				
Socket pin No.	XS40:7	XS40:8	XS40:9	XS40:10				

Diagnostic No.002	T08	T07	T06	T05				
	M42I	M41I		*SPEN				
	*OV8	*0V4	*0V2	*0V1				
Macro variable No.	#1011	#1010	#1009	#1008				
Socket pin No.	XS40：19	XS40：20	XS40：21	XS40：22				

Evaluation of #1100～1105 is 1 or 0, and output state of its interface signals can be changed.

Interface signals of system variables #1100～#1105 are defined as follows:

Diagnostic No.005			M13	M11	S04	S03	S02	S01
			U05	U04	M44	M43	M43	M41
			DOQPS	DOTWS	U03	U02	U01	U00
Macro variable No.			#1105	#1104	#1103	#1102	#1101	#1100
Socket pin No.			XS39:10	XS39:9	XS39:8	XS39:14	XS39:1	XS39:5

3.13.2 OPERATION and JUMP INSTRUCTION G65

Instruction format：

G65 H<u>m</u> P<u># i</u> Q<u># j</u> R<u># k</u>；

m: operation or jump instruction, range 01～99.

I: macro variables name for storing values.

j: macro variables name 1 for operation, can be constant.

k: macro variables name 2 for operation, can be constant.

Instruction functions: # i = #j O # k

Operating sign specified by H<u>m</u>

Example：P#200 Q#201 R#202…..#200 = #201 O #202；

P#200 Q#201 R15….#200 = #201 O 15；

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

Explanation：

- The values of macro variables have no decimal points and function of values of each macro variables is the same that of each address without decimal point;
- Macro variable name has no “#” when it is presented directly with constant.

Macro instruction list

Instruction format	Functions	Definitions
G65 H01 P#I Q#J	Evaluation	# i = # j　assign value of j to i
G65 H02 P#i Q#j R#k；	Decimal add operation	# i = # j + # k
G65 H03 P#i Q#j R#k；	Decimal subtract 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	# i = # j OR # k
G65 H12 P#i Q#j R#k；	Binary multiplication（operation）	# i = # j AND # k
G65 H13 P#i Q#j R#k；	Binary exclusive or	# i = # j XOR # k
G65 H21 P#i Q#j；	Decimal square root	# i = $\sqrt{\# j}$
G65 H22 P#i Q#j；	Decimal absolute value	# i = |# j |
G65 H23 P#i Q#j R#k；	Decimal remainder	Remainder of # i = （#j÷# k）
G65 H24 P#i Q#j；	Decimal into binary	# i = BIN(# j)
G65 H25 P#i Q#j；	Binary into decimal	# i = DEC(# j)
G65 H26 P#i Q#j R#k；	Decimal multiplication/division operation	# i = # i×# j÷# k

Instruction format	Functions	Definitions
G65 H27 P#i Q#j R#k;	Compound 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 jump	Jump to block n
G65 H81 Pn Q#j R#k;	Conditional jump 1	Jump to block n if # j = # k, otherwise the system executes in order
G65 H82 Pn Q#j R#k;	Conditional jump 2	Jump to block n if # j ≠ # k, otherwise the system executes in order
G65 H83 Pn Q#j R#k;	Conditional jump 3	Jump to block n if # j > # k, otherwise the system executes in order
G65 H84 Pn Q#j R#k;	Conditional jump 4	Jump to block n if # j < # k, otherwise the system executes in order
G65 H85 Pn Q#j R#k;	Conditional jump 5	Jump to block n if # j ≥ # k, otherwise the system executes in order
G65 H86 Pn Q#j R#k;	Conditional jump 6	Jump to block n if # j ≤# k, otherwise the system executes in order
G65 H99 Pn;	P/S alarm	（500+n）alarms

1 Operation instructions

- **Evaluation of macro variables**：# I = # J

G65 H01 P#I Q#J

（Example）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 add operation：# I = # J+# K

G65 H02 P#I Q#J R#K

（Example）G65 H02 P#201 Q#202　R15；　　（#201 = #202+15）

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

G65 H03 P#I Q#J R# K

（Example）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

（Example）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

（Example）G65 H05 P#201 Q#202 R#203；（#201 = #202÷#203）

6）Binary logic add（or）：# I = # J.OR. # K

G65 H11 P#I Q#J R#K

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

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

G65 H12 P#I Q#J R#K

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

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

G65 H13 P#I Q#J R#K

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

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

G65 H21 P#I Q#J

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

10）Decimal absolute value：# I = | # J |

G65 H22 P#I Q#J

（Example）G65 H22 P#201 Q#202 ；（#201 = | #202 |）

11）Decimal remainder：# I = # J－TRUNC(#J/#K)×# K，TRUNC：omit decimal fraction

G65 H23 P#I Q#J R#K

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

12）Decimal converting into binary：# I = BIN (# J)

G65 H24 P#I Q#J

（Example）G65 H24 P#201 Q#202 ；（#201 = BIN（#202））

13）Binary converting into decimal：# I = BCD (# J)

G65 H25 P#I Q#J

（Example）G65 H25 P#201 Q#202 ；（#201 = BCD（#202））

14）Decimal multiplication/division operation：# I =（# I×# J）÷# K

G65 H26 P#I Q#J R# k

（Example）G65 H26 P#201 Q#202 R#203；（#201 =（# 201×# 202）÷# 203）

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

G65 H27 P#I Q#J R#K

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

16）Sine：# I = # J•SIN（# K）(Unit：‰）

G65 H31 P#I Q#J R#K

（Example）G65 H31 P#201 Q#202 R#203；（#201 = #202•SIN（#203））

17）Cosine：# I = # J•COS（# K）(Unit：‰）

G65 H32 P#I Q#J R# k

（Example）G65 H32 P#201 Q#202 R#203； （#201 =#202•COS（#203））

18）Tangent：# I = # J•TAM（# K）(Unit：‰）

G65 H33 P#I Q#J R# K

（Example）G65 H33 P#201 Q#202 R#203； （#201 = #202•TAM（#203））

19）Cosine：# I = ATAN（# J /# K）(Unit：‰）

G65 H34 P#I Q#J R# k

（Example）G65 H34 P#201 Q#202 R#203； （#201 =ATAN（#202/#203））

Note 1: Unit of（P）~（S）: degree, 1‰ degree;

Note 2: Variable value is integer, and decimal is omitted. Unit: μm;

Note 3: Variable value displays correctly -9999999 ~ 9999999 in -2^{32} ~ $+2^{32}$-1, otherwise the system displays *******.

2 Jump instruction

1） Unconditional jump

G65 H80 Pn； n：Block number

（Example）G65 H80 P120;（jump to N120）

2）Conditional jump 1 #J.EQ.# K (=)

G65 H81 Pn Q#J R# K； n：Block number

(Example) G65 H81 P1000 Q#201 R#202；

The program jumps N1000 when # 201= #202 and executes in order when #201 ≠#202.

3）Conditional jump 2 #J.NE.# K (≠)

G65 H82 Pn Q#J R# K； n：Block number

(Example) G65 H82 P1000 Q#201 R#202；

The program jumps N1000 when # 201 ≠ #202 and executes in order when #201 = #202.

4）Conditional jump 3 #J.GT.# K (＞)

G65 H83 Pn Q#J R# K； n：Block number

(Example) G65 H83 P1000 Q#201 R#202；

The program jumps N1000 when # 201 ＞ #202 and executes in order when #201 ≤ #202.

5）Conditional jump 4 #J.LT.# K (＜ ＝)

G65 H84 Pn Q#J R# K； n：Block number (example) G65 H84 P1000 Q#201 R#202；

The program jumps N1000 when # 201＜ #202 and executes in order when #201≥#202.

6）Conditional jump 5 #J.GE.# K (≥)

G65 H85 Pn Q#J R# K； n：Block number (例) G65 H85 P1000 Q#201 R#202；

The program jumps N1000 when # 201 ≤ #202 and executes in order when #201 ＜ #202.

7）Conditional jump 6 #J.LE.# K (≤)

G65 H86 Pn Q#J R# K； n：Block number

(Example) G65 H86 P1000 Q#201 R#202；

8）P/S alarm

G65 H99 Pi； i：alarm number +500

(Example) G65 H99 P15；

P/S alarm 515.

Note: Block number can be specified by variables. Such as: G65 H81 P#200 Q#201 R#202; program jump to block of its the block number specified by #200

3.13.3 PROGRAM EXAMPLE with MACRO INSTRUCTION

Example ： Automatically feed rod with system variables

Program：

```
O0001
N10 G0 X100 Z100 T101；     （Set coordinate system for tool change）
G00 X50 Z1 ；               （Rapidly position）
N20 G65 H01 P#1100 Q1；     （Start rod infeed when XS39 Pin5 output low level）
G65 H82 P20 Q#1009 R1；     （Execute N20 block when XS40 Pin21 is switched off +24V; and execute the
                             next block when XS P21 is switched on. +24V）
G65 H01 P#1100 Q0；         （Close output signal of XS39 Pin5 and stop rod infeed）
G01 X30 W-10 F300；         （Start to machine workpiece）
……
……
G01 X80 Z-50；              （End of machining）
M99 P10；                   （Execute repetitively main program and automatically feed rods）
```

Chapter 4 TOOL NOSE RADIUS COMPENSATION (G41，G42)

4.1 APPLICATION

4.1.1 Overview

Part program is compiled generally for one point of tool according to a workpiece contour. The point is regarded generally as the tool nose A point in an imaginary state(there is no the imaginary tool nose point in fact and the tool nose radius can be omitted when using the imaginary tool nose point to program) or as the center point of tool nose arc(as Fig. 3-41). Its nose of turning tool is not the imaginary point but one arc owing to the technology and other requirement in the actual machining. There is the warp between the actual cutting point and the cutting point in imaginary state, which will cause the excessive or less cutting to not to gain the perfect precision.

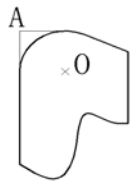
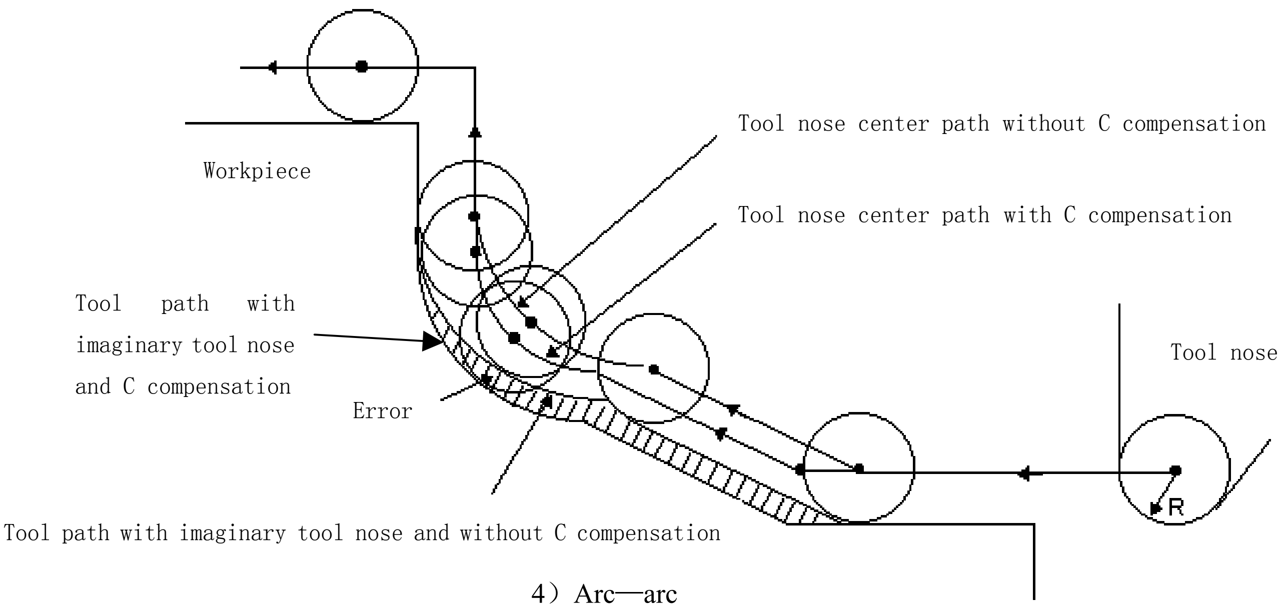


Fig. 4-1 Tool

B type tool compensation is defined that a workpiece contour path is offset one tool nose radius, which cause there is excessive cutting at a intersection of two programs because of executing motion path of next after completing the previous block.

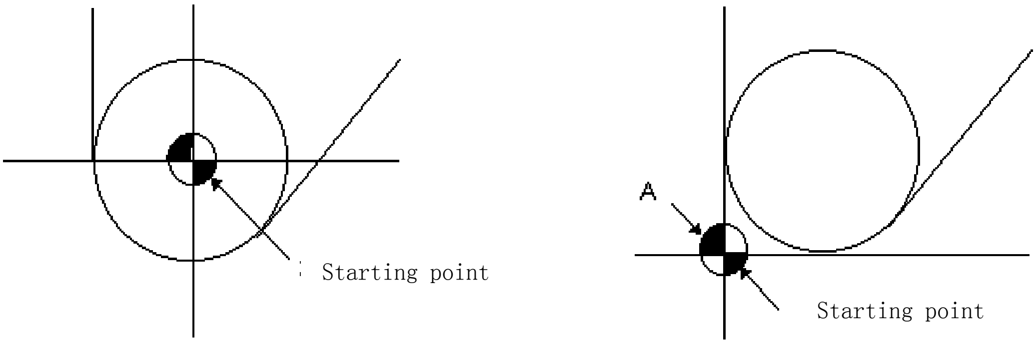
To avoid of the above-mentioned ones, the system is employed with C type tool compensation method (namely, tool nose radius compensation). The system will read next block instead of executing it immediately after reading a block in C type tool compensation method, and count corresponding motion path according to intersection of blocks. Contour can be compensated precisely because of pretreatment of reading two blocks as Fig.4-2.



4）Arc—arc

4.1.2 Imaginary Tool Nose Direction

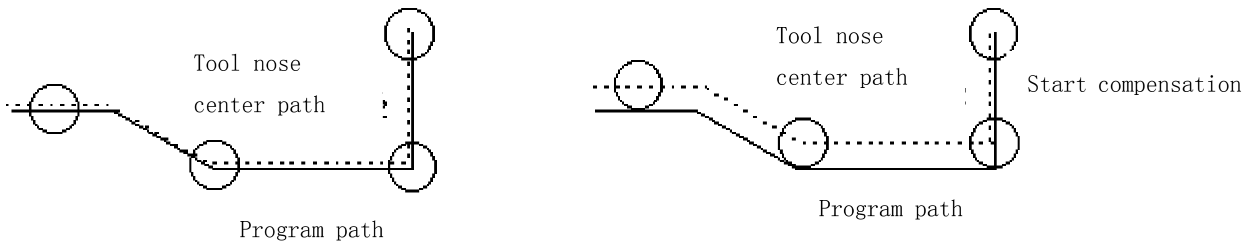
Suppose that it is generally difficult to set the tool nose radius center on the initial position as Fig. 4-3;suppose that it is easily set the tool nose on it as Fig. 4-4; The tool nose radius can be omitted in programming. Fig. 4-5 and Fig.4-6 correspond separately to the tool paths of tool nose center programming and imaginary tool nose programming when tool nose radius is executed or not.



Programming with tool nose center　　　　Programming with imaginary tool nose

Fig. 4-4

Tool nose path is the same as programming path without using tool nose radius compensation　　　　Finishing when using tool nose radius compensation



Tool nose path is the same as programming path without using tool nose radius compensation　　　　Finishing when using tool nose radius compensation

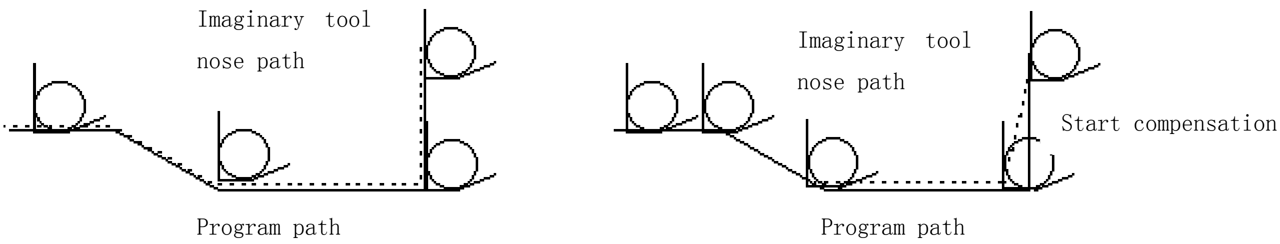
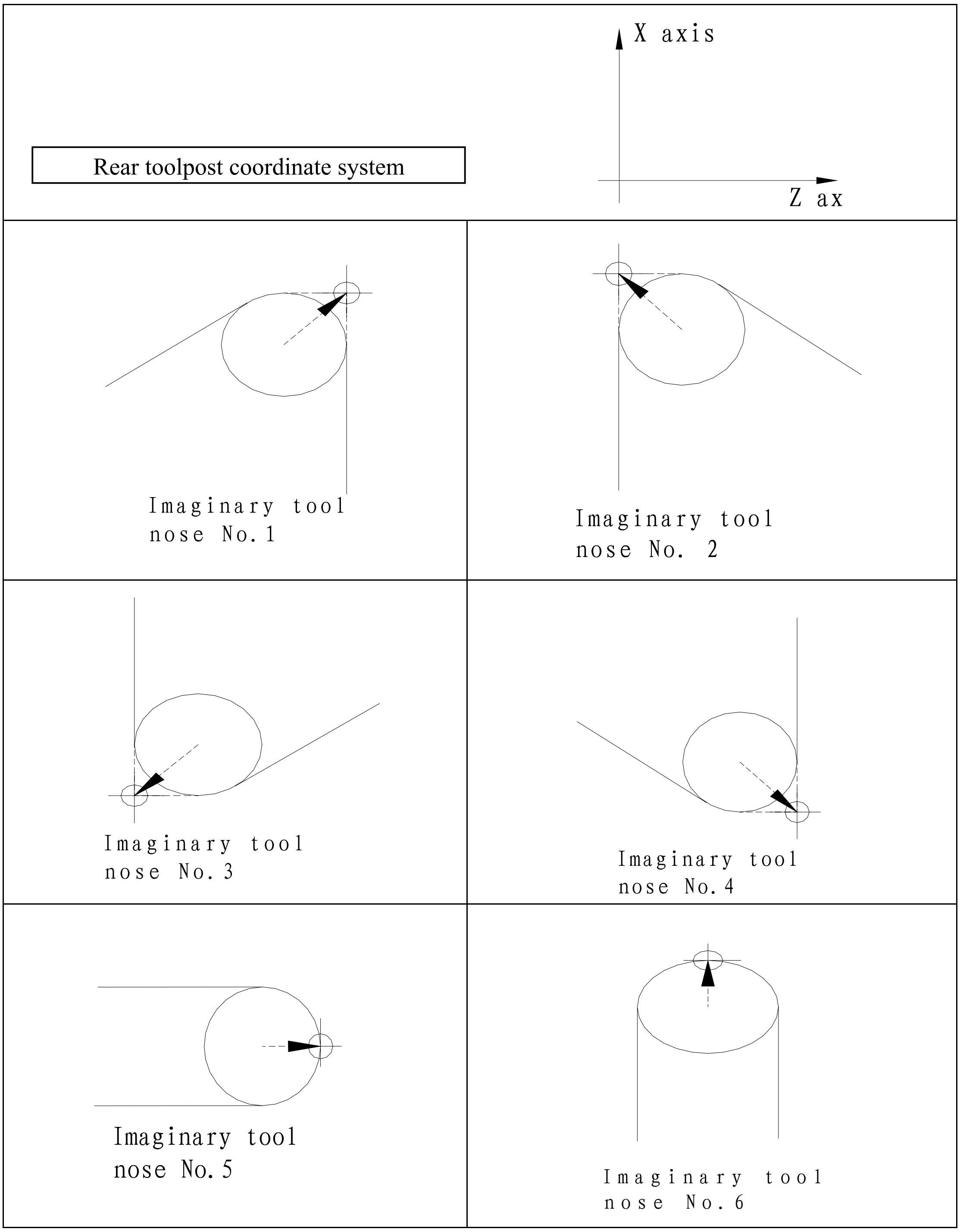


Fig. 4-6 Tool path in imaginary tool nose programming

The tool is supposed to one point in programming but the actual cutting blade is not one ideal point owing to machining technology. Because the cutting blade is not one point but one arc, machining error is caused which can be deleted by tool nose arc radius compensation. In actual machining, suppose that there are different position relationship between tool nose point and tool nose arc center point, and so it must create correct its direction of imaginary tool nose.

From tool nose center to imaginary tool nose, set imaginary tool nose numbers according to tool direction in cutting. Suppose there are 10 kinds of tool nose setting and 9 directions for position relationship. The tool nose directions are different in different coordinate system(rear toolpost coordinate system and front toolpost coordinate system) even if they are the same tool nose direction numbers as the following figures. In figures, it represents relationships between tool nose and starting point, and end point of arrowhead is the imaginary tool nose; T1～T8 in rear toolpost coordinate system is as Fig. 4-7; T1～T8 in front toolpost coordinate system is as Fig. 4-8. the tool nose center and starting point for T0 and T9 as Fig. 4-9.



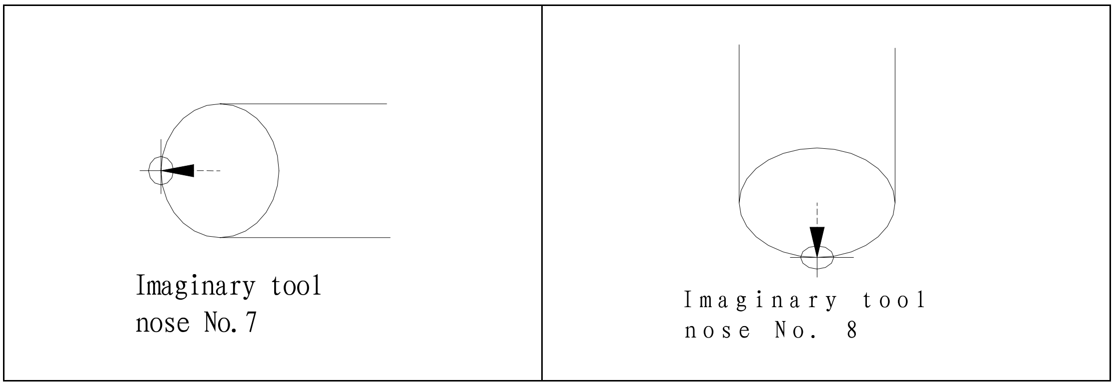
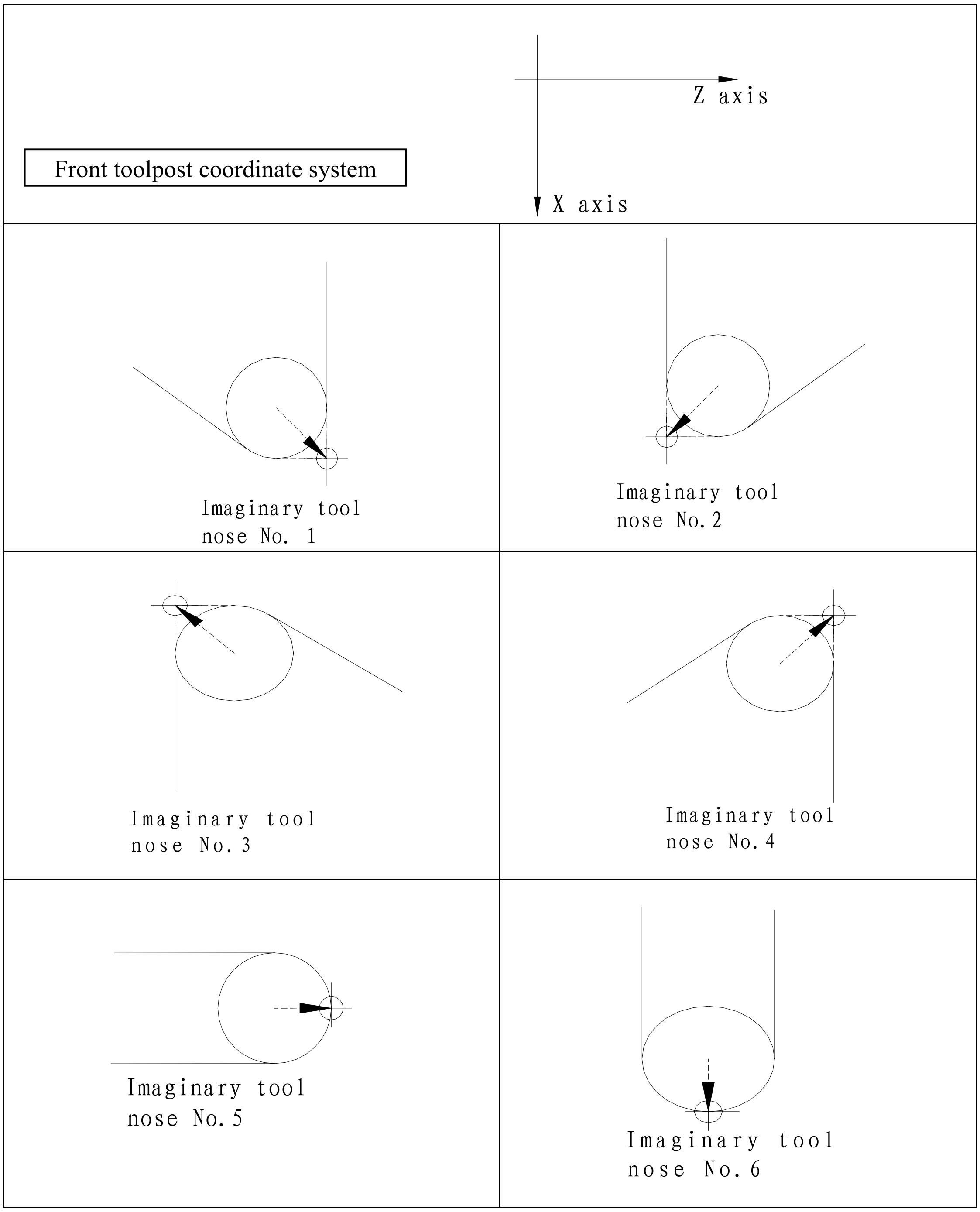


Fig. 4-7 Imaginary tool nose number in rear toolpost coordinate system



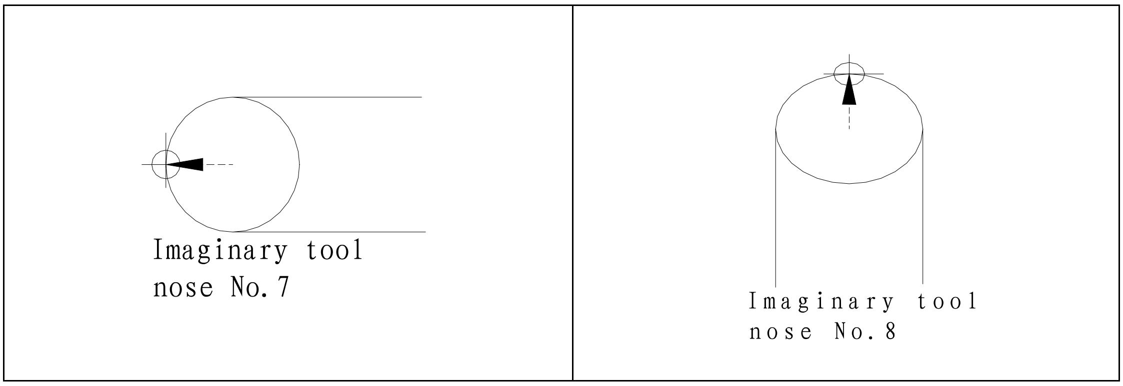


Fig. 4-8 Imaginary tool nose number in front toolpost coordinate system

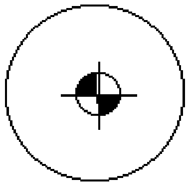


Fig. 4-9   Tool nose center on starting point

4.1.3   Compensation Value Setting

Preset imaginary tool nose number and tool nose radius value for each tool before executing tool nose radius compensation. Set the tool nose radius compensation value in ***"Offset"*** window (as Fig. 4-1), R is tool nose radius compensation value and T is imaginary tool nose number.

Table 4-1 Display window of system tool nose radius compensation value

Number	**X**	**Z**	**R**	**T**
000	0.000	0.000	0.000	0
001	0.020	0.030	0.020	2
002	1.020	20.123	0.180	3
…	…	…	…	…
032	0.050	0.038	0.300	6

Note: Tool offset value can be specified in diameter or radius in X direction, set by No.004 Bit4 ORC, offset value is in radius when ORC=1 and is in diameter when ORC=0.

In toolsetting, the tool nose is also imaginary tool nose point of Tn (n=0~9) when taking Tn(n=0~9) as imaginary tool nose. For the same tool, offset value from standard point to tool nose radius center(imaginary tool nose is T3) is different with that of ones from standard point to imaginary tool nose(imaginary tool nose is T3) when T0 and T3 tool nose points are selected to toolsetting in rear toolpost coordinate system, taking toolpost center as standard point. It is easier to measure distances from the standard point to the tool nose radius center than from the standard point to the imaginary tool nose, and so set the tool offset value by measuring distance from the standard point to the imaginary tool nose(tool nose direction of T3).

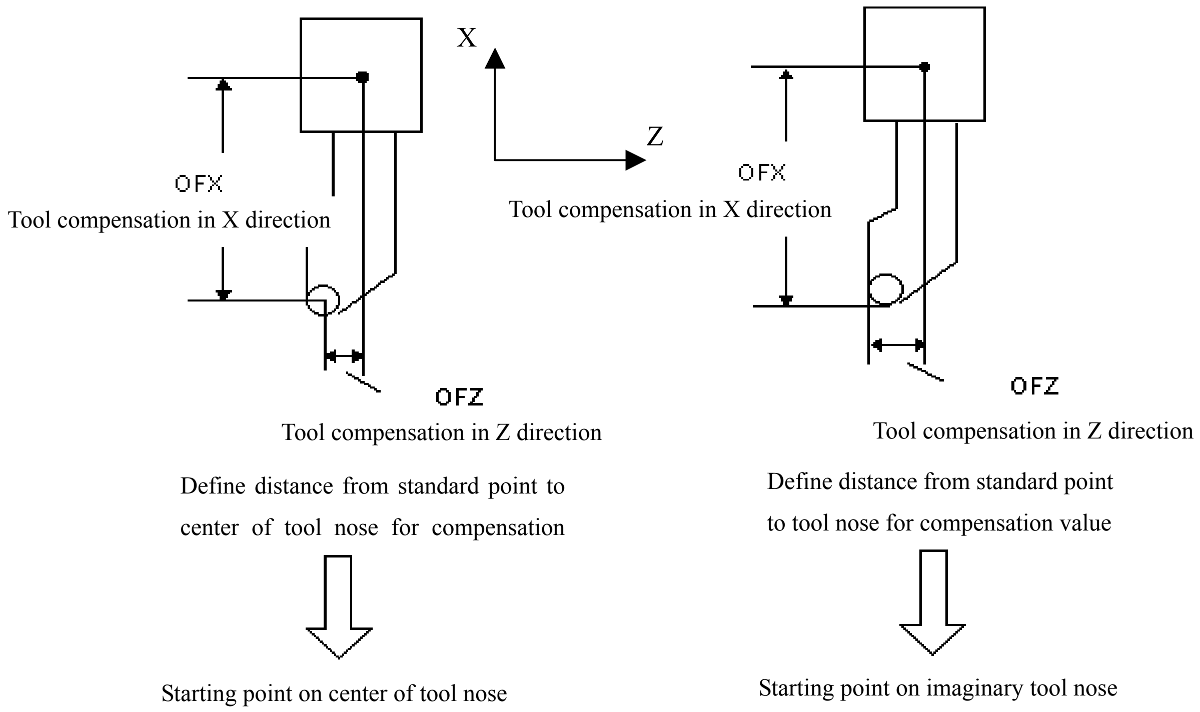


Fig. 4-10 Tool offset value of toolpost center as benchmark

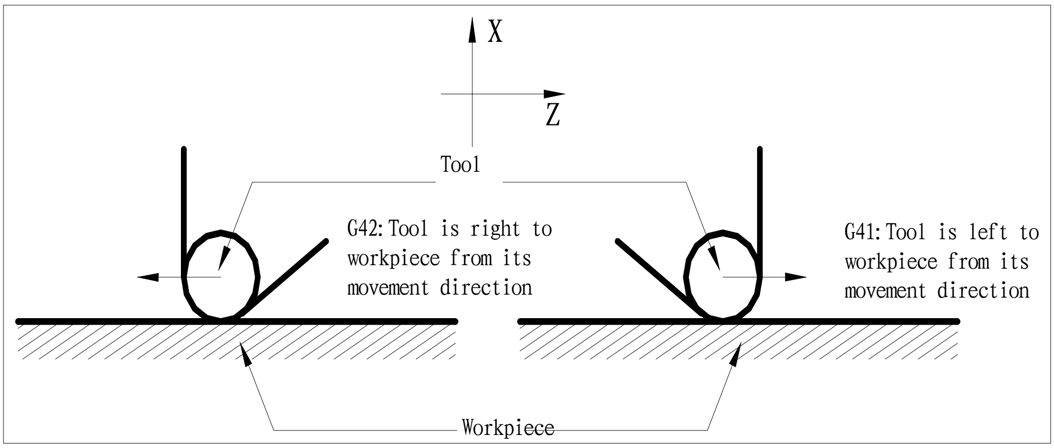
4.1.4 Instruction Format

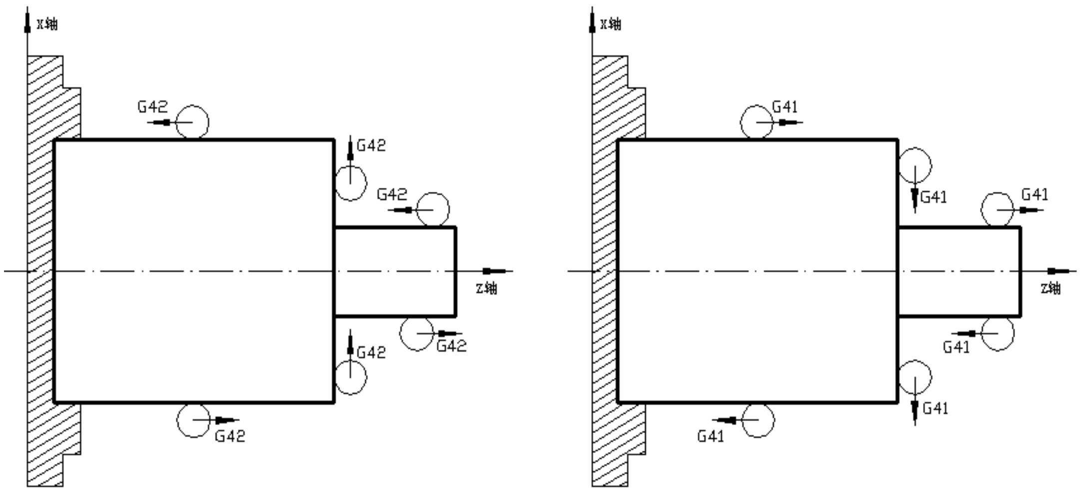
$$\begin{Bmatrix} G40 \\ G41 \\ G42 \end{Bmatrix} \begin{Bmatrix} G00 \\ G01 \end{Bmatrix} \quad X\_\_ \quad Z\_\_ \quad T\_\_ \;;$$

Instructions	Function specifications	Remark
G40	Cancel the tool nose radius compensation	See Fig.4-11 and 4-12
G41	Tool nose radius left compensation is specified by G41 in rear toolpost coordinate system and tool nose radius right compensation is specified by G41 in front toolpost coordinate system	
G42	Tool nose radius right compensation is specified by G42 in rear toolpost coordinate system and tool nose radius left compensation is specified by G42 in front toolpost coordinate system	

4.1.5 Compensation Direction

Specify its direction according to relative position between tool nose and workpiece when executing tool nose radius compensation as Fig. 4-11 and Fig.4-12.


X
Z
Tool
G42: Tool is right to workpiece from its movement direction
G41: Tool is left to workpiece from its movement direction
Workpiece


X轴
G42
G42
G42
G42
G42
G42
Z轴
X轴
G41
G41
G41
G41
G41
G41
Z轴

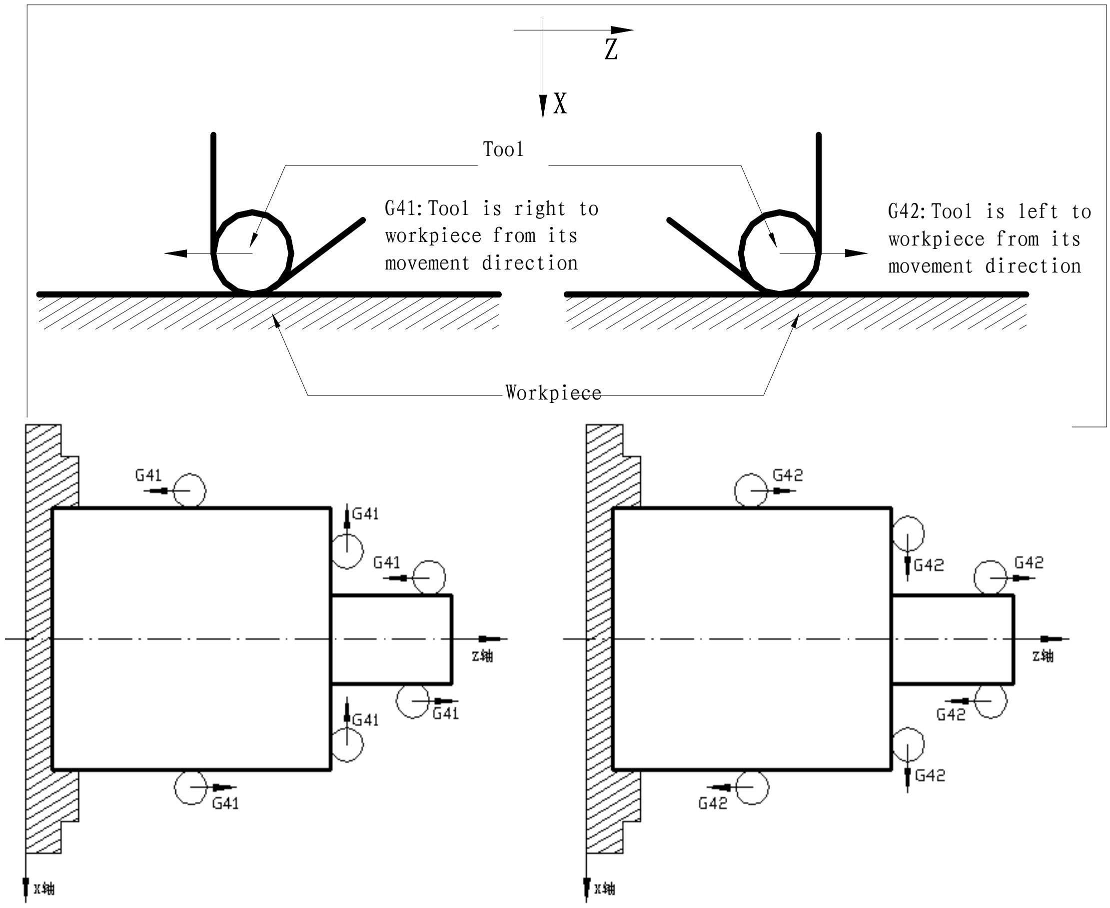


Fig. 4-12 Compensation direction of front coordinate system

4.1.6 Cautious

- The system is in tool nose radius compensation mode at initial state, and starts to create tool nose radius compensation offset mode when executing G41 or G42. When the system starts to execute compensation, it pre-read two blocks, and the next block is saved to storage for tool nose radius compensation when executing one of them. The system reads two blocks in ***“Single”*** mode and stops after executing end point of the first block.
- In tool nose radius compensation mode, the tool nose center moves to end point of previous block and is vertical to its path when the system executes two block or more than blocks without motion instruction.
- The system cannot create and cancel tool nose radius compensation
- Tool nose radius R is without negative value, otherwise there is a mistake running path.
- Tool nose radius compensation is created and cancelled in G00 or G01 instead of G02 or G03, otherwise, the system alarms.
- The system cancels the tool nose radius compensation mode when pressing ***“PESET”*** key.
- G40 must be specified to cancel offset mode before the program is ended, otherwise the tool path offsets one tool nose radius.
- The system executes the tool nose radius compensation in main program and subprogram but must cancel it before calling subprogram and then create it again in the subprogram.
- The system does not execute the tool nose radius compensation in G71, G72, G73, G74, G75, G76 and cancel it temporarily.

- The system executes the tool nose radius compensation in G90, G94, it offsets one tool nose radius for G41 or G42.

4.1.7 Application

Machine a workpiece in the front toolpost coordinate system as Fig. 3-54. Tool number: T0101, tool nose radius R=2, imaginary tool nose number T=3.

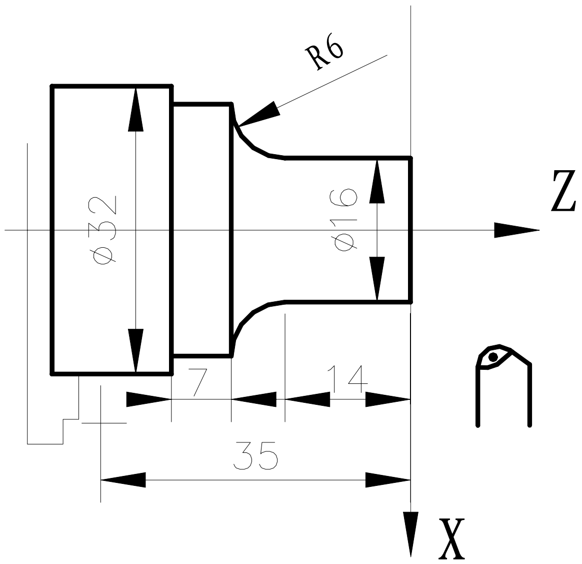


Fig. 3-54

For toolsetting in Offset Cancel mode, after toolsetting, Z axis offsets one tool nose radius and its direction is relative to that of imaginary tool nose and toolsetting point, otherwise the system excessively cuts tool nose radius when it starts to cut.

Set the tool nose radius R and imaginary tool nose direction in "***Offset***" window as Fig.

Table 3-7

Number	X	Z	R	T
001			2.000	3
002	…	…	…	…
…	…	…	…	…
007	…	…	…	…
008	…	…	…	…

Program:

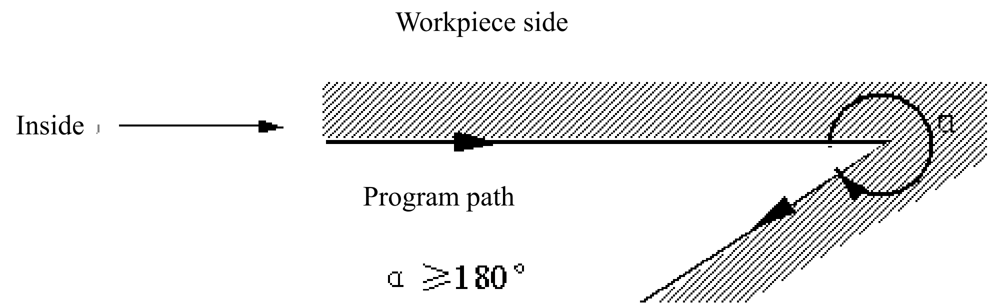
G00 X100 Z50 M3 T0101 S600；	（Position, start spindle, tool change and execute tool compensation）
G42 G00 X0 Z3；	（Set tool nose radius compensation）
G01 Z0 F300；	（Start cutting）
X16；	
Z-14 F200；	
G02 X28 W-6 R6；	

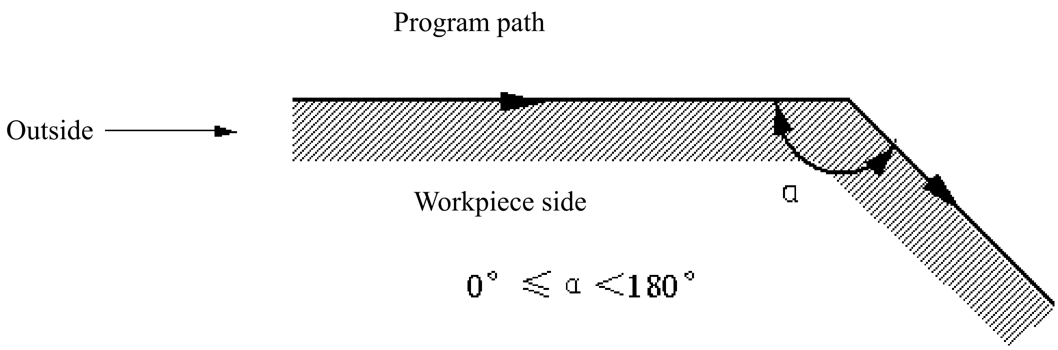
G01 W-7;
X32;
Z-35;
G40 G00 X90 Z40;　　　　　　（Cancel tool nose radius compensation）
G00 X100 Z50 T0100;
M30;

4.2 TOOL NOSE RADIUS COMPENSATION OFFSET PATH

4.2.1 Inner and Outer Side

Inside is defined that an angle at intersection of two motion blocks is more than or equal to 180°; **Outside** is 0~180°.





4.2.2 Tool Traverses when Starting Tool

3 steps to execute tool nose radius compensation: tool compensation creation, tool compensation execution and tool compensation canceling.

Tool traverse is called tool compensation creation (starting tool) from offset canceling to G41 or G42 execution.

Note: Meanings of S, L, C in the following figures are as follows:
S——Stop point of single block; L——straight line; C——arc。

（a）Tool traversing inside along corner（α ≥180°）

1）Straight line —straight line

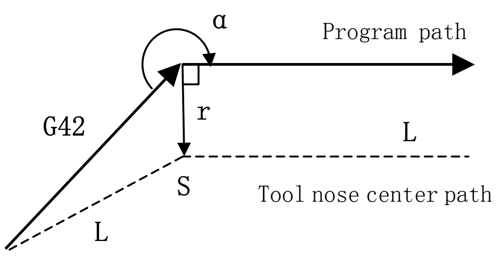


Fig.2-1a Straight line —straight line（starting tool inside）

2）Straight line —arc

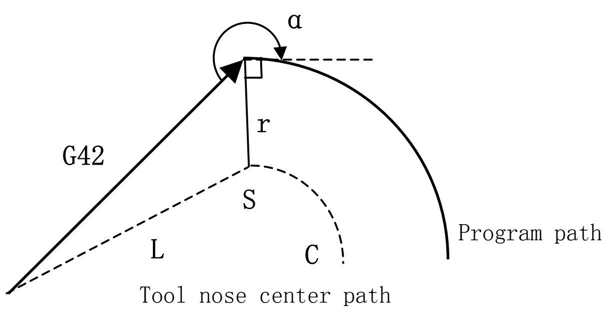


Fig. 2-1b Straight line —arc（starting tool inside）

（b）Tool traversing inside along corner（180° ＞α ≥90° ）

1）Straight line —straight line

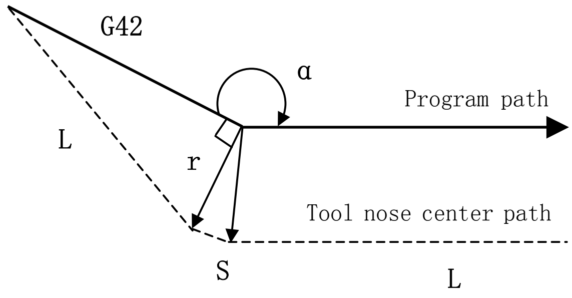


Fig.2-2a Straight line —straight line（starting tool outside）

2）Straight line—arc

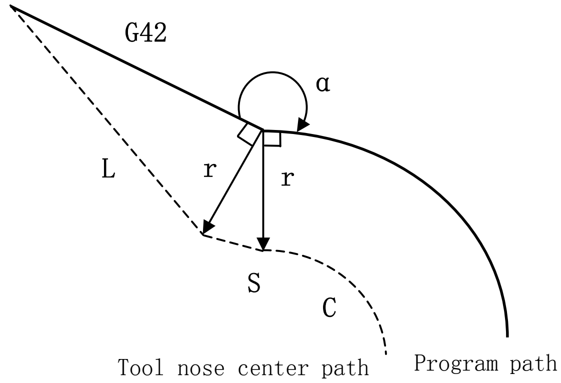


Fig.2-2b Straight line—arc（starting tool outside）

（c）Tool traversing inside along corner （α ＜90°)

1）Straight line —straight line

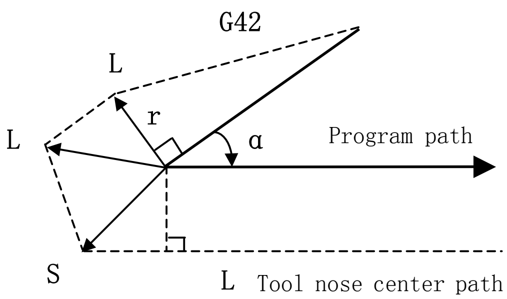


Fig.2-3a Straight line —straight line（starting tool outside）

2）Straight line—arc

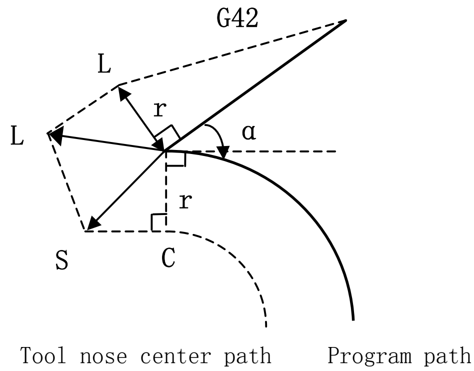


Fig. 2-3b straight line—arc（starting tool outside）

（d）Tool traversing inside along corner（α ≦1° ）,straight line →straight line

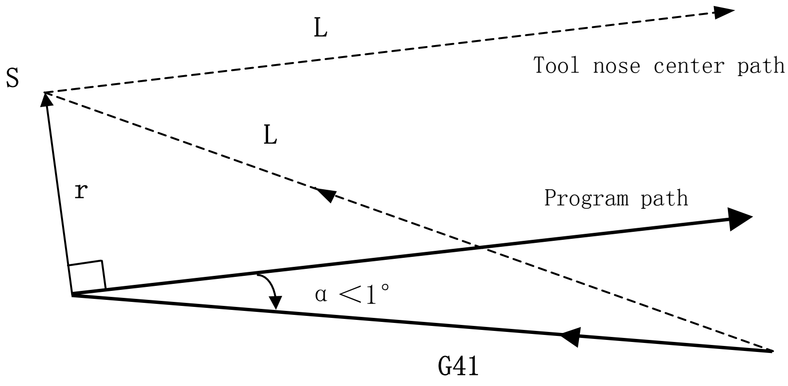


Fig. 2-4a Straight line—straight line（α＜1° , starting tool outside）

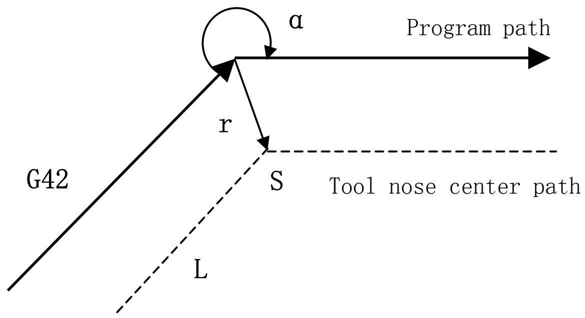
4.2.3 Tool Traversing in Offset Mode

Offset mode is called to ones after creating tool nose radius compensation and before canceling it.

- Offset path without changing compensation direction in compensation mode

（a）Tool traversing inside along corner（α ≥180°）

1）Straight line —straight line



Fi.g2-5a Straight line —straight line (moving inside)

2）Straight line—arc

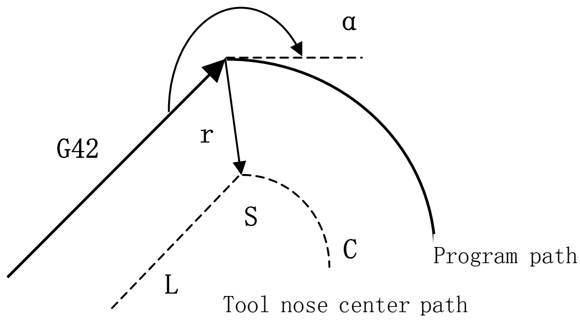


Fig.2-5b Straight line—arc moving inside)

3）Arc—straight line

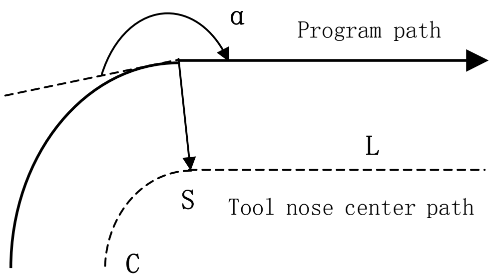


Fig. 2-5c Arc—straight line (moving inside)

4）Arc—arc

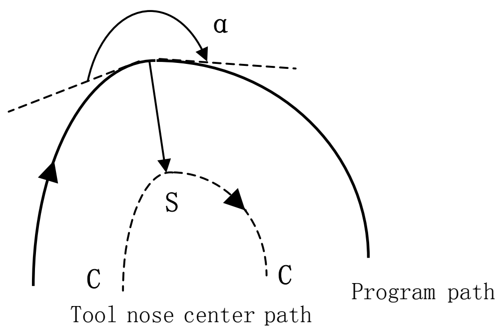


Fig. 2-5d Arc—arc (moving inside)

5）Machining inside (α＜1°) and zoom out the compensation vector

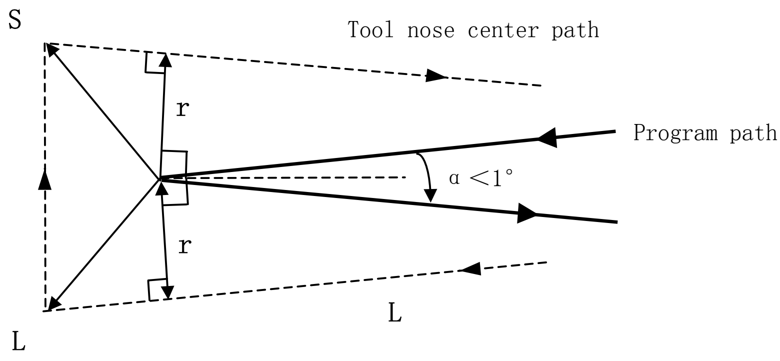


Fig. 2-5e Straight line —straight line (α＜1° , moving inside)

（b）Tool traversing outside along corner（180° ＞α ≥90° ）

1）Straight line —straight line

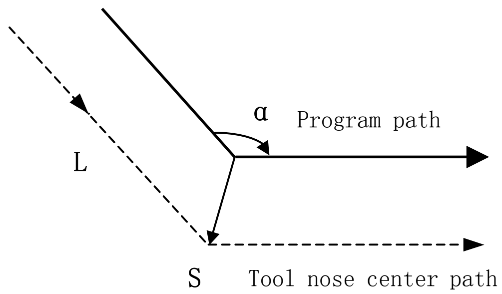


Fig. 2-6a Straight line —straight line
（180° ＞α ≥90° , moving outside）

2）Straight line—arc

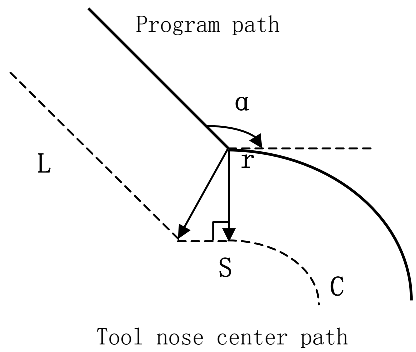


Fig. 2-6b Straight line—arc
（180° ＞α ≥90° , moving outside）

3）Straight line —straight line

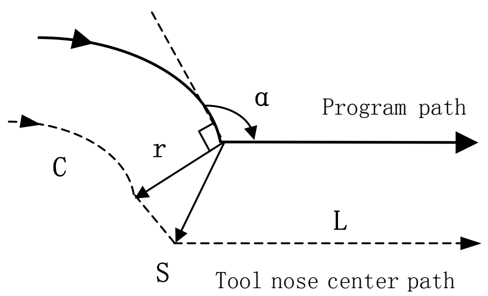


Fig. 2-6c Arc—straight line
（180° ＞α ≥90° moving outside）

4）Arc—arc

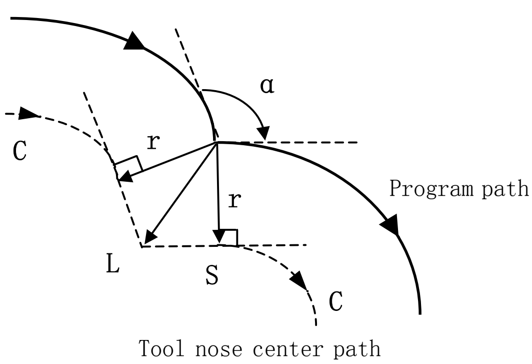


Fig. 2-6c Arc—arc
（180° ＞α ≥90° moving outside）

（c）Tool traversing outside along corner（α＜90°）

1）Straight line—straight line

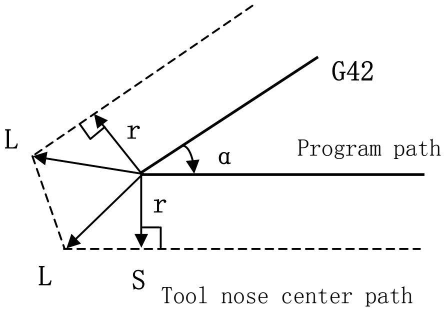


Fig.2-7a Straight line—Straight line
（α＜90°, moving outside）

2）Straight line—arc

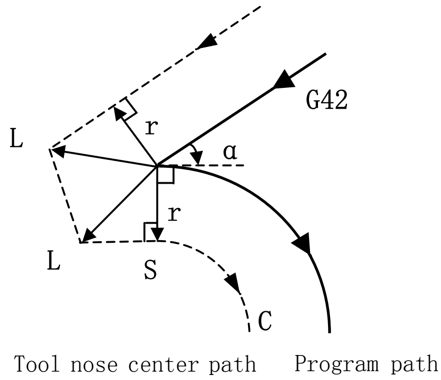


Fig.2-7b Straight line—arc
（α＜90°, moving outside）

3）Arc—straight line

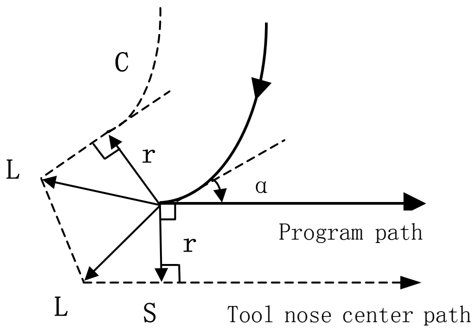


Fig.2-7c Arc—straight line
（α＜90°, moving outside）

4）Arc—arc

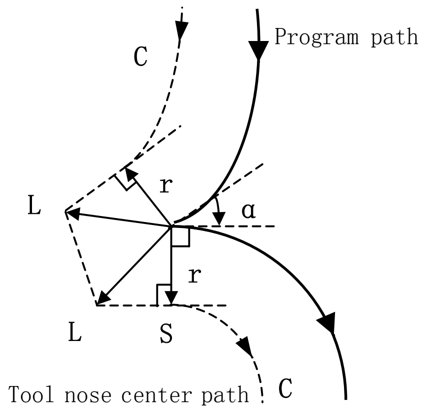
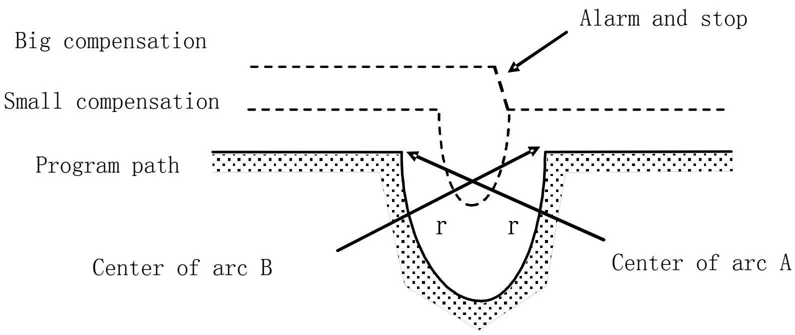


Fig.2-7c Arc—arc
（α＜90°,moving outside）

(d) Special cutting

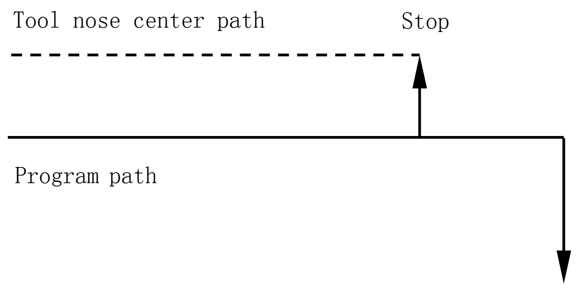
1）Without intersection



There is no intersection of compensation paths when the tool radius is small; no one when the radius is big and the tool stops at the end point of previous block and the system alarms.

Fig. 2-8　Paths without intersection after offset

2）Center point and starting point of arc being the same one



Alarm: Starting point or end point of arc is the same as center point; they are end point of previous block

(G41)

B56 G01 W20;

N6 G02 W10 I K0;

N7 G03 U-10 I-10;

Fig. 2-9 Center point and starting point of arc being the same one

- **Offset path of compensation direction in compensation mode**

The compensation direction of tool nose radius is specified by G41 and G42 and the sign symbol is as follows:

Table 2-1

Sign symbol of compensation value / G instruction	+	-
G41	Left compensation	Right compensation
G42	Right compensation	Left compensation

The compensation direction can be changed in compensation mode in special cutting, it cannot be changed at starting block and its following one. There is no inside and outside cutting when the system changes the compensation direction. The following compensation value is supposed to be positive.

1）Straight line—straight line

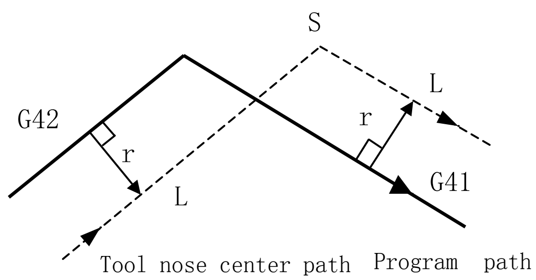


Fig. 2-10 straight line—straight line (changing compensation direction)

2）Straight line —arc

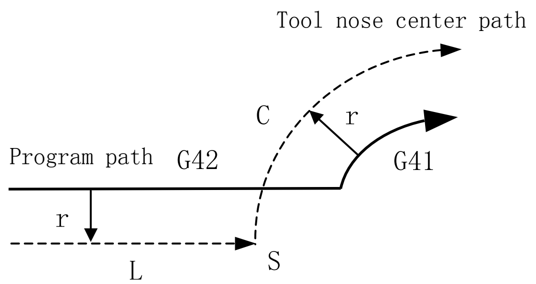


Fig. 2-10 straight line—arc (changing compensation direction)

3) Arc—straight line

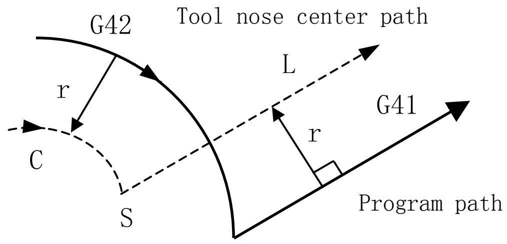


Fig. 2-12 Arc—straight line
(changing compensation direction)

4) Arc—arc

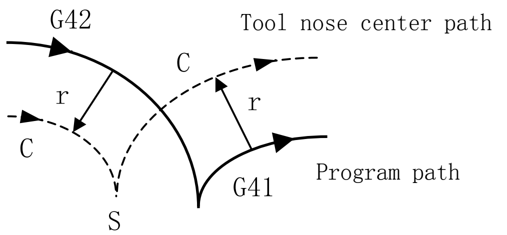


Fig. 2-13 Arc—arc
(changing compensation direction)

5) Without intersection when compensation is executed normally

When the system executes G41 and G42 to change the offset direction between block A and B, a vector perpendicular to block B is created from its starting point.

i) Straight line----Straight line

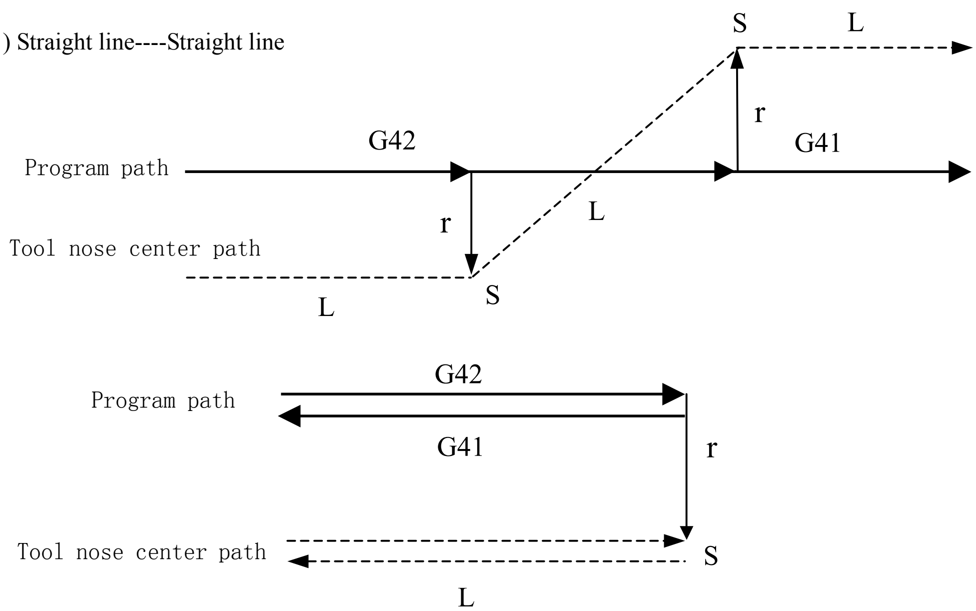


Fig. 2-14a Straight line—straight line without intersection (changing compensation direction)

ii) Straight line ---arc

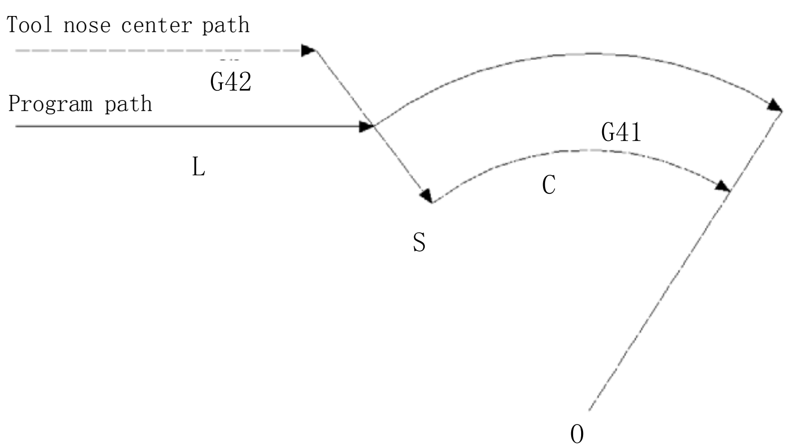


Fig. 2-14b Straight line—arc without intersection (changing compensation direction)

iii) Arc-----arc

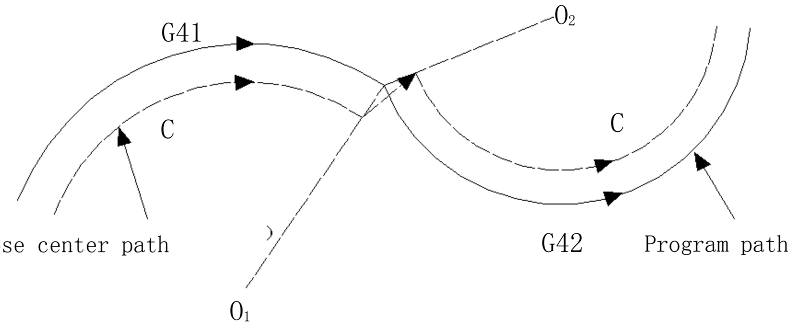


Fig. 2-14c Arc—arc without intersection（changing compensation direction）

4.2.4 Tool Traversing in Offset Canceling Mode

In compensation mode, when the system executes a block with one of the followings, it enters compensation canceling mode, which is defined to compensation canceling of block

1. Execute G40 in a program;
2. Execute M30.

The system cannot execute G02 and G03 when canceling C tool compensation (tool nose radius compensation), otherwise the system alarms and stops run.

In compensation canceling mode, the system executes the block and ones in the register for tool nose radius compensation. At the moment, the run stops after executing one block when single block is ON. The system executes the next one but does not read its following one when pressing "*Start*" button again.

When the system is still in compensation canceling mode, the next block without others to be executed is read into buffer register for tool nose radius compensation.

（a）Tool traversing inside along corner（α ≥**180**° ）

1）Straight line → Straight line

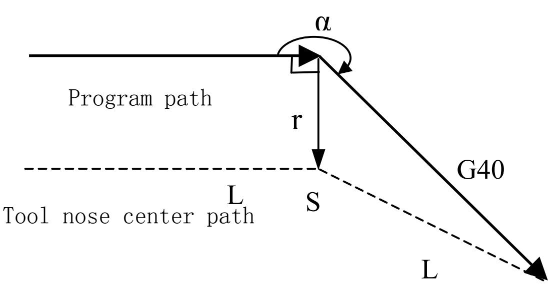


Fig. 2-15a Arc-straight line（moving inside and canceling offset）

2）Arc → Straight line

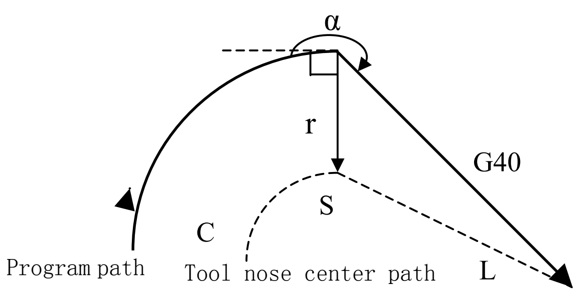
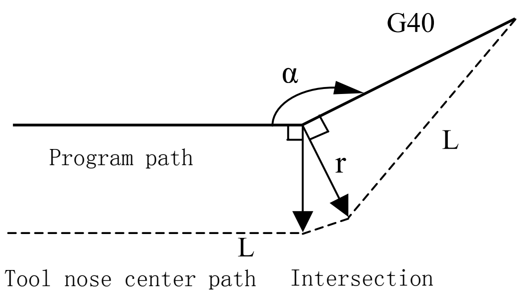


Fig. 2-15b Arc-straight line（moving inside and canceling offset）

(b) Tool traversing outside along corner (180° ＞α ≥90°)

1) Straight line →straight line



2) Arc →straight line

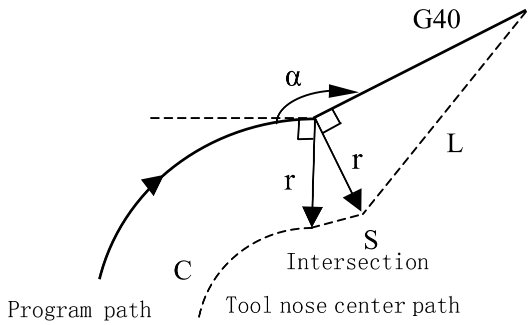
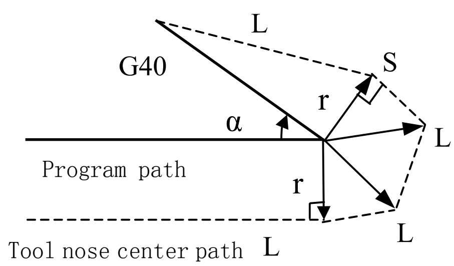


Fig. 2-16a Arc—straight line (α ≥90° moving outside and canceling offset)

Fig. 2-16b Arc—straight line (α ≥90° moving outside and canceling offset)

(c) Tool traversing outside along corner (α ＜90°)

1) Straight line→straight line



2) Arc → straight line

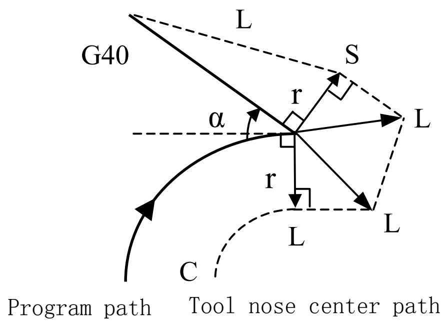


Fig. 2-17a Straight line—straight line (α ＜90° cutting outside and canceling offset)

Fig. 2-17a Straight line—straight line(α ＜90° cutting outside and canceling offset)

(d) Tool traversing outside along corner (α ＜1°);straight line→straight line

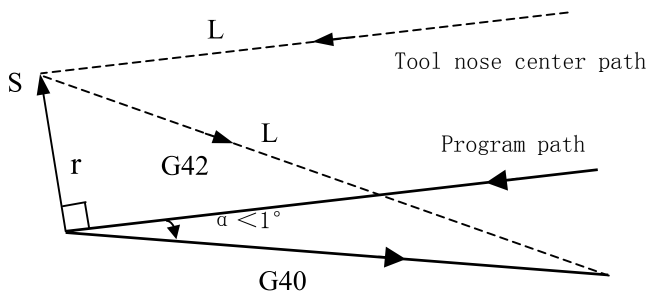


Fig. 2-18 Straight line—straight line (α ＜1° cutting outside and canceling offset)

4.2.5 Tool Interference Check

"Interference" is defined that the tool cuts workpiece excessively and it can find out excessive cutting in advance, the interference check is executed even if the excessive cutting is not created, but the system cannot find out all tool interferences.

(1) Fundamental conditions

1) The tool path direction is different that of program path(angle is 90°~270°).

2) There is a big difference (α >180°)for two angles between starting point and end point of tool nose center path, and between starting point and end point of program path.

Example: linear machining

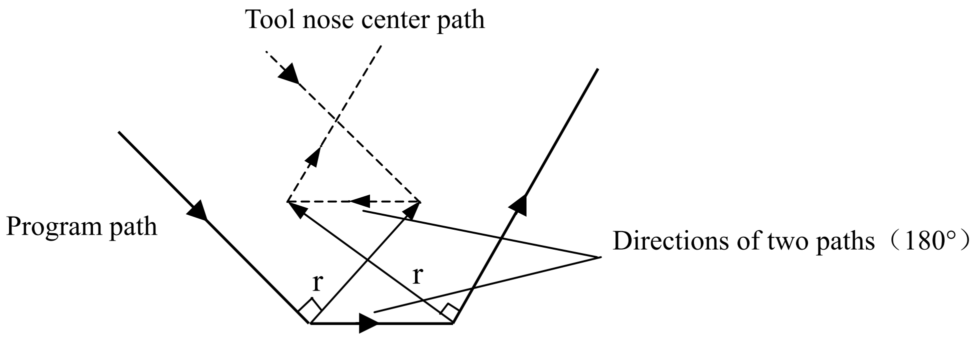


Fig. 2-19a Machining interference （1）

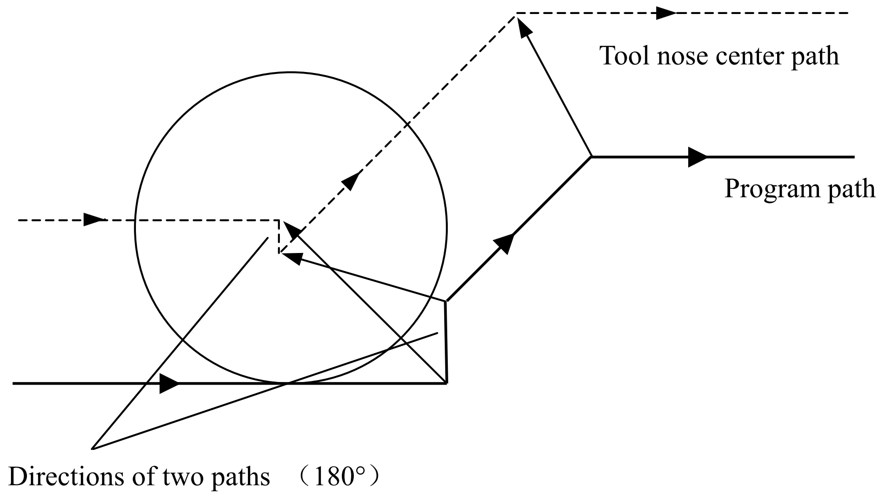


Fig. 2-19b Machining interference （2）

(2) Executing it without actual interference

1) Concave groove less than compensation value

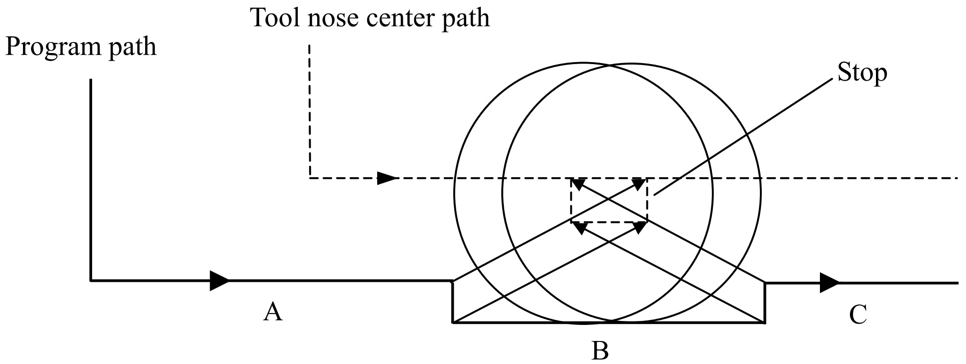


Fig. 2-21 Executing interference（1）

Directions of block B and tool nose radius compensation path are opposite without interference, the tools stops and the system alarms.

2) Concave channel less than compensation value

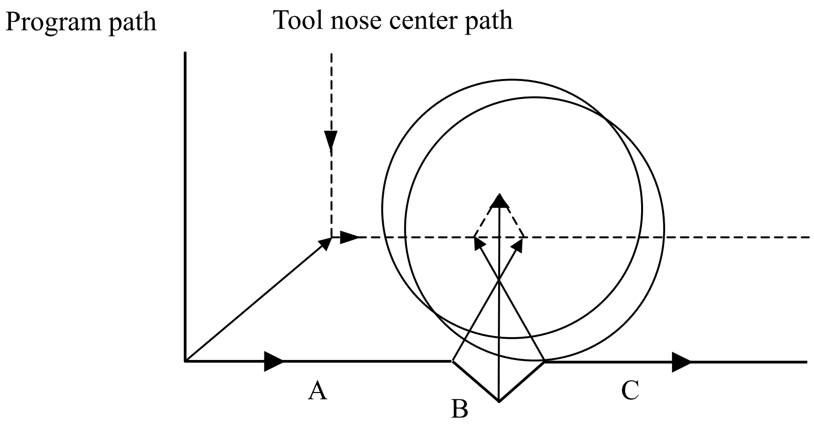


Fig. 2-22 Executing interference（2）

Directions of block B and tool nose radius compensation path are opposite without interference, the tools stops and the system alarms.

4.2.6 Instructions for Canceling Compensation Vector Temperarily

In compensation mode, the compensation vector is cancelled temporarily in G50, G71~G76 and is automatically resumed after executing the instructions. At the moment, the compensation is cancelled temporarily and the tool directly moves from intersection to a point for canceling compensation vector. The tool directly moves again to the intersection after the compensation mode is resumed.

- **Setting coordinate system in G50**

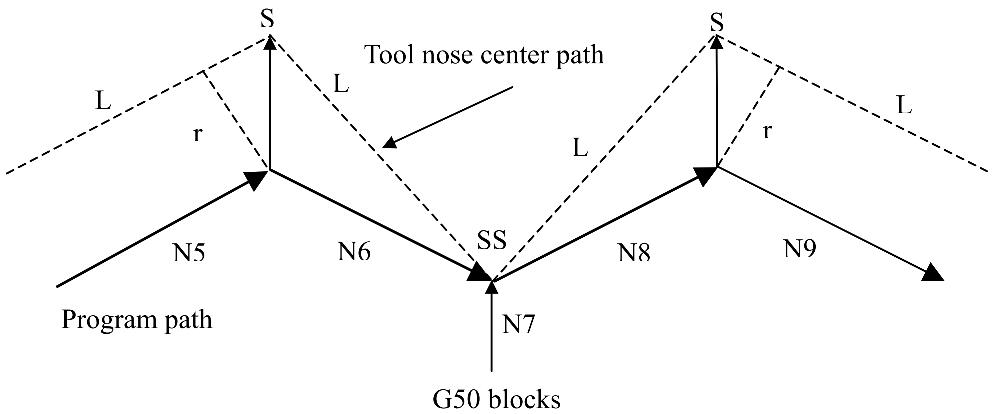


Fig. 3-1 Cancel compensation vector temporarily in G50

Note: The tool stops at S position in "Single Block" mode.

- **Reference point automatic return in G28**

In compensation mode, the compensation is cancelled in a middle point and is automatically resumed after executing the reference point return in G28.

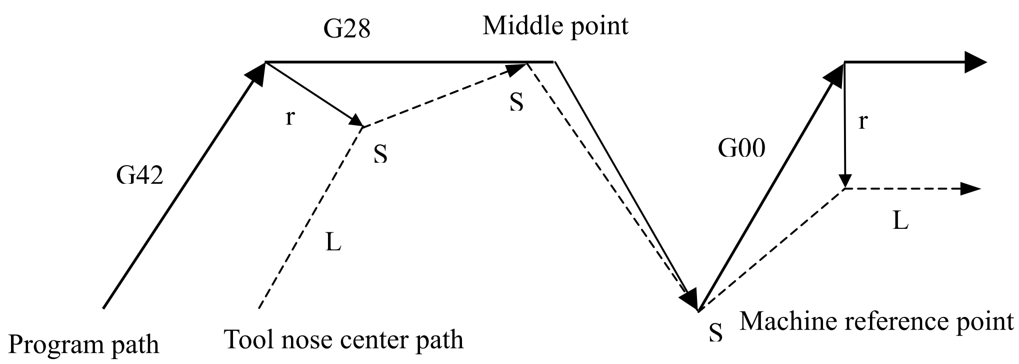


Fig. 4-23 Cancel compensation vector temporarily in G28

- **G71～G76 compound cycle; G32, G33, G34 thread cutting**

When executing G71～G76 , G32, G33, G34 , the system does not execute the tool nose radius compensation and cancel it temporarily, and executes it in the next blocks of G00, G01, G02, G03, G70.

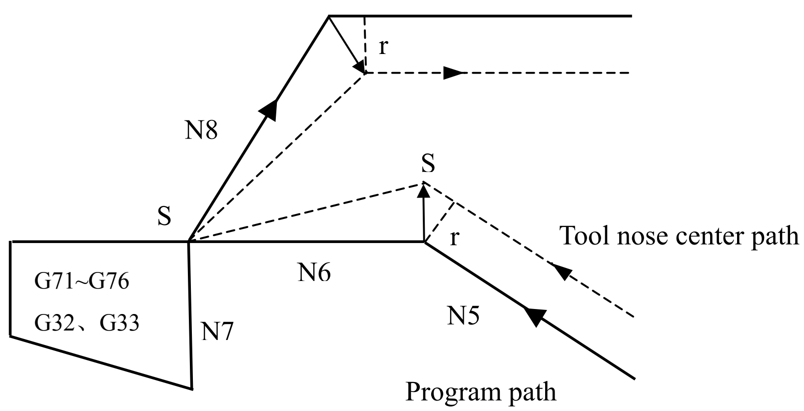


Fig. 4-23 Cancel compensation vector temporarily in G71～G76

- **G90, G94**

Compensation method of tool nose radius compensation in G90 or G94:

A. Each cycle path and tool nose center path are parallel to program path.

B. Offset directions are the same in G41 and G42 as the following figure.

C. When the system compensates the imaginary tool nose direction NO. 0, the motion path offsets a tool nose radius vector, and the system does not count any intersections in cycle.

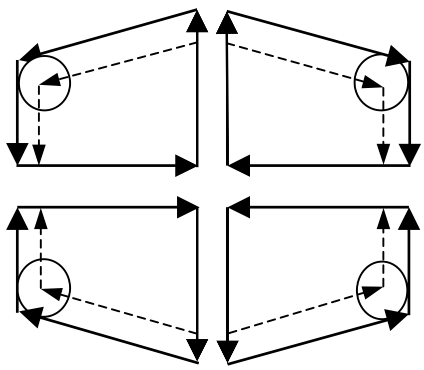


Fig. 3-3 Offset direction of tool nose radius compensation in G90

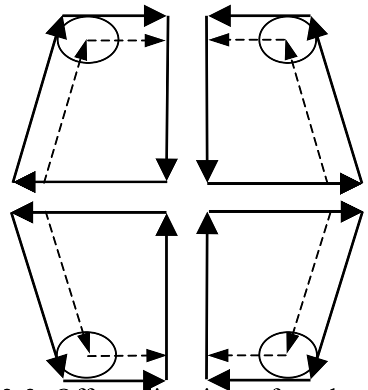


Fig.3-3 Offset direction of tool nose radius compensation in G94

4.2.7 Particular

- **Inside chamfer machining less than tool nose radius**

At the moment, the tool inside offset causes an excessive cutting. The tool stops and the system alarms (P/S41) when starting the previous block or chamfer moving. But the tool stops the end point of previous block when '**Single Block**' is ON.

- **3.6 Machining concave less than tool nose diameter**

There is an excessive cutting when the tool nose center path is opposite to program path caused by tool nose radius compensation. At the moment, the tool stops and the system alarms when starting the previous block or chamfer moving.

- **3.7 Machining sidestep less than tool nose radius**

The tool center path can be opposite to program path when the sidestep is less than tool nose radius and is an arc in program. At the moment, the system automatically ignores the first vector and directly moves end point of second vector linearly. The program stops at the end point in single block and otherwise the cycle machining is continuously executed. If the sidestep is a straight line, compensation is executed correctly and the system does not alarm(but the not-cutting is still reserved).

- **3.8 Subprograms in G instructions**

The system must be in canceling compensation mode before calling subprograms. After calling subprograms, the offset is executed and the system must be in canceling compensation mode before returning to main programs, otherwise the system alarms.

- **3.9 Changing compensation value**

(a) Change compensation value in canceling tool change mode. New compensation value is valid after tool change when the compensation value is changed in compensation mode.

(b) Compensation value sign symbol and tool nose center path

G41 and G42 are exchanged each other if the compensation value is negative(-). The tool moves along inside when its center moves along outside of workpiece, and vice versa.

Generally, the compensation value is positive(+) in programming. The compensation value is negative(-) when the tool path is as the above-mentioned (a), and vice versa.

Besides, direction of tool nose offset changes when offset value sign symbol is changed, but we suppose the direction of tool nose is not changed. Generally, the offset value sign symbol is not changed.

- **3.10 End point of programming arc out of arc**

The tool stops and the system alarms and displays “End point of arc is not on arc” when the end point of arc is not on arc in programs.

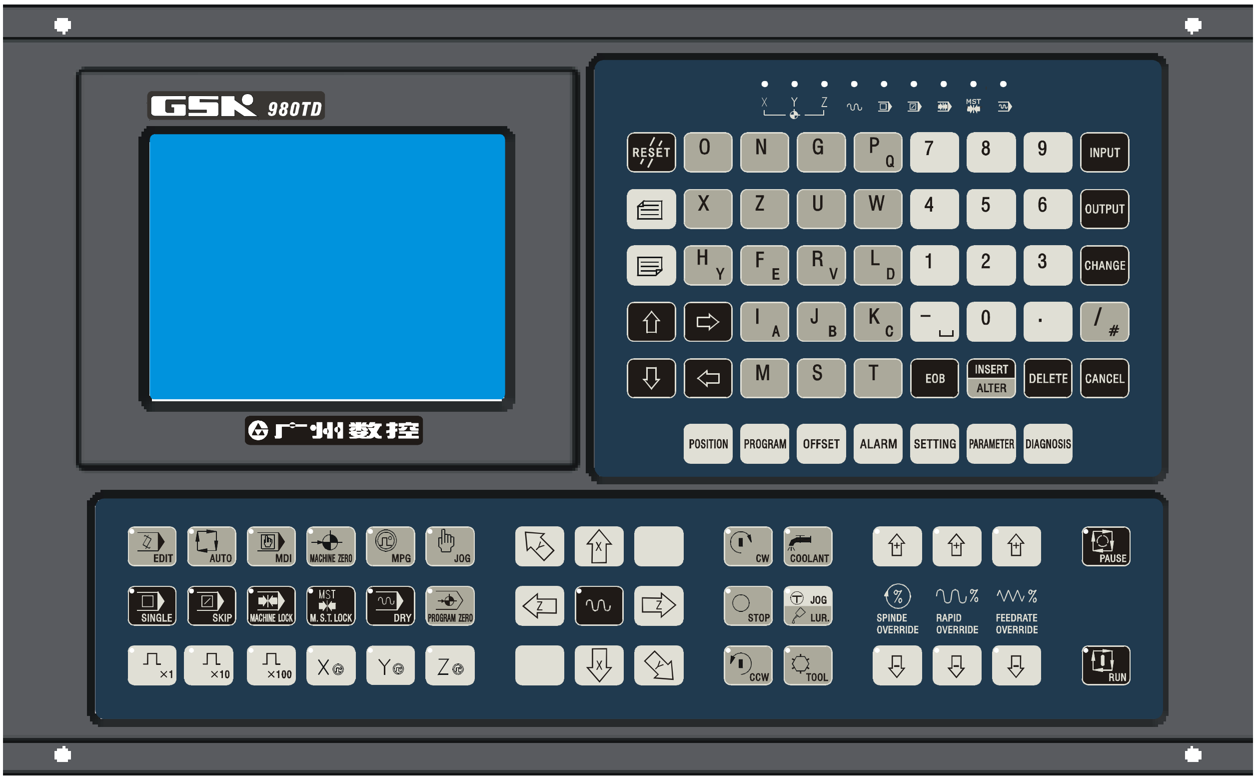
BOOK 2

OPERATION

Book2 operation

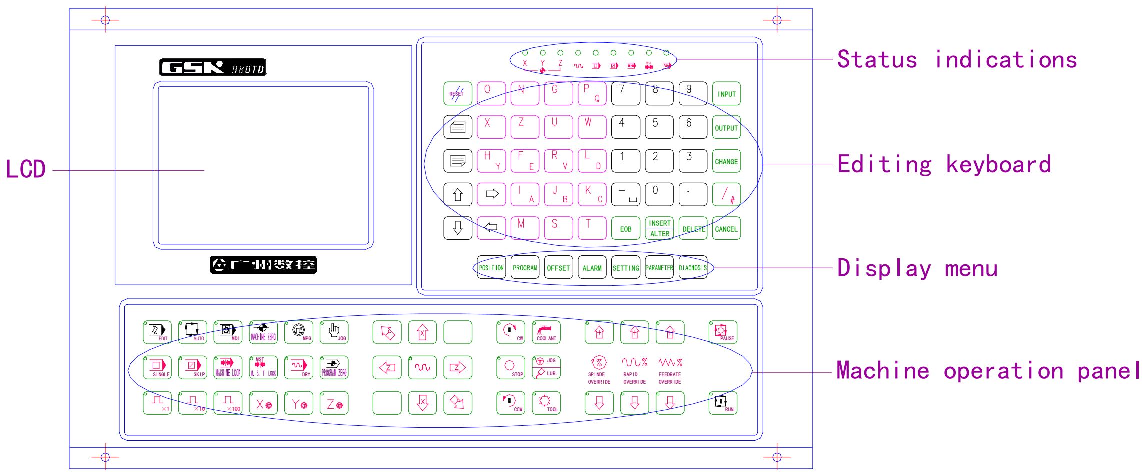
Chapter 1 OPERATION and DISPLAY

The operation panel of GSK980TD is made of aluminium alloy as below:



1.1 PANEL COMPARTMENT

GSK980TD adopts integrated operation panel, and it is compartmentalized as follows:



1.1.1 Status Indications

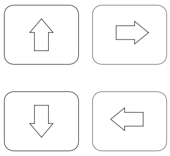
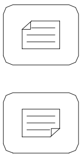
X Y Z	X, Z return completion LED		Rapid traverse LED

Book2 operation

	Single run LED		Single block LED
	Machine lock LED	MST	Auxiliary functions lock LED
	Trial run LED		

1.1.2 Editing Keyboard

Keys	Name	Description
RESET	Reset key	CNC reset, stop of the feeding and moving, etc.
O N G X Z U W M S T	Address key	Address input
H Y, F E, R V, L D, I A, J B, K C		Double-address key, switch between two addresses by repeated press
– ␣ , / #	Symbol key	Double-address key, switch between two addresses by repeated press
7 8 9 4 5 6 1 2 3 0	Digit key	Digit input
.	Radix point key	Radix point input
INPUT	Input key	Confirm the input of parameter, offset value, etc
OUTPUT	Output key	Start the communication output
CHANGE	Change key	Switch of the imfomation and display.

Keys	Name	Description
INSERT/ALTER DELETE CANCEL	Edit key	Insert, modify or delete the part program or field In editing.（INSERT/ALTER can switch between inserting and modifying by repeated press）
EOB	EOB key	End prompt of block input
	cursor move key	Move the cursor in different directions.
	Page up/down key	Page up/down on display

1.1.3　Display Menu

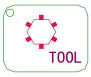
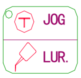
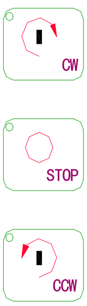
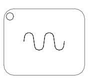
Menu keys	Comment
POSITION	Enter the position interface. There are four display models including relative coordinate, absolute coordinate, integrated coordinate, coordinate & program.
PROGRAM	Enter the program interface. There are three display models including the content of program, the list of program, the program status.
OFFSET	Enter the tool offset interface, macro variable interface (two interfaces can be switched by repeated press). Tool offset interface displays the tool offset value; macro variable interface displays the macros variable of the CNC.
ALARM	Enter the alarm interface. There are two display models including CNC alarm and PLC alarm.
SETTING	Enter the setup interface, graph interface (two interfaces can be switched by repeated press). There are switchs setup, data backup, and password setup; And there are graph setup interface and graph display interface in graph interface.
PARAMETER	Enter the interfaces of status parameter.datas parameter and screw compensation parameter interface (interfaces can be switched by repeated press)
DIAGNOSIS	Enter the interfaces of diagnosis, PLC status, PLC datas, machine soft panel, version information (interfaces can be switched by repeated press). The interfaces of diagnosis, PLC status, PLC data, the internal signal status of CNC and the status of the PLC addresses and the data. It can be operated on the machine soft panel; The version of the CNC software, hardware and PLC are displayed in the version information interface.

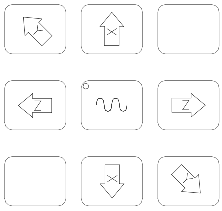
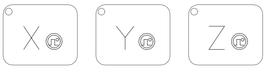
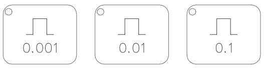
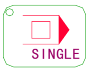
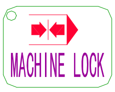
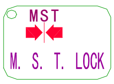
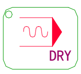
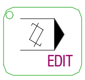
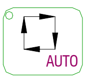
1.1.4　Machine Operation Panel

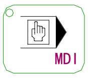
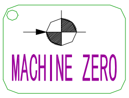
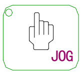
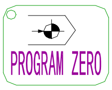
The functions of each key on the machine panel for GSK980TD is defined by PLC programs (ladder diagram). For the defined functions of each key, please refer to the operation manual of the machine supplier.

The functions of Keys in initial PLC program are as follows:

Keys	Name	Description	Available operation mode
PAUSE	Feedhold key	Program, MDI dictates pause	Auto and MDI modes
RUN	Loop start key	Start to run the programs and MDI dictates	Auto and MDI modes

Keys	Name	Description	Available operation mode
	Feedrate key	Adjust the feedrate	Auto, MDI, Edit, Machine zero point return, Handwheel(MPG), Single block, JOG, Program zero point return modes
	Rapid speed key	Adjust the rapid move speed	Auto, MDI, Machine zero point return, Manual, Program zero point return modes
	Spindle speed key	Adjust the spindle speed (available under the simulate spindle rev mode)	Auto, MDI, Edit, Machine zero point return, Handwheel (MPG), Single block, Manual, Program zero point return modes
	Manual toolchange key	Manual change of tools	Machine zero point return, Handwheel(MPG), Single block, Manual, Program zero point return modes
	Jog key Lubrication key	Spindle jog on/off Lubrication on/off	Machine zero point return, Handwheel (MPG), Single block, Manual, Program zero point return modes
	Coolant key	Coolant on/off	Auto, MDI, Edit, Machine zero point return, Handwheel (MPG), Single block, Manual, Program zero point return modes
	Spindle key	Spindle start to run forward Spindle stop Spindle start to run reverse	Machine zero point return, Handwheel (MPG), Single block, Manual, Program zero point return modes
	Rapid switch key	Change between rapid move and feed modes	Auto, MDI, Machine zero point return, Manual, Program zero point return modes

Keys	Name	Description	Available operation mode
	Manual feed key	Moving X, Y, Z axises forward/ reverse under Manual or Single block modes	Machine zero point return, Single block, Manual, Program zero point return modes
	Handwheel (MPG) selection key	Selecting one axes to control from X,Y,Z axis under handwheel (MPG) mode	Handwheel (MPG) mode
	Handwheel (MPG) /Single step,rapid override key	Handwheel unit 0.001/0.01/0.1 mm, Single step unit 0.001/0.01/0.1 mm	Auto, MDI, Machine zero point return, Handwheel (MPG), Single block, Manual, Program zero point return modes
	Single block key	Switch between single block /continuous running status, the single block running indicator LED is on if single block running status is available	Auto, MDI modes
	Segment skip key	When segment skip indicator LED is on, the segment with front '/' will be skipped.	Auto, MDI modes
	Machine lock key	The machine lock indicator LED is on when the machine is locked, X,Y axis outputs are unavailable	Auto, MDI, Edit, Machine zero point return, Handwheel (MPG), Single block, Manual, Program zero point return modes
	Auxiliary functions lock key	The auxiliary functions lock indicator LED is on when it is locked, M, S, T functions are unavailable.	Auto, MDI modes
	Dry running key	The try running indicator LED is on when it is under dry running mode, user program/MDI dictates are run but the machine is without any action.	Auto, MDI modes
	Edit mode key	Enter edit mode	Auto, MDI, Machine zero point return, Handwheel (MPG), Single block, Manual, Program zero point return modes
	Automatic mode key	Enter automatic mode	MDI, Edit, Machine zero point return, Handwheel (MPG), Single block, Manual, Program zero point return modes

Keys	Name	Description	Available operation mode
	MDI mode key	Enter MDI mode	Auto, Edit, Machine zero point return, handwheel, single block, manual operation, program zero point return modes
	Machine zero point return mode key	Enter machine zero point return mode	Auto, MDI, Edit, Handwheel (MPG), Single block, Manual, Program zero point return modes
	Single step/handwheel (MPG) mode key	Enter single step or handwheel modes (Select one mode between them by parameter)	Auto, MDI, Edit, Manual, Program zero point return modes
	Manual operation mode key	Enter manual operation mode	Auto, MDI, Edit, Machine zero point return, Handwheel, Single block, Program zero point return modes
	Program zero point return key	Enter program zero point return mode	Auto, MDI, Edit, Machine zero point return, handwheel, Single step, manual modes

1.2 GENERAL OPERATION

There are seven operation modes including edit, automatic, MDI, machine zero point return, single step/handwheel, manual operation, program zero point return.

- **Edit mode**

Under the edit mode, the programs can be set up or deleted or modified, etc.

- **Automatic mode**

Under the automatic mode, the program is run automatically.

- **MDI mode**

Under the MDI mode, the parameters and the dictates can be input and performed.

- **Machine zero point return mode**

Under the machine zero point return mode, the zero point return for X or Z axis can be performed separately.

- **Handwheel/single block mode**

Under the handwheel/single block mode, the selected axes moves according to the selected increasement.

- **Manual mode**

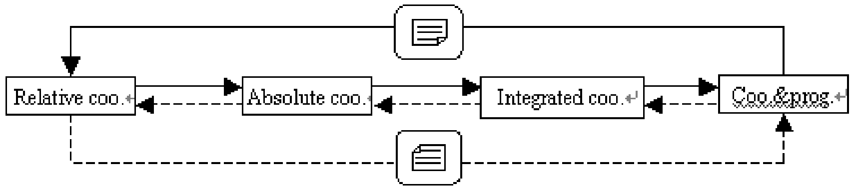
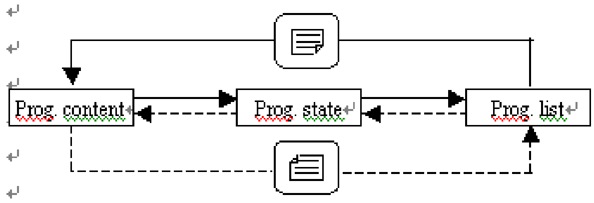
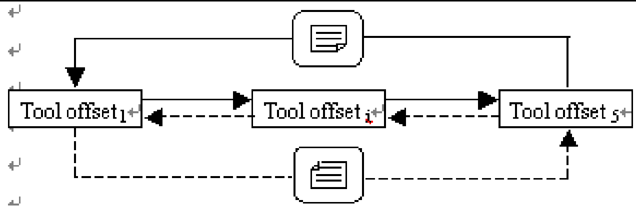
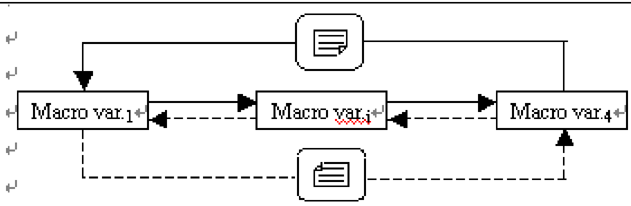
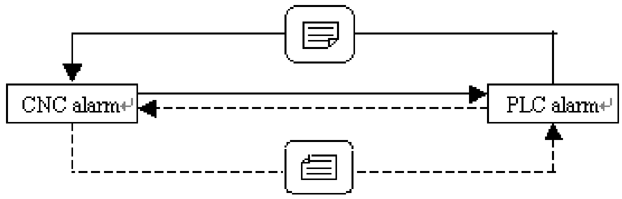
Under the manual mode, the manual feed, munual rapid move, feedreate adjustment, rapid override adjustment and spindle on/off, lubrication on/off, coolant on/off, spindle jog, manual toolchange, etc can be performed.

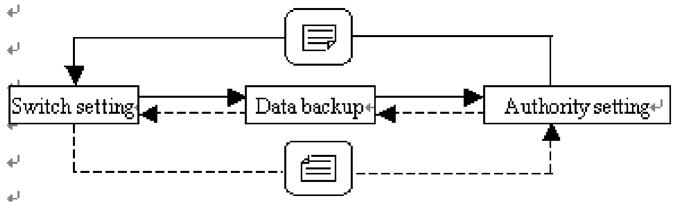
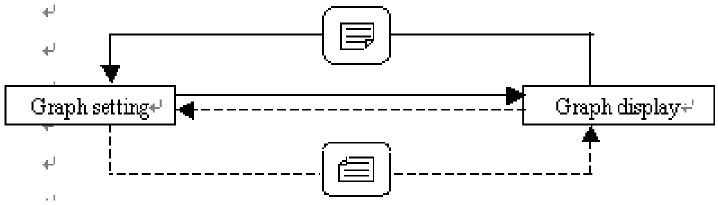
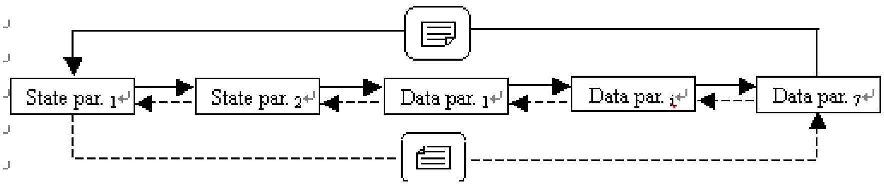
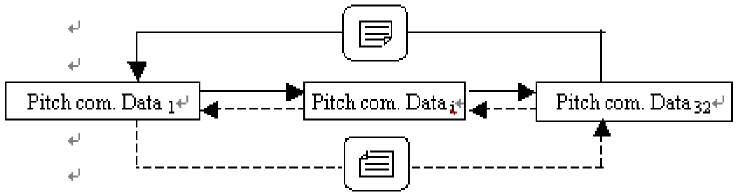
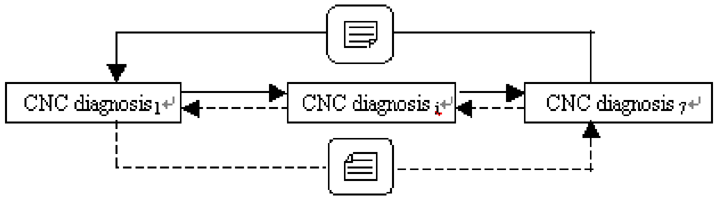
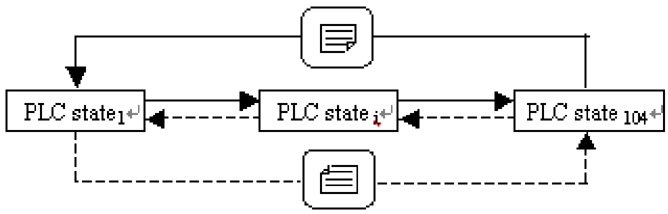
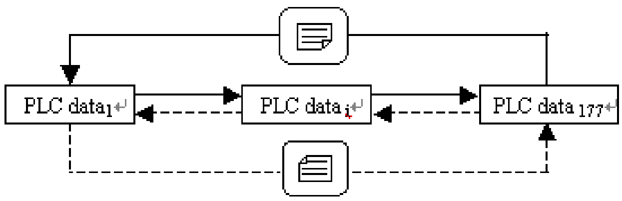
- **Program zero point return mode**

Under the program zero point return mode, the zero point return for X or Z axis can be performed separately.

1.3　DISPLAY

There are nine kinds of displays such as position interface, program interface, etc, and there are several pages (screens) in each interface. Each interfacee is nothing to do with the operation mode. Display menu, page and hiberarchy are as follows:

Keys	Interfaces	Display pages
POSITION	Position interface	
PROGRAM	Program interface	
OFFSET	Tool offset interface	
	Macro variable interface	
ALARM	Alarm interface	

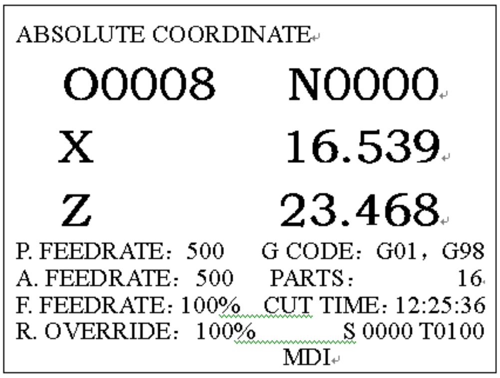
Keys	Interfaces	Display pages
SETTING	Setup interface	
	Graphic interface	
PROGRAM	Status parameter	
	Datas parameter	
	Worm offset value parameter	
DIAGNOSIS	CNC diagnosis	
	PLC status	
	PLC datas	
	Tool panel	Machine soft panel
	Version infomation	Version information

1.3.1 Position Interface

Enter the position interface by pressing the POSITION key, there are four pages including absolute coordinate, relative coordinate, integrated coordinate and coordinate/program, which can be turned over by pressing key or key.

1） Absolute coordinate page

The displaying coordinates of X and Z axises are absolute positions in current workpiece's coordinate, and the coordinates are kept when the power returns on; the coordinate of workpiece is defined by G50.



Programming speed: defined by F code in the user program.

Note: "Programming speed" can be displayed under automatic and MDI modes; under the machine zero point return, program zero point return, and manual mode, the manual feedrate will be displayed; under the handwheel mode, the handwheel increasement will be displayed; and under the single blcok mode, the single block increasement will be displayed.

Actual speed: the transformed speed by the feedrate override during the actual running.

Feedrate override: the selected override by the feedrate override keys.

G code: The modal values of G codes in group 01 and group 03. (the group 01 and 03 values of G code in the running segment?)

Part count: When the M30 (or M99 in the main program) is finished once, the workpice number will be added one accordingly.

Cut time: Count from the automatic running start, the time units take turns as hour, minute and second.

Part count and cut time memory reset after power off.

Part count reset: Press and hold the CANCEL key, and then press the N key.

Cut time reset: Press and hold the CANCEL key, and then press the T key.

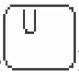
S0000: Spindle rev is feedbacked by the spindle encoder; a spindle encoder must be fixed if the actual rev display is required.

T0100: Current tool number and tool offset number.

2) Relative coordinate page

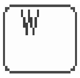
The displaying coordinated values of U and W axises are relative positions of reference point, and the coordinates can be kept when the power returns on. The U and W can be reset at any time. After reset, the current position is the new reference point. When the Bit1 of parameter No.005 is 1 in this controller, the coordinate of U and W are the same with the absolute coordinate when the absolute coordinate is defined by G50.

Relative coordinate reset:

Press the 

key until the letter U glints on relative coordinate display page, then press the 

key, U will be reset.

Press the 

key until the W glints on relative coordinate display page, then press the 

key, U will be reset.

RELATIVE COORDINATES

O0008 N0000

U 16.539

W 23.468

P. FEEDRATE: 500 G code: G01，G98
A. Feedrate: 500 Parts : 16
F. Feedrate: 100% Cut time: 12:25:36
R. Override: 100% S 0000 T0100
MDI

3) Integrated coordinate page

On the integrated page, relative coordinate, absolute coordinate, tool coordinate, distances to go are displayed (Distances to go is displayed only under the auto and MDI modes).

The displaying coordinate of machine coordinate are the same with the current position in the machinecoordinate, the machine coordinate is defined based on the machine zero point return.

Distances to go are the distances between the current position and target position that is appointed by the segment or MDI dictate.

The display is shown as follows:

INTEGRATE COORDINATES

O0008 N0000

(INCREMENTAL COO.)		(ABSOLUTE COO.)	
U	-41.651	X	-41.651
W	49.068	Z	49.068
(MACHINE COO.)		(REMAINING)	
X	-41.651	X	0.000
Z	49.068	Z	0.000

S 0000 T0100

4) Coordinate & program pages

On the coordinate & program page, the absolute and relative coordinates are displayed at the same time for the current position (If the Bit0 of parameter No.180 is 1, the absolute coordinate and distances to go for current position will be displayed), as well as the 5 segments of current program, during the processing, the displaying programs are updating momentarily, the cursor falls at the running block.

```
COO.&PROG.                          O0008
N0000
(INCREMENTAL COO.)          (ABSOLUTE COO.)
    U   -41.651               X    -41.651
    W    49.068               Z    49.068
O0008;
G50 X0 Z0;
G1 X100 Z100 F200;
G2 U100 W50 R50;
G0 X0 Z0
                               S 0000 T0100
```

1.3.2　Program Interface

Enter the porgram interface by pressing the  key, under the non-edit modes, there are three pages including program content, program status and program list. Switch them by pressing the  key or  key. Under the edit mode, there is only program contents page，by pressing the  key or  key to display all the contents of current program。

1)　Program contents page

On the program contents page, program contents including current block are displayed. The program contents can be browsed forward or reverse by pressing key or key.

```
PROG.CONTENT  LINE6   COLUMN1    O0008
N0000
O0008 (CNC PROGRAM.20051020);
G50 X0 Z0;
G1 X100 Z100 F200;
G2 U100 W50 R50;
G0 X0 Z0;
X100 Z100;
M30;
%


                               S 0000 T0100
                           EDIT
```

2) Program status page

On the program status page, the status of current G,M,S,T,F dictates will be displayed, but contents of current block will be displayed under the auto and MDI modes.

```
PROG. STATE                    O0008 N0000
        ADDRESS       MODUL VALUE
        X
        Z               G00
        U               G97
        W               G98
        R
        F
        M               G21
        S               G40
        T
        P
        Q
                            S 0000 T0100
                         MDI
```

3) Program list page

The displaying contents of program list page:

(a) Version of the software: Current sofeware version of the controlle will be displayed.

(b) Workpiece program account: The maximium number of programs can be storaged and the number of storaged programs (including the subprograms) will be displayed.

(c) Storage size: The maximium size of programs can be storaged and the size of storaged programs will be displayed.

(d) Program list: Display the names of user program orderly.

```
PROG. LIST                       O0009 N0000
   VERSION: GSK-980TD  V05.10.20
   PART-PRG NO: MOST 384;      USED:      20
   MEMORY CAPACITY: 6144 KB;  USED: 5310 KB
   PROG. LIST:
    O0000 O0002 O0003 O0004 O0005 O0006
    O0007 O0008 O0009 O0010 O0011 O0012
    O0014 O0023 O0088 O0089 O1000 O0044
    O0100 O0101


 PROGRAM CAPACITY: 16KB     NOTE: QIU TOU GAN
                             S 0000 T0100
                        MDI
```

1.3.3 Tool Offset, Macro Variable Interface

OFFSET key is a compound key，the display will come into tool offset interface under other display by pressing OFFSET once，press OFFSET again, it will come into macro veriable interface。

1.Tool offset interface

There are 5 pages of tool offset pages under the tool offset interface, and there are 33 groups of offset (No.000～No.032) are available for users, by pressing key or key to change pages, details are listed as follows:

TOOL OFFSET　　　　　O0008 N0000

NO.	X	Z	R	T
_000	0. 000	0. 000	0. 000	0
001	-90. 720	-116. 424	0. 000	0
002	0. 000	0. 000	0. 000	0
003	0. 000	0. 000	0. 000	0
004	0. 000	0. 000	0. 000	0
005	0. 000	0. 000	0. 000	0
006	0. 000	0. 000	0. 000	0
007	0. 000	0. 000	0. 000	0

INCREMENTAL COORDINATES
U　0. 000　　W　0. 000
NO.　000　　S 0000 T0100
MDI

2. Macro variable interface

There are three pages under macro variable interface, each page can be displayed by pressing the key or key, 48 groups of macro variable（No.200～No.231 and No.500～No.515）will be displayed on the page, the macro variable value can be set directly by macro dictates or keyboard. The macro variable would be hold in case of power off.

MACRO VARIABLE　　　　O0008
N0000

NO.	DATA	NO.	DATA
200	0	208	0
201	0	209	0
202	0	210	0
_203	0	211	0
204	0	212	0
205	0	213	0
206	0	214	0
207	0	215	0

INCREMENTAL COORDINATES
U　0. 000　　W　0. 000
NO.　203　　S 0000 T0100
MDI

1.3.4　Alarm Interface

Enter the alarm interface by pressing the ALARM key, there are two pages including CNC alarm page and PLC alarm, it can be checked by pressing key or key.

1) PLC alarm: Display the amount of CNC alarms and PLC alarms, and the current alarm number of PLC, it can be displayed 24 PLC alarms at the same time, the detailed corresponding infomations of each alarm can be checked out by moving the cursor. If there are two alarms, page is shown as follows:

PLC ALARM　　　　O0008 N0000

Two PLC alarms, two CNC alarms

1000　　1022

Alarm No.:1000　Information-bit address:A0000.0

1000:tool changing time is too long

ALARM　　MDI

PLC ALARM　　　　O0008 N0000

Two PLC alarms, two CNC alarms

1001　　1022

Alarm No.:1022　Information-bit address:A0002.6

Chuck is unclamped and spindle can’t be started up

ALARM　　MDI

Book2 operation

The showing page when the cursor falls at No.1000 alarm The showing page when the cursor is moved to No.1022 alarm

2) CNC alarm: Display the amount of CNC alarms and PLC alarms, and current alarm number of CNC, it can be displayed 24 pieces of CNC alarms at the same time, detail corresponding infomations of each alarm can be check out by moving the cursor.

CNC ALARM O0008 N0000	CNC ALARM O0008 N0000
Two CNC alarms, two PLC alarms	Two CNC alarms, two PLC alarms
301 044	301 044
System alarm:301	Program alarm:044,alarm in line 15 of O0008 program
Parameter switch is opened	Single cutting value exceeds its allowed range in X or Z direction in G74 or G75
ALARM MDI	ALARM MDI

The showing page when the cursor falls at No.301 alarm The showing page when the cursor is moved to No.044 alarm

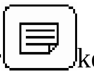
3) Alarms clear: If there are several alarms occur at the same time, only the alarmpointed by cursor can be cleared each time (under the alarm interface, all alarms can be cleared by pressing both RESET key and CANCEL key synchronously). The alarm page is shown as follows:

PLC ALARM O0008 N0000	PLC ALARM O0008 N0000
Two PLC alarms, two CNC alarms	Two PLC alarms, two CNC alarms
1000 1022	1001 1022
Alarm No.:1000 Information-bit address:A0000.0	Alarm No.:1022 Information-bit address:A0002.6
1000:tool changing time is too long	Chuck is unclamped and spindle can't be started up
ALARM MDI	ALARM MDI
Current alarm page	Display afte reset

1.3.5 Setting Interface

SETTING key is a compound key, it will come into setting interface under any other interface by pressing SETTING once. Press SETTING again, it will come into graphic interface, and it will be switched over between two interfaces by repeated pressing the SETTING key.

1,Setting interface

There are three pages under setting interface, each page can be displayed by pressing the  key or  key,

1) Switchs setting: Displaying parameter, program, and automatic number on/off.

Parameter switch: when the parameter switch is on, the parameter can be modified; otherwise, it cannot be modified.

Program switch: when the program switch is on, the program can be edited; otherwise, it cannot be edited.

Automatic number switch: when the Automatic number switch is on, the number of programs can be generated automatically; otherwise, it only can be input manually when it is needed.

```
SWITCH SETTING                          O0008
N0000

   Parameter switch ：   *off    on
   Program switch   ：   *off    on
   Sequence number：     *off    on



                                 S 0000 T0100
                          MDI
```

2) Datas backup: on this page, it can be backuped or resumed for CNC datas (including status parameter, data parameter, worm offset, tool offset, etc.).

Datas backup (user): for users to backup the CNC datas (preserve)

Datas resume (user): for users to resume the CNC datas (read)

Resume default parameter 1 (test): for users to read the original parameter datas for CNC testing.

Resume default parameter 2 (step): for users to read the original parameter datas for matching step driver.

Resume default parameter 3 (servo): for users to read the original parameter datas for matching servo driver.

```
PARAMETER OPERATION              O0008
N0000


    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)

                   MDI
```

```
PARAMETER OPERATION              O0008
N0000


    Backup par.    (machine factory)
    Resume par.    (machine factory)
    Resume default par.1 (test)
    Resume default par.2 (step)
   ■Resume default par.3 (servo)


Press [IN] + [P] to confirm(power on
again)

                   MDI
```

The displaying page for users of 3,4,5 level　　The displaying page for users of 2 level

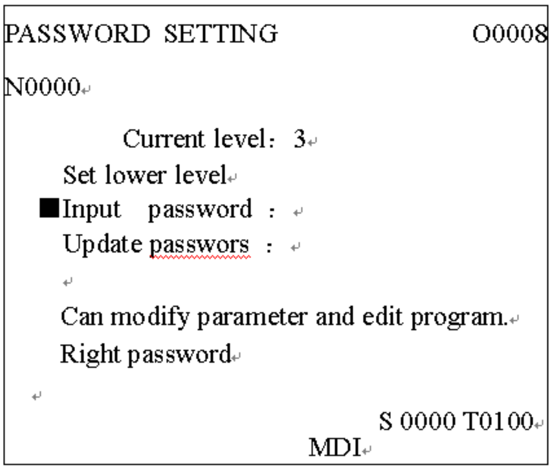
3)　Password setting: display , setting users level

There are four password levels of GSK980TD, from high to low they are machine builder (2 level), equiment administrator (3 level), technician (4 level), operator (5 level)

Machine builder level: be able to modify the status parameter, datas parameter, worm offset values, programs editing (including variable macros), PLC ladderlike graph edit and modify, download the ladderlike graphs of CNC.

Equiment administrator level: the original password is 12345,be able to modify the status parameter, datas parameter, worm offset values, programs editing.

Technician level: the original password is 1234, be able to modify the tool offset value (for tool position correcting), variable macros, programs editing.

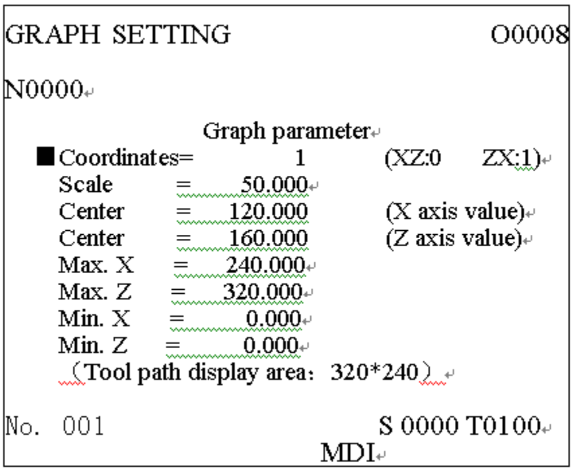


2,Graphic interface

There are two pages including graph setting and graph display under graphic interface, each page can be displayed by pressing the key or key,

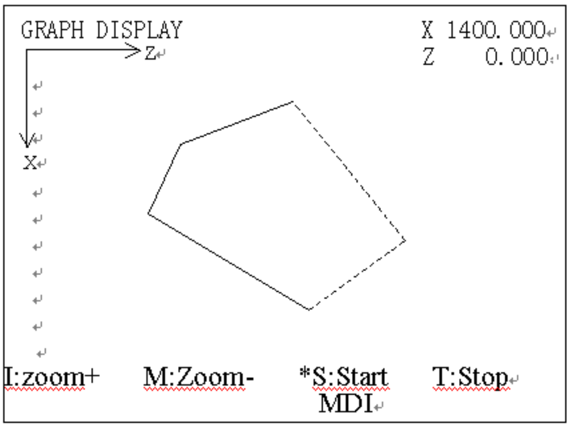
1) Graph setting page

On the graph setting page, the coordinates, scale and area, etc, of graph display can be selected.



2) Graph display page

On the graph display page, display the track in each parameter range (takes absolute coordinate as reference) in the graph setting page.



1.3.6 Status Parametr, Data Parameter, Pitch Error Compensation Parameter Interfaces

PARAMETER key is a compound key, it can be enter several interfaces such as status parameter, data parameter and worm offset, etc, by repeated pressing this key.

1, Status parameter interface

Enter the status parameter interfaces by pressing the PARAMETER key. There are 30 status parameters listed by two pages, each page can be displayed or to modify the parameters on them by pressing the key or key, details are as follows:

It can be learnt from the status parameter page, there are two rows of parameter contents under the page, the first row displays the Chinese meaning of which the cursor falls at currently, the displaying parameter can be changed by pressing the L D key or W key; The second row displays the abbreviations of all the English which the cursor falls at currently.

STATE PARAMETER O0008

N0000

No.	Data	No.	Data
001	00010001	009	00000010
002	11101000	010	00101111
003	01010100	011	10101000
_004	01000000	012	00110011
005	00010001	013	00000000
006	00000000	014	00000111
007	00000000	164	11100100
008	00000000	168	00000000

BIT0:1/0:reserved

*** RDRN DEC1 ORC TOC DCS PROD ***

No. 004 S0000 T0200

MDI

2, Data parameter interface

Enter the data parameter interface by pressing the PARAMETER key (pressing the key under the status parameter page), there are 110 pieces of parameters listed by seven pages, each page can be displayed or to modify the related parameters on them by pressing the key or key, details are as follows:

It can be learnt from the parameter page that, there is Chinese words under the page, displays the parameter meaning which the cursor falls at currently.

DATA PARAMETER O0008

N0000

No.	Data	No.	Data
015	1	023	7600
016	1	024	50
017	1	025	50
018	1	026	100
019	5	027	8000
020	2	028	500
021	645	029	100
_022	4000	030	10

Max. X rapid traverse speed(mm/min)

No. 022 = S0000 T0200

MDI

3, pitch error compensation parameter interface

Enter the worm offset value interface by pressing the PARAMETER key, there are 256 pieces of worm offset value parameters listed by 32 pages, each page can be displayed by pressing the key or key:

PITCH ERROR COM. PARAMETER O0008 N0000

No.	X	Z
000	55	32
001	-23	15
_002	0	0
003	0	0
004	0	0
005	0	0
006	0	0
007	0	0

No. 002 S0000 T0200

MDI

1.3.7 CNC Diagnosis, PLC Status, PLC Data, Tool Panel, Version Information Interfaces

DIAGNOSIS key is a compound key, it can be enter CNC diagnosis, PLC status, PLC datas, Tool panel, version information interfaces by repeated pressing this key.

1,CNC diagnosis interface

The status of input/output between CNC and machine, the signals transmiting between CNC and PLC, the inside datas of PLC and the inside status of CNC, all these infomations can be displayed by the diagnosis. Enter the CNC diagnosis page by pressing the DIAGNOSIS key, there are keyboad diagnosis, status diagnosis and auxiliary parameters, etc, can be displayed.and they can be check out by pressing key or key。

On the CNC diagnosis page, there are two rows of Chinese words under the page, the first row displays the Chinese meaning of which the cursor falls at currently, the displaying parameter can be changed by pressing the L D key or W key; The second row displays the abbreviations of all the English which the cursor falls at currently.

CNC DIAGNOSIS　　　　O0008 N0000

No.	Data	No.	Data
_000	00000000	008	00110011
001	00000000	009	00000000
002	00000000	010	00000000
003	00000000	011	00000000
004	00100000	012	00000000
005	00000000	013	00000000
006	00000000	014	00000000
007	00000000	015	00000000

Bit6: chuck control signal（machine to PLC）
TCP DIQP XDEC BDT T04 T03 T02 T01
No. 000　　　　S0000 T0200
MDI

2, PLC status interface

There displaying in turn of some address status such as X0000~X0029, Y0000~Y0019, F0000~F0255, G0000~G0255, A0000~A0024, K0000~K0039, R0000~R0999 under the PLC status interface. Enter the PLC status interface by repeated pressing the DIAGNOSIS key. The singnal status for each PLC addresses can be checked out by pressing the key or key.

On the PLC status page, there are two rows of words under the page, the first row displays the Chinese meaning of which the cursor falls at currently, the displaying parameter can be changed by pressing the L D key or W key; The second row displays the abbreviations of all the English which the cursor falls at currently.

PLC STATE　　　　O0008 N0000

No.	Data	No.	Data
_X0000	00000000	X0008	00000000
X0001	00000000	X0009	00000000
X0002	00000000	X0010	00000011
X0003	00000000	X0011	00001100
X0004	00000000	X0012	00000000
X0005	00000000	X0013	00000000
X0006	00000000	X0014	00000011
X0007	00000000	X0015	00000000

Bit7: Toolpost locking signal (TCP)
TCP DIQP ESP T5 DECX BDT T4 T3
No. X0000　　　　S0000 T0200
MDI

3, PLC data interface

There displaying in turn of some datas of register such as T0000～T0099,D0000～D0999,C0000～C0099,DT000～DT099,DC000～DC099 under the PLC status interface. Enter the PLC status interface by repeated pressing the DIAGNOSIS key. The singnal status for each PLC addresses can be checked out by pressing the key or key.

On the PLC data page, there is a row of Chinese words under the page, shows the meaning of the parameter that the cursor falls at currently.

PLC VALUE O0008 N0000

No.	Current value	preset value
T0000	0	0
T0001	0	0
T0002	0	0
T0003	0	0
T0004	0	0
T0005	0	0
T0006	0	0
T0007	0	0

Timer
No. T0000 S0000 T0200
MDI

4, Tool panel

Enter the tool panel by repeated pressing the 

key, the machine can be controlled by the panel on this page, the page of tool panel are displayed as follows:

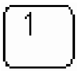
MACHINE CONTROL PANEL O0008 N0000

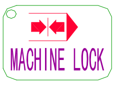
Machine locked	（key1） :	off	*on
M.S.T locked	（key2） :	off	*on
Single block	（key3） :	off	*on
Dry run	（key4） :	off	*on
Optional jump	（key5） :	off	*on

Incremental coordinates
U 372.815
W -10.584

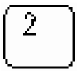
S0000 T0200
MDI

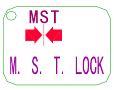
Manual:

By repeated pressing the 

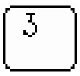
key, the function of machine lock can be switched between on and off, function the same with 

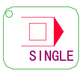
key.

By repeated pressing the 

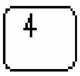
key, the function of auxiliary lock can be switched between on and off, function the same with 

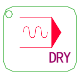
key.

By repeated pressing the 

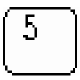
key, the function of single step can be switched between on and off, function the same with 

key.

By repeated pressing the 

key, the function of try running can be switched between on and off, function the same with 

key.

By repeated pressing the 

key, the function of block skip can be switched between on and off, function

Book2 operation

the same with  key.

5, Version infomation

Enter the version information interface by repeated pressing the  key, there are software, hard ware, PLC version infomations of current CNC system displaying on the version information page. Displaying page are as follows:

VERSION MESSAGE　　　　O0008 N0000

Product type　:　GSK-980TD

Software version: V05.10.20

Hardware version: 2.03.002—05.10.20

Ladder design　:　GSK

Ladder version :　V2.0 05.09.08

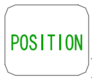
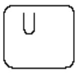
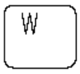
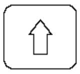
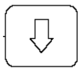
Verifying code :　6CE7

Note:GSK980TD-2 standard ladder（980TA interface）

S0000 T0200

MDI

1.3.8　LCD Brightness Adjusting

Enter the relative coordinate displaying page by pressing the key（Also pressing the  key or  key if necessary），pressing the  key or  key until the LED of the key start to glitter，and then press the  key，the contrast of LCD will be lower（darker），press the  key，the contrast of LCD will be higher（brighter）。

RELATIVE COORDINATE

O0008　　N0000

X　　16.539

Z　　23.468

P. FEEDRATE: 500　　G CODE: G01，G98

A. FEEDRATE: 500　　PARTS:　　16

F. FEEDRATE: 100%　　CUT TIME: 12:25:36

R. OVERRIDE: 100%　　S 0000 T0100

MDI

1.4 COMMON OPERATION TABLE

Class	Function	Operation	Operation modes	Displaying page	Password level	Program switch	Parameter switch	Comment

Class	Function	Operation	Operation modes	Displaying page	Password level	Program switch	Parameter switch	Comment
Reset	Relative coordinate value of X axis reset	U , CANCEL		Relative coordinates				Chapter 2 1.3.1
	Relative coordinate value of Z axis reset	W , CANCEL		Relative coordinate				
	Part count reset	CANCEL + N		Relative or absolute coordinate				
	Cut time reset	CANCEL + T						
	Tool offset value of X axis reset	X , INPUT		Tool offset	Level 2,3,4			Chapter 2 7.4.3
	Tool offset value of Z axis reset	Z , INPUT		Tool offset	Level 2,3,4			Chapter 2 7.4.3
Data setting	Status parameter	Parameter value, INPUT	MDI mode	Status parameter	Level 2,3		On	Chapter 2 7.4.3
	Data parameter	Parameter value, INPUT	MDI mode	Data parameter	Level 2,3		On	
	Worm offset value of X axis input	X , offset value, INPUT	MDI mode	Worm offset value parameter	Level 2		On	Chapter 2 10.1.3

Class	Function	Operation	Operation modes	Displaying page	Password level	Program switch	Parameter switch	Comment
	Worm offset value of Z axis input	Z, offset value, INPUT	MDI mode	Worm offset value parameter	Level 2		On	Chapter 2 10.1.3
	Macro variable	Macro variable value, INPUT		Macro variable	Level 2,3,4			
	Tool offset increment of X axis input	U, Offset increasement		Tool offset	Level 2,3,4			Chapter 2 7.4.2
	Tool offset increment of Z axis input	W, offset increasement		Tool offset	Level 2,3,4			Chapter 2 7.4.2
Searching	Search down from current cursor position	Letter, ↓	Edit mode	Program contents	Level 2,3,4	On		Chapter 2 6.1.3
	Search up from current cursor position	Letter, ↑	Edit mode	Program contents	Level 2, 3,4	On		Chapter 2 6.1.3
	Search down from current program	O, ↓	Edit or automatic mode	Program contents, list or status	Level 2,3,4			Chapter 2 6.4.1
	Search up from current program	O, ↑			Level 2,3,4			Chapter 2 6.4.1
	Searching for the appointed program	O ,Name of program, ↓			Level 2,3,4			Chapter 2 6.4.2

Class	Function	Operation	Operation modes	Displaying page	Password level	Program switch	Parameter switch	Comment
	Searching for the status parameter, data parameter or worm offset value parameter	P Q number of parameter INPUT		Related pages of data				Chapter 2 10.1.3
	Searching for the PLC status and data	P Q, address number, INPUT		PLC status & data				
Delete	The letter with cursor will be deleted	DELETE	Edit mode	Program contents	Level 2,3,4	On		Chapter 2 6.1.6
		CANCEL	Edit mode	Program contents	Level 2,3,4	On		
	Single block detecting	Move the cursor to the head of row, DELETE	Edit mode	Program contents	Level 2,3,4	On		The block has it's number Chapter 2 6.1.7
	Blocks delecting	CANCEL , N , number, DELETE	Edit mode	Program contents	Level 2,3,4	On		Chapter 2 6.1.8
	Block delecting	CHANGE , letter, DELETE	Edit mode	Program contents	Level 2,3,4	On		Chapter 2 6.1.9
	Single program delecting	O , name of program, DELETE	Edit mode	Program contents	Level 2,3,4	On		Chapter 2 6.3.1

Class	Function	Operation	Operation modes	Displaying page	Password level	Program switch	Parameter switch	Comment
	All programs delecting	O, −999, DELETE	Edit mode	Program contents	Level 2,3,4	On		Chapter 2 6.3.2
Rename	Rename for programs	O, name of program, INSERT ALTER	Edit mode	Program contents	Level 2,3,4	On		When the program number is larger than 9000, password of level 2 is needed. Chapter 2 6.6
Copy	Copy program	O, name of program, CHANGE	Edit mode	Program contents	Level 2,3,4	On		When the program number is larger than 9000, password of level 2 is needed. Chapter 2 6.7
CNC → CNC (Send)	Tool offset	OUTPUT	Edit mode	Tool offset	Level 2,3		On	Chapter2 11.6
	Status parameters	OUTPUT	Edit mode	Status parameter	Level 2,3		On	
	Data parameters	OUTPUT	Edit mode	Data parameters	Level 2,3		On	
	Worm offset value parameter	OUTPUT	Edit mode	Worm offset value parameter	Level 2		On	

Class	Function	Operation	Operation modes	Displaying page	Password level	Program switch	Parameter switch	Comment
	Transmitting for single program	O, name of program, OUTPUT	Edit mode	Program contents	Level 2,3,4	On		
	Transmitting for all programs	O, -, 999, OUTPUT	Edit mode	Program contents	Level 2,3,4	On		
CNC → CNC (receive)	Tool offset		Edit mode		Level 2,3,4		On	Chapter 2 11.6
	Status parameter		Edit mode		Level 2,3		On	
	Data parameter		Edit mode		Level 2,3		On	
	Worm offset value parameter		Edit mode		Level 2		On	
	Program of workpiece		Edit mode		Level 2,3,4	On		
CNC →PC (upload)	Tool offset	OUTPUT	Edit mode	Tool offset	Level 2,3,4		On	Chapter 2 11.5.3
	Status parameter	OUTPUT	Edit mode	Status parameter	Level 2,3,4		On	Chapter 2 11.5.4
	Data parameter	OUTPUT	Edit mode	Data parameter	Level 2,3		On	
	Worm offset value parameter	OUTPUT	Edit mode	Worm offset value parameter	Level 2		On	
	Transmitting for single program	O, name of program, OUTPUT	Edit mode	Program contents	Level 2,3,4	On		Chapter 2 11.5.1

Class	Function	Operation	Operation modes	Displaying page	Password level	Program switch	Parameter switch	Comment
	Transmitting for all program	O, – 999, OUTPUT	Edit mode		Level 2,3,4	On		Chapter 2 11.5.2
PC→CNC (Download)	Tool offset		Edit mode		Level 2,3,4		On	Chapter two 11.4.2
	Status parameter		Edit mode		Level 2,3		On	Chapter 2 11.4.3
	Data parameter		Edit mode		Level 2,3		On	
	Worm offset value parameter		Edit mode		Level 2		On	Chapter 2 11.4.3 level 2 password is needed
	Program of workpiece		Edit mode		Level 2,3,4	On		Chapter 2 11.4.1 When the program number is larger than 9000, password of level 2 is needed. Chapter 2 6.7
LCD contrast	Contrast become higher (brighter)	(U , ⇩)or(W , ⇩)	Edit mode	Relative coordinate				Chapter 2 1.3.8

Class	Function	Operation	Operation modes	Displaying page	Password level	Program switch	Parameter switch	Comment
	Contrast become lower (darker)	([U], [⇧])or([W], [⇧])	Edit mode	Relative coordinate				
Switch setting	Turn on the parameter switch	[DL]		Switch setting	Level 2,3			Chapter 2 10.1.1
	Turn on the program switch	[DL]		Switch setting	Level 2,3,4			
	Turn on the automatic number	[DL]		Switch setting				
	Turn on/off the parameter switch	[W]		Switch setting	Level 2,3			
	Turn off the parameter switch	[W]		Switch setting	Level 2,3,4			
	Turn off the automatic number	[W]		Switch setting				

Note:Symbol “,” in the operation table means there is distance between pressing the two keys, Symbol “+” in the operation table means there is no distance between pressing two keys, the keys have to be pressed at the same time.

For example: [U], [CANCEL] means to press the [U] key first，and then press the [CANCEL] key ；[CANCEL] + [N] means to press the two keys at the same time.

Chapter 2 POWER on/off and SAFETY OPERATION

2.1 POWER on

Before turning on the GSK980TD, Please confirm:

1、 The machine is normal.

2、 The voltage of power supply is up to the mustard.

3、 The connections are connected correctly and steadily.

Below picture will turn up on the screen after the power on of GSK980TD.

At the moment the system is processing the self-inspect and initialization. And after that, the screen will show the page of current position (relative coordinates).

RELATIVE COORDINATE	
O0008	N0000
X	16.539
Z	23.468
P. FEEDRATE：500	G CODE：G01，G98
A. FEEDRATE：500	PARTS： 16
F. FEEDRATE：100%	CUT TIME：12:25:36
R. OVERRIDE：100%	S 0000 T0100
	MDI

2.2 POWER off

Before turning off the machine, please confirm:

1、 The X and Z axies of CNC are stopped.

2、 Auxiliary function (such as spindle, coolant,etc.) off.

3、 Cut off the CNC power before cutting off the machine power.

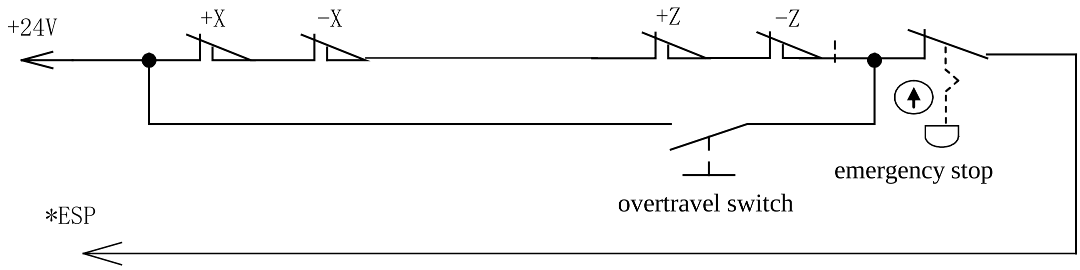
Note: About cutting off the machine power please refer to the operation manual from the machine factory.

2.3 OVERRIDE PROTECTION

To avoid damage of the machine due to the overrides of X and Z axies, the machine has to take steps of override protection.

2.3.1 Override Protection for Hardware

Install stroke limit inhibition switches for X and Z axies separately, and connect the wire according to below chart, at this moment the bit3 (MESP) of No.172 status parameter must set as 0. When the tool moves beyond the stroke end set, the stroke limit inhibition switches start to works, GSK980TD will be stopped and the emergency stop alarm will be displayed.

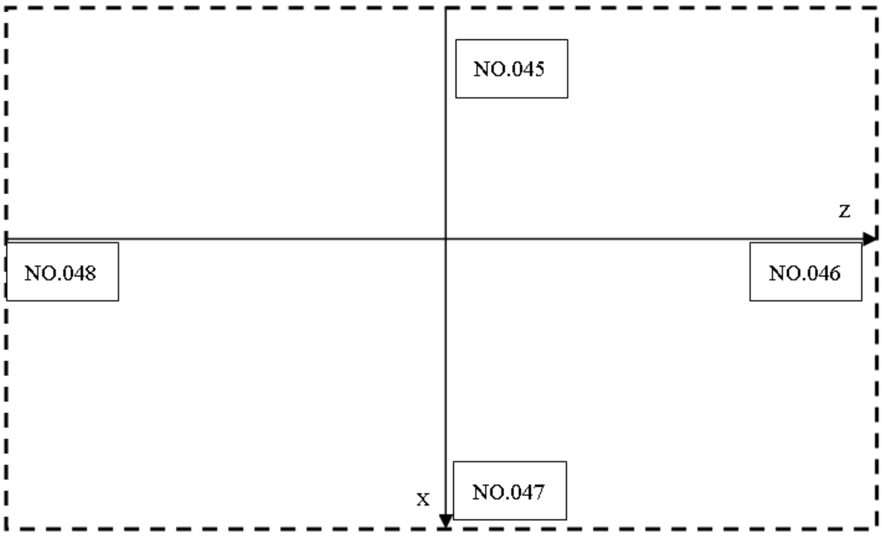


When the override of hardware occurs, the emergency stop alarm will be displayed. To clear the alarm please follow below steps: depress the override releasing button, move the worktable in a reverse direction (if the override occurs in a positive direction, reverse it to the negative way; otherwise, reverse it to the positive way) to divorce from the stroke limit inhibition switche.

2.3.2 Override Protection for Software

When the bit4 of No.172 status parameter is set as 0, the stroke limit inhibition of software is effective.

The journey range of software is set by NO.045、NO.046 、NO.047、NO.048 data parameters, which take the machine coordinate value as reference.Displaying as below chart, X and Z are the axies in the machine coordinate, NO.045 and NO.047 are the maximum journeys in positive and negative directions of X axies, NO.046 and NO.048 are the maximum journeys in positive and negative directions of Z axies, the area inside the broken lines is the journey range of software.



If the tool position (machine coordinate) exceeds the area inside of the broken lines, the override alarm will occurs. To clear the alarm please follow below steps: Press the reset key to clear the alarm, move the worktable in a reverse direction (if the override occurs in a positive direction, reverse it to the negative way; otherwise, reverse it to the positive way) to divorce from the override alarm.

2.4 EMERGENCY OPERATION

During the processing, there would be something beyond expectation occurs on account of the program editing, operation by user or product failure, etc. at the very moment GSK980TD should be stopped immediately. Here just list the disposals can be realized by GSK980TD under the emergency, for the disposals of the machine under the emergency please refer to the related manual which is offered by the machine builder.

2.4.1 Reset

When the output and the axies are abnomal, GSK980TD will be reset by pressing the **key.**

1、 All axies are stopped.

2、 Output of M and S functions are unavailable (Whether shut down the spindle rotates forward/reverse, lubrication, coolant signalm, or PLC ladderlike chart define automatically can be set by the parameter by pressing the **key.**)

3、 Automatic running is terminated, modal status functions and status are remained.

2.4.2 Emergency Stop

Pressing the emergency stop button during the processing under the dangerous or emergency situations (when the outer emergency stop signal is available), The CNC will be turned in enmergency stop,at the very moment the moving of machine will be stopped immediately, all output (such as rotates of the spindle, coolant, etc.) is shut down. Clear the emergency stop alarm by releasing the emergency stop button, CNC will be turned into reset status. The connection of the circuit is described in section 2.2.1 of this chapter.

Note1: Please confirm all failure is settled before releasing the emergency stop alarms.

Note2: Depress the emergency stop button before power on/off may reduce the electricity impact of the equipment.

Note3: In order to ensure the correctness of coordinate position, machine zero point return needs to be performed again after clearing the emergency stop alarm (If there is no zero point installed, please do not perform this action).

Note4: The outer emergency stop will only be in effect based on the Bit3（MESP）is set as 0 of No.172 parameter.

2.4.3 Feed Hold

Depress the key can hold the running during the process. It must be especially reminded that during the threading cutting and cycle running dictates, this function can not stop the running immediately.

2.4.4 Cutting off the Power

In order to avoid any accidents happen the machine power can be cut off immediately under the dangerous or emergency situations during the process. It must be reminded that there may be some different between the display coordinate and the actual position after the power off, so the tool adjusting and related operations have to be re-done again.

Chapter 3 MANUAL OPERATION

Enter the manual operation mode by pressing the key, manual feed, spindle control, override adjusting, tool change, etc, can be performed under the manual operation mode.

Attention!

The function of each key on the operation panel of 980TD is defined by PLC programs (ladderlike chart), for the detail functions please refer to the manual which is offered by the machine builder.

The following mentioned functions are described according to the standard PLC program of 980TD, please take note!

3.1 MOVEMENT of AXIES

Under the manual operation mode, manual feeding, manual rapid feeding of two axies can be performed.

3.1.1 Manual Feed

Depress and hold the or of can move the X axies forward or reverse, the axies will be stopped by releasing the keys; Depress and hold the or can move the Z axies forward or reverse, the axies will be stopped by releasing the keys. User can also depress the two keys of X and Z directions to realize the moving together. The federate override adjusting of real time is effective.

During the manual feed operation, depress the key, the LED of in the status indicating area will be turned on and the machine is on manual rapid feeding status.

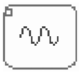
3.1.2 Manual Rapid Traverse

Depress the key of and hold it until the LED of rapid traverse in the status indicating area is turned on, then depress the or key can move the X axies rapidly forward or reverse, the axies will be stopped by releasing the keys; Depress and hold the or can move the Z axies rapidly forward or reverse, the axies will be stopped by releasing the keys. User can also depress the two keys of X and Z directions to realize the moving together. The federate override adjusting of real time is effective.

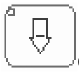
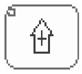
During the manual rapid feeding operation, depress the key, the LED of in the status indicating area will be turned off, the rapid traverse is unavailable, please use manual feeding.

Note1: After the power on or release the emergency stop, if the reference point has not been returned, and when the rapid traverse switch is on (namely,the indicating LED of rapid traverse is on),the rapid moving speed is whether manual feeding speed or rapid traverse speed is decided by the Bit0 (ISOT)

of No.012 status parameter in GSK980TD.

Note2:  **is unavailable under the EDIT/MANUAL modes.**

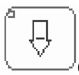
3.1.3 Speed Adjusting

Under the manual mode, the manual feed override can be modified by pressing the  or  key in 

, there are 16 levels. The connections between feed override and feedrate are as follows.

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% error in the table.

Under the manual rapid traverse, manual rapid traverse override can be modified by pressing the  or

Book2 operation

key in RAPID OVERRIDE (it can be also modified by pressing the 0.001, 0.01, 0.1 kdys,the corresponding overrides are Fo，50%，100%), there are four levels including Fo，25％，50％，100％ of speed override of manual rapid traverse. (The speed of Fo is set by the No.032 data parameter)

The selection of rapid traverse is effective under below status.

(1) G00 rapid traverse (2) Rapid traverse during the fixed cycle. (3) The rapid traverse under G28 (4) Manual rapid traverse

3.1.4 Reset for Relative Coordinates Value

1) Enter the position interface by pressing the POSITION key , and then select the relative coordinate page by pressing the or key.

```
RELATIVE COORDINATE
   O0008        N0000
   X           16.539
   Z           23.468
P. FEEDRATE:  500    G CODE:  G01，G98
A. FEEDRATE:  500    PARTS:          16
F. FEEDRATE: 100%   CUT TIME: 12:25:36
R. OVERRIDE:  100%         S 0000 T0100
                   MDI
```

2) Depress and hold the U key until the LED of letter U in the page start to glitter, and then press the CANCEL key.

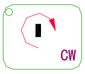
```
RELATIVE COORDINATE
   O0008        N0000
   X            0.000
   Z           23.468
P. FEEDRATE:    0    G CODE:  G01，G98
A. FEEDRATE:    0    PARTS:          16
F. FEEDRATE: 100%   CUT TIME: 12:25:36
R. OVERRIDE:  100%         S 0000 T0100
                   MDI
```

3) Depress and hold the W key until the LED of letter W in the page start to glitter, and then press the CANCEL key.

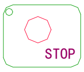
```
RELATIVE COORDINATE
   O0008        N0000
   X            0.000
   Z            0.000
P. FEEDRATE:    0     G CODE: G01, G98
A. FEEDRATE:    0     PARTS:        16
F. FEEDRATE: 100%     CUT TIME: 12:25:36
R. OVERRIDE: 100%          S 0000 T0100
                  MDI
```

3.2 Other MANUAL OPERATIONS

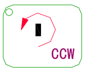
3.2.1 Spindle Rotation forward/reverse and Stop Control



: Under the manual operation, the spindle rotates forward by pressing this key.

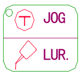


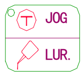
: Under the manual operation, the spindle stops by pressing this key.



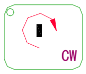
: Under the manual operation, the spindle rotates reverse by pressing this key.

3.2.2 Spindle Step Feed

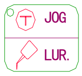


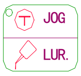
: When the Bit7 of No.175 status parameter is set as 1,depress the 

key to turn on the indicating LED, at this moment the spindle is under step feeding mode.

Under the spindle step feeding mode, depress the 

key, the spindle rotates forward on step feeding. The time and the speed of are set by No.108 and No.109 status parameters separately.

If the Bit7 of No.175 status parameter is set as 0 (at this moment the 

dey is lubrication switch) or when the LED of 

is off, the spindle step feeding is unavailable.

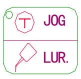
3.2.3 Coolant Control

: Under the manual operation, the coolant can be switched on/off by pressing this key.

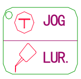
3.2.4 Lubrication Control

1. Non-automatic lubrication

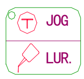
DT17 =0：**Non-automatic lubrication**。

When the status parameter NO.175.7 =1, the 

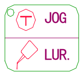
key in the panel is for spindle step feeding.

=0, the 

key in the panel is for lubricating.

When the No.112 of status parameter is set as 0, it is exchangeable output of lubrication, depress the 

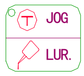
key on the panel, the lubrication will be output, press it again the lubrication will be cancelled. When the M32 is running, the lubrication is on, and then run the M33, it will be cancelled.

When the data parameter No.112>1, it is lubrication time-lapse output , depress the 

key on the panel, the lubrication is on, when the time which is set by the data parameter No.112 pass through, the lubrication will be cancelled; When the M32 is running, lubrication is on, when the time which is set by the data parameter No.112 pass through, the lubrication will be cancelled. If the time is not yet up, if the M33 is performed, then the lubrication will be cancelled.

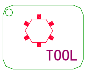
2. Automatic lubrication ：

DT17>0：**Automatic lubrication, the lubrication time and interval can be set by DT17 and DT16 separately.**

The lubrication will be on during the time which was set by DT17 when the power on of GSK980TD,after that the lubrication output will be stopped, and then after the interval which was set by DT16, the lubrication will be on again, the rest may be deduced by analogy. During the automatic lubrication, the M32, M33 dictates and the 

key on the panel are all unavailable.

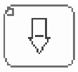
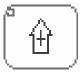
3.2.5 Manual Tool Change



: Under the manual operation, depress this key, the tool will be changed by manual (if the current tool is the No.1, depress this key, the tool will be changed to No. 2, if the current tool is the No.4, depress this key, the tool will be changed to No. 1).

3.2.6 Spindle Override Adjusting

Under the manual operation, when the simulative voltage output is selected to control the spindle speed, the spindle speed can be adjusted.

Depress the spindle override key  or  of 

, the speed can be changed by adjusting the spindle override, there are 8 levels of real time adjusting whose range is from 50％ to 120％ of spindle override can be realized.

Chapter 4 HANDWHEEL/ SINGLE STEP OPERATION

Under the handwheel/single step operations, the machine moves according to the selected increasement.

Attention

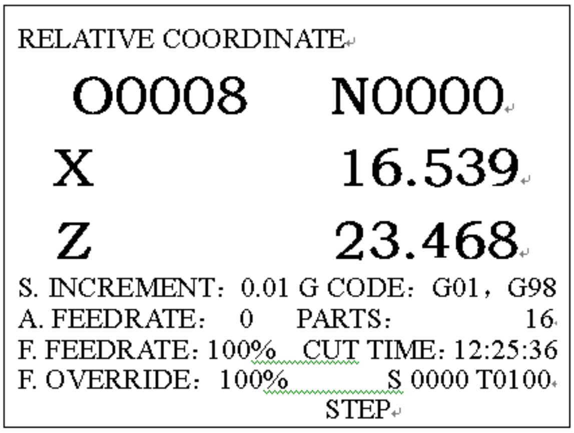
The function of each key on the operation panel of 980TD is defined by PLC programs (ladderlike chart), for the detail functions please refer to the manual which is offered by the machine builder.

The following mentioned functions are described according to the standard PLC program of 980TD, please take note!

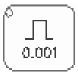
4.1 SINGLE STEP FEED

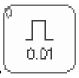
Set the Bit3 of system parameter as 0, enter the single operation mode by pressing the 

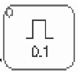
key, the page will be shown as below:

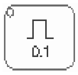


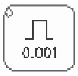
4.1.1 Increment Selecting

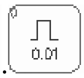
Press the 

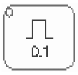
. 

or 

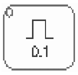
key to select the moving increment, the moving increment will be displayed on the page. When the Bit1 （SINC） of No.173 status parameter is set as 1, the step length value of 

is not effective; when the Bit1 is set as 0, 

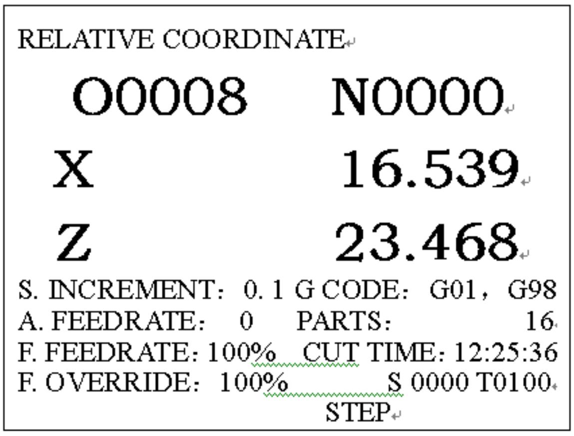
. 

. 

keys are all effective。

If the 

key is pressed, the page will be shown as below:



Note: Under other operation modes besides from Edit and handwheel/single step, the rapid traverse can be

modified by pressing the 0.001, 0.01, 0.1 keys, the corresponding overrides are Fo，50%，100%.

4.1.2 Moving Direction Selecting

By pressing the or key, the X axis will be moved forward or reverse according to the single increasement; By pressing the Z or Z key, the Z axis will be moved forward or reverse according to the single increasement.

4.2 HANDWHEEL FEED

Set the Bit3 of system parameter as 1, enter the handwheel operation mode by pressing the MPG key, the page will be shown as below:

RELATIVE COORDINATES

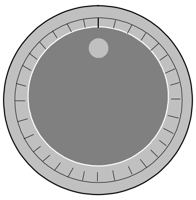
O0008 N0000

U 16.539

W 23.468

H. Increment: 0.001 G code: G01，G98
A. Feedrate: 0 Parts : 16
F. Feedrate: 100% Cut time: 12:25:36
R. Override: 100% S 0000 T0100
HW

The figure of handwheel are as follows:



The figure of the handwheel

4.2.1 Increasement Selecting

Select the increasement by pressing the 0.001, 0.01 or 0.1 key, the moving increasement will be shown on the page. When the Bit1 （SINC）of No.173 status parameter is set as 1, the step length value of 0.1 is not effective; when the Bit1 is set as 0, 0.001, 0.01, 0.1 are all effective. The page will be displayed as below if the 0.1 is pressed:

Book2 operation

```
RELATIVE COORDINATES
   O0008        N0000
   U            16.539
   W            23.468
H. Increment: 0. 1     G code:    G01， G98
A. Feedrate:       0   Parts  :          16
F. Feedrate:    100%   Cut time:   12:25:36
R. Override:  100%            S 0000 T0100
                    HW
```

4.2.2 The axis to be Moved and Direction Selecting

Select the corresponding axises by pressing or key under the manual operation mode. The page will be displayed as follows if key is pressed.

The handwheel feeding directions are decided by handwheel rotating directions. In general, it feeds forward when the handwheel rotates deasil, and it feeds reverse when the handwheel rotates widdershins. If it doesn't run as this, the A and B signals of the handwheel can be exchanged.

4.2.3 Other Operations

1.Spindle rotates forward/reverse and stop control

: Under the handwheel/single step operation, the spindle rotates forward by pressing this key.

: Under the handwheel/single step operation, the spindle stops by pressing this key.

: Under the handwheel/single step operation, the spindle rotates reverse by pressing this key.

2.Spindle step feeding

： When the Bit7 of No.175 status parameter is set as 1,depress the key to turn on the indicating LED, at this moment the spindle is under step feeding mode.

Under the spindle step feeding mode, depress the key, the spindle rotates forward on step feeding.The time and the speed of are set by No.108 and No.109 status parameters separately.

If the Bit7 of No.175 status parameter is set as 0 (at this moment the dey is lubrication switch) or when the LED of is off, the spindle step feeding is unavailable.

3.Coolant control

: Under the handwheel/single step operation, the coolant can be switched on/off by pressing this key.

4.Lubrication control

1) Non-automatic lubrication

DT17 =0：Non-automatic lubrication。

When the status parameter NO.175.7 =1, the key in the panel is for spindle step feeding.

=0, the key in the panel is for lubricating.

When the No.112 of status parameter is set as 0, it is exchangeable output of lubrication, depress the key on the panel, the lubrication will be output, press it again the lubrication will be cancelled. When the M32 is running, the lubrication is on, and then run the M33, it will be cancelled.

When the data parameter No.112>1, it is lubrication time-lapse output , depress the key on the panel, the lubrication is on, when the time which is set by the data parameter No.112 pass through, the lubrication will be cancelled; When the M32 is running, lubrication is on, when the time which is set by the data parameter No.112 pass through, the lubrication will be cancelled. If the time is not yet up, if the M33 is performed, then the lubrication will be cancelled.

2) Automatic lubrication ：

DT17>0：Automatic lubrication, the lubrication time and interval can be set by DT17 and DT16 separately.

The lubrication will be on during the time which was set by DT17 when the power on of the CNC,after that the lubrication output will be stopped, and then after the interval which was set by DT16, the lubrication will be on again, the rest may be deduced by analogy. During the automatic lubrication, the M32, M33 dictates and the key on the panel are all unavailable.

4. Manual tool change

: Under the handwheel/single step operation,, depress this key, the tool will be changed according to the sequence.

5. Spindle override adjusting

Under the handwheel/single step operation, when the simulative voltage output is selected to control the spindle speed, the spindle speed can be adjusted.

Depress the spindle override key or of , the speed can be changed by adjusting the spindle override, there are 8 levels of real time adjusting whose range is from 50％ to 120％ of spindle override can be realized.

4.2.4 Notes

1. The connections between the handwheel graduations and the machine moving distances are list as follows:

	The moving distance per graduation of handwheel		
Increasement of handwheel	0.001	0.01	0.1
The appointed coordinate value	0.001mm	0.01mm	0.1mm

2. The rotating speed of handwheel can not exceed 5 rounds per second, if it exceeds, the graduations and the distance may not matched.

3. The and key are only effective under the handwheel operation.

Chapter 5 MDI OPERATION

Under the MDI operation, parameter setting, dictates input and performing can be practiced.

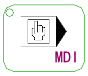
Attention

The function of each key on the operation panel of 980TD is defined by PLC programs (ladderlike chart), for the detail functions please refer to the manual which is offered by the machine builder.

The following mentioned functions are described according to the standard PLC program of 980TD, please take note!

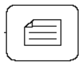
5.1 DICTATE INPUT

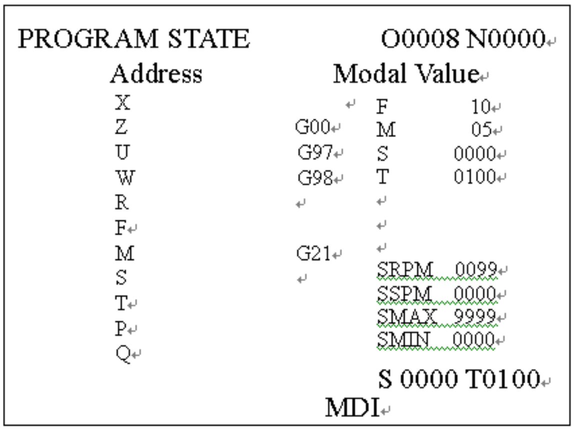
Select the MDI operation, then enter the program status page, and input a part program of G50 X50 Z100, the operation steps are as follows:

1. Enter the MDI operation by pressing the 

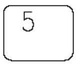
key;

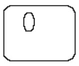
2. Enter the program status page by pressing the 

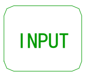
key（and press the  or  key if necessary）:



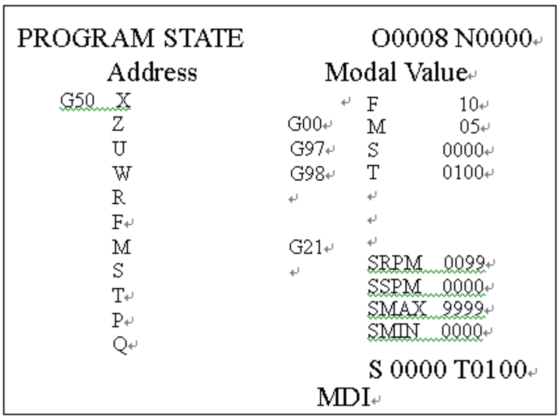
3. Input the address key 

,and numerical key 

, 

and 

key in sequence，the page will be displayed as follows：



Book2 operation

4. Input the address key Z, and numerical key 1, 0, 0 and INPUT key in sequence.

5. Input the address key X, and numerical key 5, 0 and INPUT key in sequence.

After all above operations are performed the page will be displayed as follows:

```
PROGRAM STATE                       O0008 N0000
        Address               Modal Value
  G50  X   50.000                   F        10
       Z   100.000         G00      M        05
       U                   G97      S        0000
       W                   G98      T        0100
       R
       F
       M                   G21
       S                            SRPM  0099
       T                            SSPM  0000
       P                            SMAX  9999
       Q                            SMIN  0000
                                    S 0000 T0100
                           MDI
```

5.2 DICTATE PERFORMING

After the dictates input, the MDI dictates can be performed by pressing the RUN key. During the processingthe MDI dictates can be stopped by pressing the PAUSE key or // key or emergency stop button.

Note: The transferred dictates of subprograms (M98 P___; etc.), and the compound cutting cycle dictates (such as G70, G71, G72, G73, G74, G75, G76, etc.) will not be effective under the MDI mode.

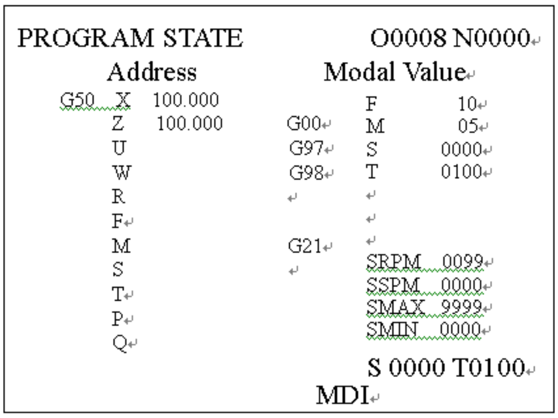
5.3 PARAMETER SETTING

Under the MDI mode, enter the parameter interface the parameter value can be modified, details please refer to Chapter 10.

5.4 DATA MODIFYING

Under the program status page of MDI mode, if there is something wrong during the dictates input before performing the input datas, all contents can be cleared by pressing the // key, and then input the correct datas again; Or input the correct one instead of the error again.Take the error input of X50 in section 5.1 of this chapter for example, if the correct one X100 needs to be input, then please press the address key X, and numeric keys 1, 0, 0 and INPUT key, instead of the error input of X50, the page will be displayed as below after all set:

Book2 operation



5.5 OTHER OPERATION

1. Perform the MDI dictates by pressing the 

key: when the Bit2 (DCS) of No.004 system parameter is set as 1, the current input dictates can be performed by pressing the 

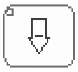
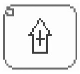
key.

2. Coolant control: Under the MDI operation, the coolant can be switched on/off by pressing 

key.

3. Adjustable spindle override

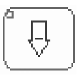
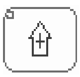
Under the handwheel/single step operation, when the simulative voltage output is selected to control the spindle speed, the spindle speed can be adjusted.

Depress the spindle override key  or  of 

, the speed can be changed by adjusting the spindle override, there are 8 levels of real time adjusting whose range is from 50% to 120% of spindle override can be realized.

4. Adjustable rapid traverse.

5. Adjustable feeding traverse.

Under the operation mode, Depress the  or  key of 

, the speed can be changed by adjusting the feeding traverse, there are 16 levels of real time adjusting whose range is from 0% to 150% of actual speed which appointed by F dictates can be realized.

6. Machine lock, auxiliary lock and dry running can be operated.

7. Automatic lubrication function (details please refer to chapter 3).

Chapter 6 PROGRAM EDIT and MANAGEMENT

Under the editing operation, program establishing, selecting, modifying, reproducing and deleting can be perform, it also can achieve two-way communication between CNC and CNC or CNC and PC.

Program switch is installed for GSK980TD to prevent the programs be modified or deleted accidentally.The program switch must be turned on before editing, detail setting of the program switch please refer to section 10.1.1.

To facilitate the management, GSK980TD provided a three-level user competence. Only 4 level or above operating level (4 levels, Level 3, etc.) are able to open the procedure switch and edit the programs, The allowed operation for each level please refer to section 10.3.

6.1 SET up PROGRAM

6.1.1 Generation of Program Sequence

In the program, the program sequence may added or not, the programs are performed by sequence of input time (transfer is a exception).

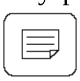
When the “automatic sequence” switch on the switch setting page is off, CNC can not generate the program sequence automatically, but the sequence can be added by hand during the program editing.

When the “automatic sequence” switch on the switch setting page is on, CNC generates the program sequence automatically, the sequence of next program will be generated automatically by pressing the EOB key, the increase value of the program sequence is set by the No.042 data parameter (the setting of automatic sequence please refer to section 10.1.1)

```
SWITCH SETTING                          O0008
N0000

   Parameter switch  :     *off     on
   Program switch    :     *off     on
   Sequence number   :     *off     on



                                S 0000 T0100
                           MDI
```

6.1.2 Input of Program Content

1. .Enter the Editing operation mode by pressing the EDIT key;

2. Enter the program interface by pressing the PROGRAM key, select the program contents displaying page by pressing the  or  key.

```
PRG.CONTENT LINE6 COLUMN1 O0008 N0000
O0008; (CNC PROGRAM. 20051020)
G50 X0 Z0;
G1 X100 Z100 F200;
G2 U100 W50 R50;
G0 X0 Z0;
X100 Z100;
M30;
%


                              S 0000 T0100
                         EDIT
```

3. Input the address key O ,and numeric key 0 , 0 , 0 , 1 in sequence（Take the example of setting up the program O0001）。

```
PRG.CONTENT LINE6 COLUMN1 O0008 N0000
O0008; (CNC PROGRAM. 20051020)
G50 X0 Z0;
G1 X100 Z100 F200;
G2 U100 W50 R50;
G0 X0 Z0;
X100 Z100;
M30;
%


  O0001                       S 0000 T0100
                         EDIT
```

4. Set up a new program by pressing the EOB key.

```
PRG.CONTENT LINE2 COLUMN1 O0001 N0000
O0001;（O0001）
;
%









                              S 0000 T0100
                         EDIT
```

5, Input the fixed programs one by one, the characters will be displayed on the screen immediately when they are input (For the compound key, please press it repeatedly to realize the alternate input), depress the EOB key to terminate a part program when it is finished.

6, The input of program or other part program can be performed according to step 5.

6.1.3 Searches of Characters

1. Scan: Scan the characters one by one with the cursor

Enter the editing operation mode by pressing the EDIT key, select the program contents displaying page by

Book2 operation

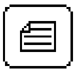
pressing the PROGRAM key.

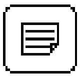
1) The cursor will be moved up a row by pressing the ⇧ key; If the current row is longer than the previous one, the cursor will be moved to the end of previous part program by pressing the ⇧ key (under the “;”).

2) The cursor will be moved down a row by pressing the ⇩ key; If the current row is longer than the previous one, the cursor will be moved to the end of next part program by pressing the ⇩ key (under the “;”).

3) The cursor will be moved to right for a character by pressing the ⇨ key; If the cursor is locating at the end of current row, it will be moved down to the head of next part program.

4) The cursor will be moved to left for a character by pressing the ⇦ key; If the cursor is locating at the head of current row, it will be moved up to the end of previous part program.

5) Turn over the previous page by pressing the  key, the cursor will be moved to the first character of previous page;If turn over to the first page of the program contents, the cursor will be moved the first character of the second row.

6) Turn over the next page by pressing the  key, the cursor will be moved to the first character of next page;If turn over to the last page of the program contents, the cursor will be moved the first character of the last row.

2. Search: Search over or down for the desired character from the current position of the cursor.
The operation steps are as follows:

1) Select the editing operation mode by pressing the EDIT key.

2) Display the program content page by pressing the PROGRAM key;

3) Enter the searching by pressing the CHANGE key, and input the desired characters with the max. 10 digits, the excess character will take place of the tenth character. For example, moves the cursor to G2, the page will be shown as below:

```
PRG.CONTENT LINE6 COLUMN1 O0008 N0000
O0008; (CNC PROGRAM. 20051020)
G50 X0 Z0;
G1 X100 Z100 F200;
G2 U100 W50 R50;
G0 X0 Z0;
X100 Z100;
M30;
%


 Searching G2                 S 0000 T0100
                          EDIT
```

Book2 operation

4) The page will be shown as below by pressing the ⇧ key (according to the ubiety between the desired character and current one to decide ⇧ key or ⇩ key to be pressed):

```
PRG.CONTENT LINE4 COLUMN1 O0008 N0000
O0008; (CNC PROGRAM. 20051020)
G50 X0 Z0;
G1 X100 Z100 F200;
G2 U100 W50 R50;
G0 X0 Z0;
X100 Z100;
M30;
%

 Searching G2                S 0000 T0100
                      EDIT
```

5) After the searching, the CNC is still on the seaching mode, press ⇧ key or ⇩ key again, it is available to search another character, or exit the searching by pressing the . CHANGE key.

6) “Search failure” will be displayed if there is no desired character.

Note: The character in the subprogram will not be searched during the searching.

3. Return to the head of program

1) The cursor will return to the head of program by pressing the . RESET key on the program displaying page under the editing operation mode.

2) Look up the first character of the program according to the method which described in section 6.1.3.

6.1.4 Character insert

Operation steps are as follows:

1) Select the program contents displaying page under the editing operation mode;

2) Enter the insert mode by pressing the INSERT ALTER key (the underline is the cursor) , the page is as follows:

Inserting state

```
PRG.CONTENT LINE4 COLUMN5 O0008 N0000
O0008; (CNC PROGRAM. 20051020)
G50 X0 Z0;
G1 X100 Z100 F200;
G2 U100 W50 R50
N30 G0 X0 Z0;
X100 Z100;
M30;
%

                             S 0000 T0100
                      EDIT
```

3) Insert the character (take above page as a example, insert the G98 dictate in front of the G2, input 9, 8, − ␣), the page will be shown as below;

Book2 operation

```
PRG.CONTENT LINE4 COLUMN5 O0008 N0000
O0008; (CNC PROGRAM. 20051020)
G50 X0 Z0;
G1 X100 Z100 F200;
G98 G2 U100 W50 R50
N30 G0 X0 Z0;
X100 Z100;
M30;
%



                              S 0000 T0100
                         EDIT
```

Note1: Under the insert mode, the blank will be generated automatically when the dictate address is input provided the cursor is not at the head of the row, if the cursor is at the head of the row, the blank must be inserted by hand as it will not be generated automatically.

Note2: Under the insert mode, the "0" will be added automatically when inputing the address provided there is a decimal infront of the cursor and the cursor is not at the end of the row.

Note3: Under the insert mode, the "0" will be added automatically follows the decimal by pressing the EOB key provided **there is a decimal infront of the cursor and the cursor is not at the end of the row.**

6.1.5 Character Deleting

Operation steps are as follows:

1) Select the program contents displaying page under the editing operation mode;

2) Delete the character in front of the cursor by pressing the CANCEL key; delete the character on the cursor by pressing the DELETE key.

6.1.6 Character Modifying

There are two ways of modifying the character

Insert modifying: Delete the character first, and then insert the desired character,details please refer to section 6.1.5.

Direct modifying: 1) Select the program contents displaying page under the editing operation mode;

2) Enter the modifying by pressing the INSERT/ALTER key (the cursor is an ashy rectangle), the page will be displayed as follows:

modifying state

```
PRG.CONTENT  LINE3  COLUMN1  O0008  N0000
O0008; (CNC PROGRAM.20051020)
G50 X0 Z0;
G1 X100 Z100 F200;
G98 G2 U100 W50 R50
N30 G0 X0 Z0;
X100 Z100;
M30;
%
                              S 0000 T0100
                       EDIT
```

3) Input the desired characters (take above page as a example, modify the X100 to U898 by input the U, 8, 9, 8)

```
PRG.CONTENT  LINE3  COLUMN8  O0008  N0000
O0008; (CNC PROGRAM.20051020)
G50 X0 Z0;
G1 U898 Z100 F200;
G98 G2 U100 W50 R50
N30 G0 X0 Z0;
X100 Z100;
M30;
%
                              S 0000 T0100
                       EDIT
```

Note1: Under the modifying, the current character will be modified as input one when inputing the character, and the cursor will be moved forward accordingly.

Note2: Under the modifying,if the cursor is on the "；",the input character will take place of "；", the next part program will be moved up to previous row. For example, the page will be displayed as below if input the "0".

```
PRG.CONTENT LINE3 COLUMN18 O0008 N0000
O0008; (CNC PROGRAM.20051020)
G50 X0 Z0;
G1 U898 Z100 F200;
G98 G2 U100 W50 R50
N30 G0 X0 Z0;
X100 Z100;
M30;
%
                        S 0000 T0100
                 EDIT
```

```
PRG.CONTENT LINE3 COLUMN19 O0008 N0000
O0008; (CNC PROGRAM.20051020)
G50 X0 Z0;
G1 U898 Z100 F2000G98 G2 U100 W50 R50;
G0 X0 Z0;
X100 Z100;
M30;
%
                        S 0000 T0100
                 EDIT
```

6.1.7 Single Block Deleting

This function is only feasible for the blocks with sequence numbers and the numbers are located at the head of the row, or the blocks with only blank in front of the sequence number.

Operation steps are as follows:

1) Select the program contents displaying page under the editing operation mode;

2) Move the cursor to the head of row where the block to be deleted (the first character), and press the DELETE **key.**

Note: If there is no sequence number for this block, input an "N" at the head of the row, and then move the cursor to the N , and press the DELETE key.

6.1.8 Part Programs Deleting

From the current character with the cursor, delete the appointed part programs (search down), the appointed block must have sequence number.

```
PRG.CONTENT LINE3 COLUMN9 O0008 N0000
O0008; (CNC PROGRAM.20051020)
G50 X0 Z0;
G1 U898 Z100 F200;
N30 G2 U100 W50 R50;
G0 X0 Z0;
X100 Z100;
M30;
%



                                S 0000 T0100
                            EDIT
```

Operation steps are as follows:

1) Select the program contents displaying page under the editing operation mode;

2) Enter the searching mode by pressing the CHANGE key, and then enter the part program sequence number;

```
PRG.CONTENT LINE3 COLUMN9 O0008 N0000
O0008; (CNC PROGRAM.20051020)
G50 X0 Z0;
G1 U898 Z100 F200;
N30 G2 U100 W50 R50;
G0 X0 Z0;
X100 Z100;
M30;
%



  Searching  N30                S 0000 T0100
                            EDIT
```

3) The page will be displayed as below by pressing the DELETE key:

```
PRG.CONTENT LINE3 COLUMN9 O0008 N0000
O0008; (CNC PROGRAM.20051020)
G50 X0 Z0;
G1 U898 X100 Z100;
M30;
%





                                S 0000 T0100
                            EDIT
```

Book2 operation

6.1.9 Block deleting

From the current character with the cursor, delete the appointed dictates.

```
PRG.CONTENT LINE3 COLUMN9 O0008 N0000
O0008; (CNC PROGRAM.20051020)
G50 X0 Z0;
G1 U898 Z100 F200;
G2 U100 W50 R50;
G0 X0 Z0;
X100 Z100;
M30;
%

                                S 0000 T0100
                          EDIT
```

Operation steps are as follows:

1) Select the program contents displaying page under the editing operation mode;

2) Enter the searching mode by pressing the CHANGE key, and then input the character;

```
PRG.CONTENT LINE3 COLUMN9 O0008 N0000
O0008; (CNC PROGRAM.20051020)
G50 X0 Z0;
G1 U898 Z100 F200;
G2 U100 W50 R50;
G0 X0 Z0;
X100 Z100;
M30;
%

  Searching W50                S 0000 T0100

                          EDIT
```

3) The page will be displayed as below by pressing the DELETE **key.**

```
PRG.CONTENT LINE3 COLUMN9 O0008 N0000
O0008; (CNC PROGRAM.20051020)
G50 X0 Z0;
G1 U898 R50;
G0 X0 Z0;
X100 Z100;
M30;
%

                                S 0000 T0100
                          EDIT
```

Note1: If there is no desired character or the desired character is in front of the cursor, the "search failure" will be displayed. If there are several the same desired character below, then the nearest one is the default.

Note2: If only input the dictate address, the following dictates will be deleted together.

6.2 PROGRAM REMARK

6.2.1 Set up the Program Remark

Operation steps are as follows:

1) Select the program contents displaying page under the editing operation mode;

2) The page will be displayed as below by pressing the CHANGE key.

```
PRG.CONTENT LINE2  COLUMN1  O0008  N0000
O0008; (CNC PROGRAM.20051020)
G50 X0 Z0;
G1 U898 Z100 F2000;
G2 U100 W50 R50;
G0 X0 Z0;
X100 Z100;
M30;
%



Searching                     S 0000 T0100
                         EDIT
```

3) Input the program remark (20 characters is the max.besides the brackets can be input), then the page will be displayed as below:

```
PRG.CONTENT LINE2  COLUMN1  O0008  N0000
O0008; (O0008)
G50 X0 Z0;
G1 U898 Z100 F2000;
G2 U100 W50 R50;
G0 X0 Z0;
X100 Z100;
M30;
%



Searching(CNC PRPGRAM.20051020)    S 0000 T0100
                         EDIT
```

4) The remark is set up by pressing the INPUT key, the page will be displayed as below:

```
PRG.CONTENT LINE2  COLUMN1  O0008  N0000
O0008; (CNC PRPGRAM.20051020)
G50 X0 Z0;
G1 U898 Z100 F2000;
G2 U100 W50 R50;
G0 X0 Z0;
X100 Z100;
M30;
%



                              S 0000 T0100
                         EDIT
```

Note1: If there is no remark added after set up the program, the default program name of CNC is the remark.

Note2: Only English remarks can be added in CNC, but the Chinese remark is also can be displayed (Chinese decimal is an exception). Adding Chinese remark can be realized as below: Download the Chinese remark which is edited in PC from the PC via communication software.

6.2.2 Program Remark Modifying

The operation steps are the same with the set up which is described in section 6.2.1.

6.3 PROGRAM DELETING

6.3.1 Single Program Deleting

Operation steps are as follows:

1) Select the program contents displaying page under the editing operation mode;

2) Input the address key O and numeric keys 0, 0, 0, 1 in sequence (Take program 00001 as an example);

3) The program O 0001 will be deleted by pressing the DELETE **key.**

6.3.2 All Programs Deleting

Operation steps are as follows:

1) Select the program contents displaying page under the editing operation mode;

2) Input the address key O and symbol key – and numeric keys 9, 9, 9 in sequence;

3) All programs will be deleted by pressing the DELETE key**.**

6.4 PROGRAM SELECTING

When there are several programs existing, there are three ways to select the program.

6.4.1 Searching

1) Select the editing or automatic operation mode;

2) Enter the program contents displaying page by pressing the PROGRAM key.

3) Press the address key O, and then input the program sequence number;

4) The desired program will be displayed on the screen by pressing the ⇩ key or EOB key, if the program does not exist,CNC alarm will occurs.

Note: In the step 4, If the program does not exist, CNC will set up a new program by pressing the EOB key.

6.4.2 Scaning

1) Select the editing or automatic operation mode;

2) Enter the program contents displaying page by pressing the PROGRAM key.

3) Press the address key O;

4) To display the next or previous program by pressing the ⇩ key or ⇧ key.

5) By repeat step 3 and step 4, the programs will be displayed one by one.

6.4.3 Cursor Confirming

1) Select the automatic operation mode (must in the non-running status)

2) Enter the program contents displaying page by pressing the PROGRAM key;

```
PROGRAM LIST                              O0008 N0000
    Version: GSK-980TD                      V05.10.20
    Part-prog No.: Most 384;          Used:        20
    Memory capacity: 6144 KB;         Used:   5310 KB
    Program list:
     O0000  O0002  O0003  O0004  O0005  O0006
     O0007  O0008  O0009  O0010  O0011  O0012
     O0014  O0023  O0088  O0089  O1000  O0044
     O0100  O0101


Program capacity: 16KB   Note: CNC PROGRAM.20051020
                                     S 0000 T0100
                               MDI
```

3) Move the cursor to the desired program name by pressing the ⇧, ⇩, ⇦ or ⇨ key (When the cursor is moving,"program size" and "remark" will be changed accordingly)

```
PROGRAM LIST                              O0008 N0000
    Version: GSK-980TD                      V05.10.20
    Part-prog No.: Most 384;          Used:        20
    Memory capacity: 6144 KB;         Used:   5310 KB
    Program list:
     O0000  O0002  O0003  O0004  O0005  O0006
     O0007  O0008  O0009  O0010  O0011  O0012
     O0014  O0023  O0088  O0089  O1000  O0044
     O0100  O0101


Program capacity: 16KB   Note: QIU TOU GAN
                                     S 0000 T0100
                               MDI
```

4) Press the EOB key.

```
PROGRAM LIST                              O0009 N0000
    Version: GSK-980TD                      V05.10.20
    Part-prog No.: Most 384;          Used:        20
    Memory capacity: 6144 KB;         Used:   5310 KB
    Program list:
     O0000  O0002  O0003  O0004  O0005  O0006
     O0007  O0008  O0009  O0010  O0011  O0012
     O0014  O0023  O0088  O0089  O1000  O0044
     O0100  O0101


Program capacity: 16KB   Note: QIU TOU GAN
                                     S 0000 T0100
                               MDI
```

Book2 operation

6.5 PROGRAM PERFORMING

Select the program to be executed according to section 6.4, select the automatic operation, the program will be run automatically (it also can be run by pressing the outer cycle running button if it is installed).

▪6.6 RENAME of PROGRAM

1) Select the program contents displaying page under the editing operation mode;

2) Input a new program name by pressing the address key O.

3) Press the INSERT ALTER key.

6.7 COPY PROGRAM

Save the current program in other place:

1) Select the program contents displaying page under the editing operation mode;

2) Press the address key of O, and then input a new program sequence number.

3) Press the CHANGE key.

6.8 PROGRAM MANAGEMENT

6.8.1 Program Content

Enter the program contents displaying page by pressing the PROGRAM key under the non-editing operation modes.on this page, the programs will be displayed as a directory list and the max. is 36 pieces in one page, if the quantity exceeds 36 pieces, the balance will be displayed on next page and they can be turned over by pressing the PROGRAM key.

```
PROGRAM LIST                          O0009 N0000
   Version: GSK-980TD                    V05.10.20
   Part-prog No.: Most 384;          Used:      20
   Memory capacity: 6144 KB;     Used: 5310 KB
   Program list:
    O0000  O0002  O0003  O0004  O0005  O0006
    O0007  O0008  O0009  O0010  O0011  O0012
    O0014  O0023  O0088  O0089  O1000  O0044
    O0100  O0101


Program capacity: 16KB    Note: QIU TOU GAN
                               S 0000 T0100
                           MDI
```

6.8.2 Soft Version

The current soft version will be displayed for this item.

6.8.3 Program Amount of Workpieces

The max. storage (max. 384 pieces) and saved program amount of workpiece will be displayed for this item.

6.8.4 Memory Size and Used Size

The total memory size of CNC (total 6144KB) and used size will be displayed for this item.

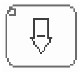
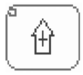
6.9 Other OPERATION under EDIT MODE

The panel operation could be defined by PLC program (Ladderlike chart) under the editing operation of GSK980TD, details please refer to the operation manual from the machine builder.

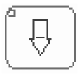
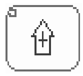
Please kindly note the functions described below are only for the standard PLC program of GSK980TD.

1. Coolant control, can be switched by pressing the 

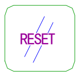
key.

2. Spindle override adjusting, can be realized by pressing the  or  key of 

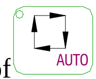
.

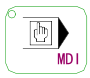
3. Feeding override adjusting, can be realized by pressing the  or  key of 

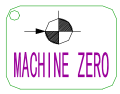
.

4. CNC reset, can be realized by pressing the 

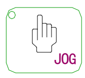
key.

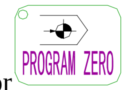
5. Operations switching can be realized by pressing any key of 

, 

, 

, 

, 

or 

.

6. Data transmission, details please refer to Chapter 11.

7. Automatic lubrication control (details please refer to Chapter 3)

Book2 operation

Chapter 7 TOOL OFFSET and TOOL ADJUSTING

To simplify the program editing, the actual positions of the tools can be ignored during the editing, there are three ways of tool adjusting including the fixed point, trial cutting and machine zero point return, the tool offset data can be collected by tool adjustings.

7.1 TOOL OFFSET INPUT by MOVING the TOOL to a FIXED POINT

Operations are as follows:

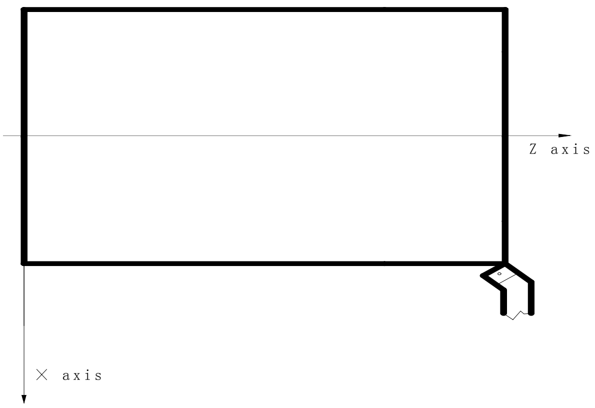


Fig. A

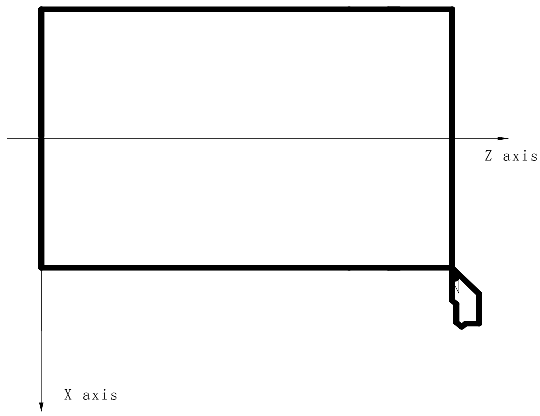
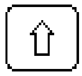
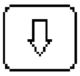
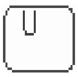
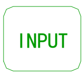
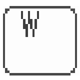


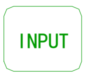
Fig. B

1. Please confirm if the tool offset of X and Z axises are zero, if not, please reset all of them.
2. Set the offset number as 00 (such as T0100，T0300)
3. Select a tool (normally this tool should be the first tool in process,select it as the standard tool)
4. Move the tool to a point (fixed point), please refer to chart A;
5. Set the workpiece coordinate by G50 X__ Z__ under the MDI operation,program status page;
6. Reset the related coordinate values U and W to 0;
7. Move the tool to a safe position and then select another tool, move this tool to the fixed point, Please refer to chart B;
8. Press the 

key, and then select the corresponding tool offset number by pressing the  or  key.
9. Press the 

key, and then press 

key, the offset value of X direction will be set to the corresponding offset number;
10. Press the address key 

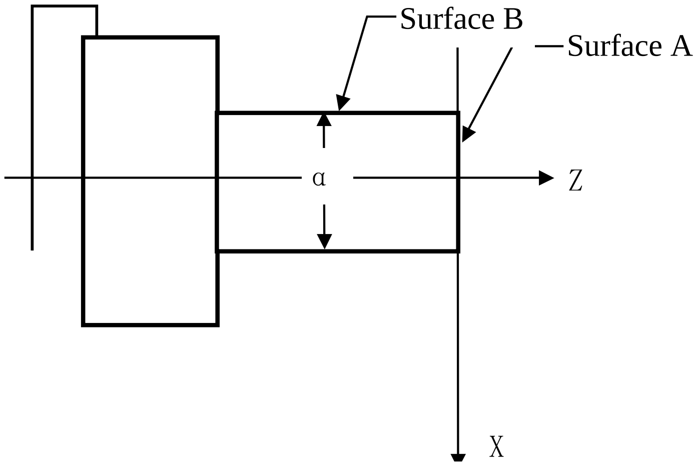
, and then press the 

key, the offset value of Z direction will be set to the corresponding offset number;
11. Other tools can be adjusted by repeat the step7 to step 10.

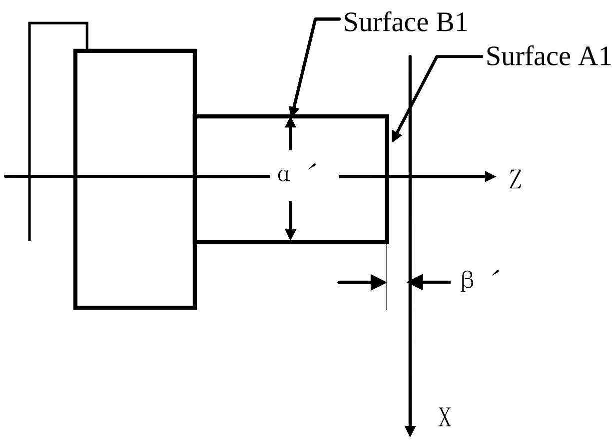
7.2 DIRECT INPUT of TOOL OFFSET by TRAIL CUTTING

It is effective or not for the trial cutting depends on the bit5 setting of No.012 CNC parameter.

The operation steps are as follows (Set up a workpiece coordinate on the cross section) :



1. Select a tool and cut the surface A ;
2. Draw back the tool along the X axis and remain the Z zxis, and then stop the spindle;
3. Enter the offset interface by pressing the [OFFSET] key, select the tool offset page, and then select the corresponding offset number by moving the [⇧] or [⇩] key;
4. Input the address key [Z], numeric key [0] and [INPUT] key in sequence;
5. Cut the surface B by this tool;
6. Draw back the tool along the Z axis and remain the X zxis, and then stop the spindle;
7. Measure the diameter “α” (suppose α =15)
8. Enter the offset interface by pressing the [OFFSET] key, select the tool offset page, and then select the corresponding offset number by moving the [⇧] or [⇩] key;
9. Input the address key [X], numeric key [1], [5] and [INPUT] key in sequence;
10. Move the tool to a safe position and then change another tool;



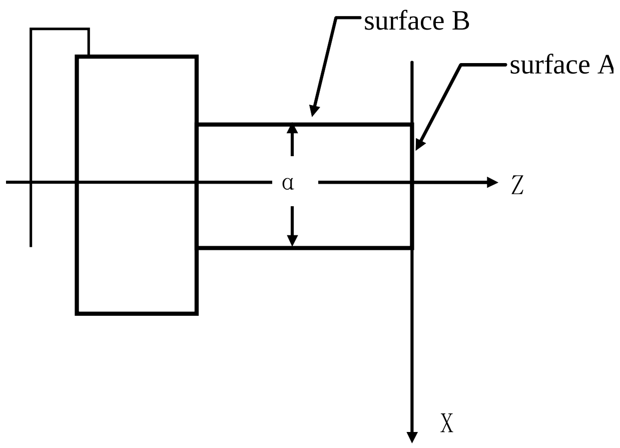
11. Cutting along surface A
12. Draw back the tool along the X axis and remain the Z zxis, and then stop the spindle;
13. Measure the distance "β´" between surface A and the reference point of workpiece coordinate (suppose β´ = 1)

14. Enter the offset interface by pressing the OFFSET key, select the tool offset page, and then select the corresponding offset number by moving the ⇧ or ⇩ key;
15. Input the address key Z, symbol key – and numeric key 1 and INPUT key in sequence;
16. Cut along surface B;
17. Draw back the tool along the Z axis and remain the X zxis, and then stop the spindle;
18. Measure the distance "α´" (suppose α´=10)
19. Enter the offset interface by pressing the OFFSET key, select the tool offset page, and then select the corresponding offset number by moving the ⇧ or ⇩ key;
20. Input the address key X, numeric key 1, 0 and INPUT key in sequence;
21. Other tools can be adjusted by repeat the step 10 to step 20.

Note: The offset value would be large by this way, so the tool compensation of CNC should be set based on the coordinate offset mode (The Bit4 of No.003 CNC parameter is set as 1), also, the tool length compensation of the first part program should be executed by dictate T, or dictate T is included in the first part program.

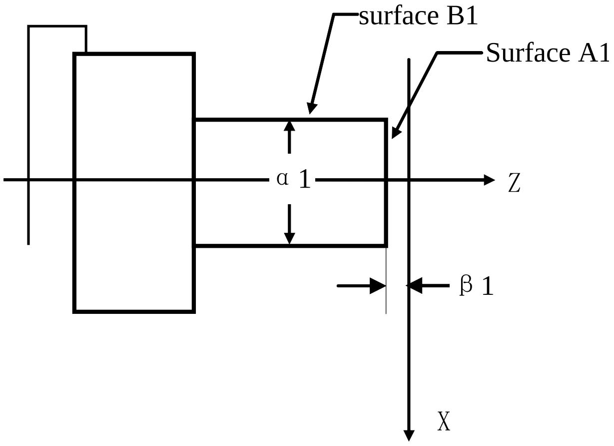
7.3 TOOL ADJUSTING by RETURNING to MACHINE ZERO POINT

There is no standard tool by this way, the tool can be re-adjusted when it is attrited or any tool needs to be re-adjusted. Before the tool adjusting please retun the tool to the machine zero point. In case of power off the process can be carried on after returning the machine zero point, it is easy to operated.



1. Enter the machine zero point return mode by pressing the MACHINE ZERO key, reset two axises to the machine reference point.
2. Select a tool and set the offset number as 00 (such as T0100，T0300)
3. Cut along surface A.
4. Draw back the tool along the X axis and remain the Z zxis, and then stop the spindle;

5. Enter the offset interface by pressing the OFFSET key, select the tool offset page, and then select the corresponding offset number by moving the ⇧ or ⇩ key;
6. Input the address key Z, numeric key 0 and INPUT key in sequence, the offset value of Z axis is set.
7. Cut along surface B.
8. Draw back the tool along the Z axis and remain the X zxis, and then stop the spindle;
9. Measure the distance "α´" (suppose α´=15)
10. Enter the offset interface by pressing the OFFSET key, select the tool offset page, and then select the corresponding offset number by moving the ⇧ or ⇩ key;
11. Input the address key X, numeric key 1, 5 and INPUT key in sequence, the offset value of X axis is set.
12. Move the tool to a safe position and then change another tool;
13. Select another tool and set the offset number as 00 (such as T0100，T0300)



14. Cut along surface A1.
15. Draw back the tool along the X axis and remain the Z zxis, and then stop the spindle; Measure the distance "β 1" between surface A1 and the reference point of workpiece coordinate (suppose β 1= 1)
16. Enter the offset interface by pressing the OFFSET key, select the tool offset page, and then select an offset number by moving the ⇧ or ⇩ key;
17. Input the address key Z, the symbol key – and numeric key 1 and INPUT key in sequence, the offset value of Z axis is set.
18. Cut along surface B1.
19. Draw back the tool along the Z axis and remain the X zxis, and then stop the spindle;
20. Measure the distance "α 1" (suppose α 1= 1)

Book2 operation

21. Enter the offset interface by pressing the OFFSET key, select the tool offset page, and then select an offset number by moving the ⇧ or ⇩ key;
22. Input the address key X and numeric key 1, 0 and INPUT key in sequence, the offset value of X axis is set.
23. Move the tool to a safe position;
24. All tools can be adjusted by repeat the step 15 to step 23.

Note1: The tool adjusting by returning to the machine zero point is only available based on the machine zero point is installed.

Note2: The workpiece coordinate could not be performed by G50 after the tool adjusting by returning to the machine zero point.

Note3: The tool compensation of CNC should be set based on the coordinate offset mode (The Bit4 of No.003 CNC parameter is set as 1), also, the tool length compensation of the first part program should be executed by dictate T, or dictate T is included in the first part program.

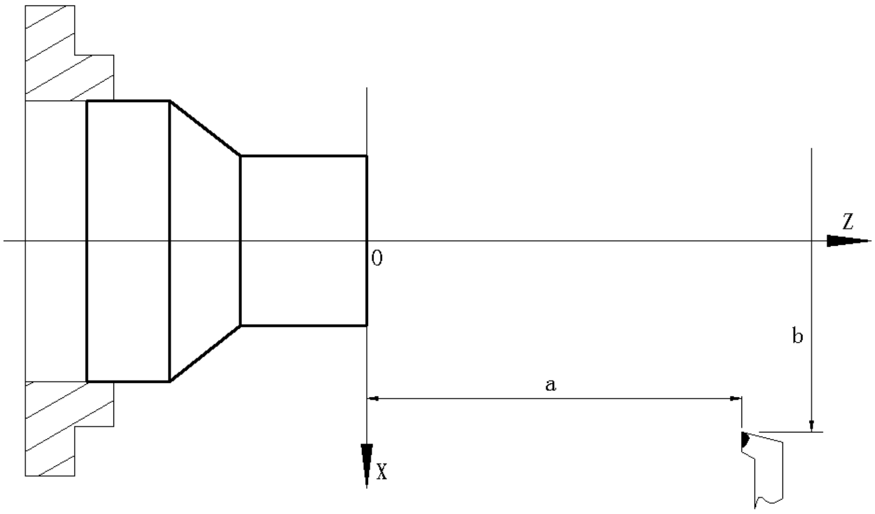
Note4: The corresponding parameters should be set as below:

The Bit7 of No.004 CNC parameter should be set as 0;

The Bit5 of No.012 CNC parameter should be set as 1;

The Bit7 of No.012 CNC parameter should be set as 1;

Note45: No.049 and 050 of CNC parameter should be set similar to the absolute coordinate values in the workpiece coordinate X0Z of the machine zero point, details please find below:



For example: After the machine zero point return, the absolute coordinate value is (a, b) for the tool in the workpiece coordinate, then the No.049 of CNC parameter should be similar to "a", and the 050 of CNC parameter should be similar to "b".

7.4 TOOL OFFSET MODIFYING

Enter the offset interface by pressing the OFFSET key, display the No. 000～No.032 offset number by pressing the  or  **key.**

TOOL OFFSET O0008 N0000

No.	X	Z	R	T
_000	0.000	0.000	0.000	0
001	-90.720	-116.424	0.000	0
002	0.000	0.000	0.000	0
003	0.000	0.000	0.000	0
004	0.000	0.000	0.000	0
005	0.000	0.000	0.000	0
006	0.000	0.000	0.000	0
007	0.000	0.000	0.000	0

Incremental coordinates
U 0.000 W 0.000
No. 000 S 0000 T0100
MDI

7.4.1 Absolute Value Input

1. Enter the offset interface by pressing the OFFSET key, display the desired page by pressing the  or  key.
2. Move the cursor to the position where the tool offset number should be input.

   Scan: Move the cursor in sequence by pressing the ⇧ and ⇩ key.

   P Q +offset number+ INPUT

   Searching: The cursor can be move to the desired position directly by pressing below key in sequence.

   P Q +offset number+ INPUT

3. After pressing the address key X or Z, in put the number (Decimal can also be input)
4. The tool offset value can be calculated automatically by CNC after pressing the INPUT key, and the result will be displayed on the page.

7.4.2 Increasement Input

1. Move the cursor to the desired position of tool offset number which to be modified according to the method described in section 7.4.1.
2. If the tool offset value of X axis has to be changed, please input U, and for Z axis, please input W.
3. Input the increasement;
4. Add the current tool offset value and the increasement by pressing the INPUT key, the result will be displayed as a new tool offset value.

   For example: The tool offset value of X axis is 5.678

   Input the increasement (U 1.5) by the keyboard

   Then the new tool offset value of X axis is 7.178(=5.678+1.5).

7.4.3 Tool Offset Modifying under the Communication Mode

Modifying and setting the tool offset value under the communication mode, detail operation please refer to Chapter 11.

Note1: The new tool offset value will be effective when the T code is executed after the tool offset value

change.

Note2: When the actual dimensions are not matched with the desired designed ones, if the actual one is bigger then please add the error value based on the original offset value,otherwise please reduce the error value.

For example: The desired diameter of the workpiece isΦ55.382, and the number 01 tool offset is effective, the tool offset are as follows before the process:

Sequence	X	Z	T	R
00	0	0	0	0
01	16.380	-24.562	0	0

After the process, the actual diameter isΦ55.561, then the number 01 tool offset can be modified as follows:

sequence	X	Z	T	R
00	0	0	0	0
01	16.559	-24.562	0	0

→ 16.380+(55.561−55.382)

7.4.4 Reset the Tool Offset Value

1. Move the cursor to the offset number which has to be reset.
2. Method 1:

To reset the tool offset value of X axis, please press the X and INPUT key in sequence, the value will be reset.

To reset the tool offset value of Z axis, please press the Z and INPUT key in sequence, the value will be reset.

Method 2:

If the current tool offset value in X direction is α, input the U-α and then press the INPUT, the value of X axis will be reset to 0.

If the current tool offset value in Z direction is β, input the W-β and then press the INPUT, the value of Z axis will be reset to 0.

Chapter 8 AUTOMATIC OPERATION

Attention

The function of each key on the operation panel of 980TD is defined by PLC programs (ladder like chart), for the detail functions please refer to the manual which is offered by the machine builder.
The following mentioned functions are described according to the standard PLC program of 980TD, please take note!

8.1 AUTOMATIC RUN

8.1.1 Automatic Run Selecting

1. Searching

1) Select the editing or automatic running operation;

2) Enter the program contents and display by pressing the PROGRAM key;

3) Press the address key O, and then input the program number;

4) The result programs will be displayed on the screen by pressing the ⇩ or EOB key.If the program does not exist,CNC alarm will be occurred.

Note: At the step 4, if the program does not exist, a new program will be set up by CNC by pressing the EOB key.

2. Scan

1) Select the editing or automatic running operation;

2) Enter the program contents and display by pressing the PROGRAM key;

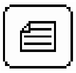
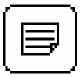
3) Press the address key O;

4) To display the next or previous program by pressing the ⇩ or ⇧ key.

5) Display the memorized programs one by one by repeat the c and d sdteps.

3. Cursor confirming

a) Select the automatic opearation (must under the non-running mode)

b) Enter the program content lisplaying page by pressing the PROGRAM key (Press the  or  key if necessary), (Details please refer to section 8.2.1)

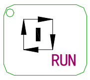
c) Move the cursor to the desired program name by pressing the ⇧, ⇩, ⇦, ⇨ key.

d) Press the EOB key.

8.1.2 Start Automatic Run

1. Select the automatic operation by pressing the AUTO key;

2. Start the process by pressing the RUN key, the program will be run automatically.

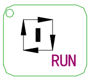
Note: The process will be started from the row which the cursor located in, so please check and confirm the whether the cursor locates in the desired part program before pressing the 

key, and if the whole program has to be run and the cursor is not in the first row please move the cursor back to the first row.

8.1.3 Stop Automatic Run

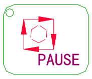
Stop dictates (M00)

After the part program with M00 executed, the stop auto running, modal functions and status are all been saved.

The program will be resumed by pressing the 

key or outer running key.

- **Stop by pressing related keys**

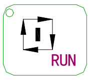
1, Press the the 

key or outer pause key during the auto running, the machine will:

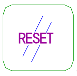
1) The machine feeding will be stop gradually;

2) When the pause dictat is executing (dictate G04), the machine will be paused after the execution.

3) Modal functions and status are saved;

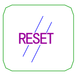
4) The process will be resumed by pressing the 

key.

2. Press the reset key 

.

1) All axises are stopped.

2) M, S functions output are of no effect (whether shut off the spindle rotates forward/reverse, lubrication and coolant signals automatically by pressing the 

key can be set by the the parameter);

3) When the automatic running is finished, the modal functions and status remain.

3. Emergency button

Depress the emergency button (when the outer emergency signal is effective) under the dangerous or emergency situations during the machine running, the CNC will come into emergency stop status, the moving of machine will be stopped immediately, all outputs (such as spindle rotates, coolant, etc.) will be shut off. CNC will come into reposition status by releasing the emergency button to release the alarm.

4. Operations switching

When the automatic running operation is switched to machine zero point return, handwheel/single step, manual or program zero point return, etc, the current part program will be “paused” immediately; when it is switched to editing and MDI modes, the part program will be remain running till end and then the “pause” will be displayed.

Note1: Please confirm all failures are settled before clearing the emergency stop alarm;

Note2: Depressing the emergency stop button before power on/off may reduce the power impacts to the equipment;

Note3: The machine zero point return should be re-done after the emergency stop alarm releasing, to make sure the correctness of the coordinate position (If the machine zero point is not installed, the operation can not be performed)

Note4: The outer emergency stop will only be effective based on the Bit3 (MESP) of No.172 status parameter is set as 0.

8.1.4 Automatic Run from any Part Program

1. Enter the editing operation mode by pressing the EDIT key, enter the program interface by pressing the PROGRAM key, then select the program content page by pressing the or key;
2. Move the cursor to the part program which is to be run (if start from row 4 then please move the cursor to the head of row 4);

```
PRG.CONTENT LINE4 COLUMN1 O0008 N0000
O0008;（CNC PROGRAM.20051125）
G50 X0 Z0;
M03 T0101;
G0 X30 Z2;
G01 X50 Z-30 F500;
W-100;
…
M30;
%
                              S 0000 T0100
                         EDIT
```

3. If the modals（Dictate G,M,T,F）are default in the part program which the cursor located at, and they are not matched with the desire of current part program, then the related modal functions must be executed before next step;
4. Enter the automatic operation mode by pressing the AUTO key, and then start the program by pressing the RUN key.

8.1.5 Feed and Rapid Traverse Adjusting

The running speed can be adjusted by Feeding and rapid traverse adjusting under the automatic running mode, and there is no need to change the speed values which are set in the program or parameter.

- Feeding override adjusting

FEEDRATE OVERRIDE

Depress the or key of , there are 16 levels of real time adjusting of feeding override can be realized.

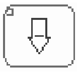
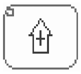
The feeding override will be added one step by pressing the key once, until 150％;

The feeding override will be reduced one step by pressing the key once, until 0;

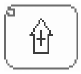
Note1: The appointed value by F in the feeding override adjusting program;

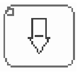
Note2: Actual feeding speed =Speed appointed by F*feeding override

- Rapid traverse adjusting

Depress the  or  key of 

, there are 4 levels of real time adjusting whose range with FO, 25%, 50%, 100% of rapid traverse can be realized.

The feeding override will be added one step by pressing the  key once, until 100%

The feeding override will be reduced one step by pressing the  key once, until 0.

Note1: The rapid traverses of X and Z axises are set by CNC parameter No.022 and No.023 separately;

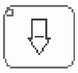
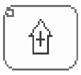
Actual rapid traverse of X axis= Value of No.022*rapid traverse

Actual rapid traverse of Z axis= Value of No.023*rapid traverse

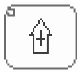
Note2: When the rapid traverse is F0, the lowest speed of the rapid traverse is set by No.032.

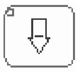
8.1.6 Spindle Speed Adjusting

Under the automatic running, the spindle rotate speed can be adjusted when the simulated voltage output is selected to control the spindle speed.

Depress the  or  key of 

, the speed can be changed by adjusting the spindle override, there are 8 levels of real time adjusting whose range is from 50% to 120% of spindle override can be realized.

The feeding override will be added one step by pressing the  key once, until 120%;

The feeding override will be reduced one step by pressing the  key once, until 50%.

Note1: Actual output of simulated voltage value= simulated voltage value which calculated by parameter*spindle override

For example:When the No.037 CNC parameter is set as 9999, and No.021 is set as 645, to execute the dictate S9999, and select the spindle override 70%, then the actual output simulated voltage is 10*70%=7V approximately.

Note2: The change of spindle override will be displayed under the left corner, the rapid override and spindle override can be switched by pressing the 

key.

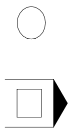
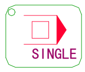
RELATIVE COORDINATES
O0008　　N0000
U　　16.539
W　　23.468
P. FEEDRATE:　0　　G code:　G01，G98
A. Feedrate:　0　　Parts ：　16
F. Feedrate:　100%　　Cut time:　12:25:36
R. Override: 100%　　S 0000 T0100
AUTO

RELATIVE COORDINATES
O0008　　N0000
U　　16.539
W　　23.468
P. FEEDRATE:　0　　G code:　G01，G98
A. Feedrate:　0　　Parts ：　16
F. Feedrate:　100%　　Cut time:　12:25:36
S. Override: 120%　　S 0000 T0100
AUTO

8.2 STATUS under RUN

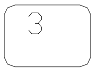
8.2.1 Single Run

To prevent the program error occurs at the first running, the single running can be selected. Under the automatic operation, the single program switch can be turned on by this way:

Method1: To light the single running indicator LED  by pressing the 

key in the status indicating area, the single running is selected;

Method2: Enter the machine soft panel page by pressing the 

key, the symbol "*" will be turned into the single program mode by pressing the numeric key 

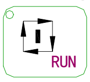
.

MACHINE CONTROL PANEL　　O0008 N0000

Machine locked	(key1) :	off	*on
M.S.T locked	(key2) :	off	*on
Single block	(key3) :	off	*on
Dry run	(key4) :	off	*on
Optional jump	(key5) :	off	*on

Incremental coordinates
U　372.815
W　-10.584

S0000 T0200
MDI

During the single running, CNC will be stopped when the current part program is executed; to run the next part program, the 

key should be pressed again,

Note1: When the dictate G28 is running, the single program will be stopped at the middle point;

Note2: When the fixed cycle dictates G90，G92，G94，G70 ~ G76 are running, the single status please refer to the Chapter 1 of <Editing manual>;

Note3: The single program will not be effective when the subprogram transferring (M98_) and subprogram transferring return (M99) are executing. But in the program with M98 and M99, the single stop is effective excluding the address N, O and P.

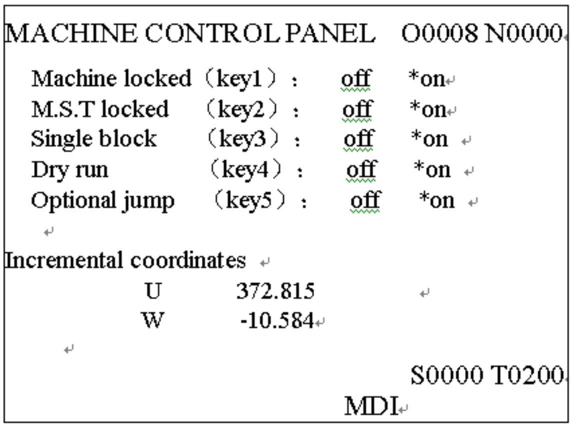
8.2.2 Dry running

To prevent the program error occurs at the auto running, the dry running can be selected to test the program before automatic running.

Under the automatic operation mode, the dry running can be turned on as follows:

Method1: To light the dry running indicator LED by pressing the SINGLE key in the status indicating area, the dry running is selected;

Method2: Enter the machine soft panel page by pressing the DIAGNOSIS key, the symbol "*" will be turned into the dry running mode by pressing the numeric key 4.



Under the dry running, machine feeding and auxiliary functions are available (providing the machine lock and auxiliary lock are off), namely, there is no impact to execute the machine feeding and auxiliary under the dry running, the appointed speed by the program is not effective, CNC will be run according to below table.

	Program dictates	
	Rapid traverse	Cutting
Rapid traverse on	Rapid traverse	The highest speed of manual feeding
Rapid traverse off	Manual feeding or rapid traverse (note1)	Manual feeding speed

Note1: Manual feeding speed or rapid traverse can be set by the Bit6 of No.004 CNC parameter.

Note2: Under the dry running, the rapid switching would not effect the current part program, it will be effective in next part program.

8.2.3 Machine Lock Run

Under the automatic operation, the machine lock switch can be turned on as follows:

Method1: To light the machine lock running indicator LED by pressing the MACHINE LOCK key in the status indicating area, the machine lock running is selected;

Method2: Enter the machine soft panel page by pressing the DIAGNOSIS key, the symbol "*" will be turned

Book2 operation

into the machine lock running mode by pressing the numeric key 1.

```
MACHINE CONTROL PANEL    O0008 N0000
  Machine locked （key1）：  off   *on
  M.S.T locked   （key2）：  off   *on
  Single block   （key3）：  off   *on
  Dry run        （key4）：  off   *on
  Optional jump  （key5）：  off   *on

Incremental coordinates
              U      372.815
              W      -10.584

                              S0000 T0200
                          MDI
```

Usually the machine lock running is processed with the auxiliary lock to test the program. Under the machine lock running:

1. The machine would not be moved, the “machine coordinate” in the integrated coordinate page of position interface remains, but the relative coordinate, absolute coordinate and distance to go will change at all times just as the same with the machine lock is off.
2. Ditates M, S, and T can be performed normally.

8.2.4 Auxiliary Lock Run

Under the automatic operation, the auxiliary lock switch can be turned on as follows:

Method1: To light the auxiliary lock running indicator LED MST by pressing the MST M. S. T. LOCK key in the status indicating area, the auxiliary lock running is selected;

Method2: Enter the machine soft panel page by pressing the DIAGNOSIS key, the symbol “*” will be turned into the auxiliary lock running mode by pressing the numeric key 2.

```
MACHINE CONTROL PANEL    O0008 N0000
  Machine locked （key1）：  off   *on
  M.S.T locked   （key2）：  off   *on
  Single block   （key3）：  off   *on
  Dry run        （key4）：  off   *on
  Optional jump  （key5）：  off   *on

Incremental coordinates
              U      372.815
              W      -10.584

                              S0000 T0200
                          MDI
```

At the moment dictates M,S and T would not be performed, the machine moves. Usually the machine lock running is processed with the auxiliary lock to test the program.

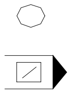
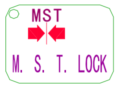
Note: There is no impact for M00, M30, M98, M99 executing when the auxiliary

lock is on.

8.2.5 Part Program Skipping

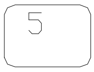
If there is a part program won't be executed or deleted, the part program skipping can be selected.The program with a symbol "/" on its head will be skipped under the automatic running if the part program switch is on (when the keys of machine panel or the outer input of part program skipping is effective)

Under the automatic operation, the part program skipping switch can be turned on as follows:

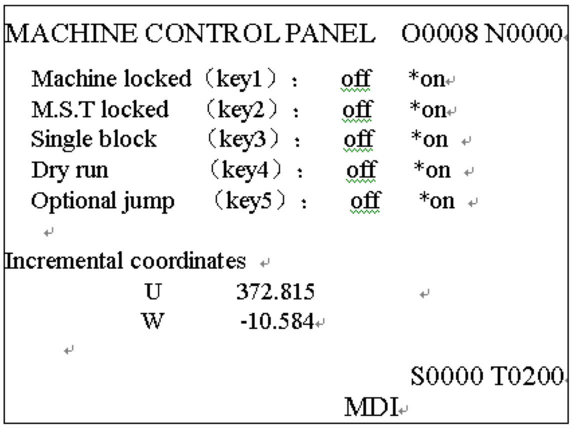
Method1: To light the part program skipping indicator LED  by pressing the 

key in the status indicating area, the part program skipping running is selected;

Method2: Enter the machine soft panel page by pressing the 

key, the symbol "*" will be turned into the auxiliary lock running mode by pressing the numeric key 

.

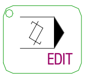


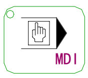
Note: When the part program skipping switch is off, the program with a symbol "/" on its head will not be skipped under the automatic running.

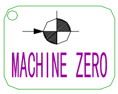
8.3 Other OPERATION

1. Under the automatic operation, the coolant on/off will be switched by pressing the 

key.

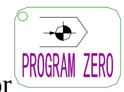
2. By pressing any key among 

, 

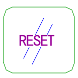
, 

, 

, 

or 

, operations can be switched.

3. CNC can be reset by pressing the 

key.

4. Automatic lubrication function (details please refer to Chapter 3)

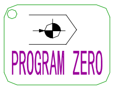
Chapter 9 ZERO POINT RETURN

9.1 PROGRAM ZERO POINT RETURN

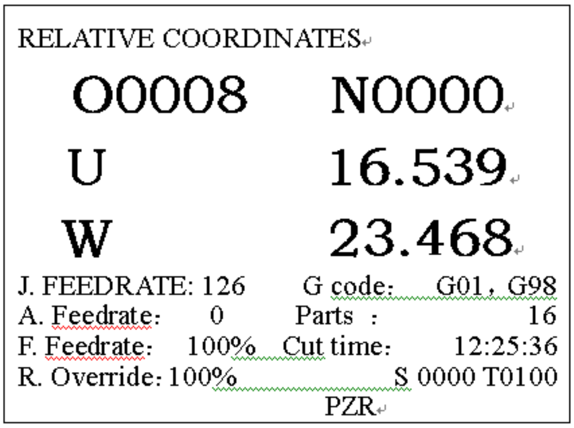
9.1.1 Program Zero Point

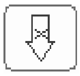
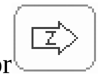
After the workpiece is installed on the machine, set the absolute coordinate of current tool position by dictate G50 according to the related position between the tool and the workpiece, the workpiece is set in the CNC. The current tool position is so-called **program zero point,** the tool will be return to it by perform the program zero point return.

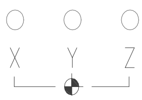
9.1.2 The Steps of Program Zero Point Return

1. Enter the program zero point return operation by pressing the 

key,



2. Select the program zero point return for X or Z zxis by pressing the  or  key.
3. The axis moves towards the program zero point, when it arrives, the axis will stops and the zero point return finish indicating LET will be on.



Indicator lamp for reference point return finished

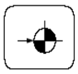
Note: After the zero point return operation, the current tool offset status would not be changed, the returned point is set by G50 and the tool offect is included if the tool offset is effective.

9.2 MACHINE ZERO POINT RETURN

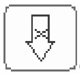
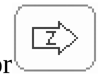
9.2.1 Machine Zero Point

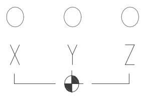
The machine coordinate is the standard coordinate for coordinate calculating of CNC, it is a original coordinate, the original point of it is so-called machine zero point (or machine reference point), the machine zero point is decided by the zero point switch or zero point return switch which is installed in the machine, usually the the zero point switch or zero point return switch is installed at the maximum positive journey of X and Z axises.

9.2.2 The Steps of Machine Zero Point Return

1. Enter the machine zero point return operation by pressing the  key, there will be "machine zero point return " displayed under the screen as follows:

```
RELATIVE COORDINATES
   O0008        N0000
   U            16.539
   W            23.468
J. FEEDRATE: 126    G code:    G01，G98
A. Feedrate:    0   Parts  :         16
F. Feedrate:  100%  Cut time:  12:25:36
R. Override: 100%         S 0000 T0100
                   MZR
```

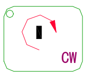
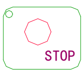
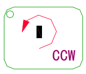
2. Select the machine zero point return for X or Z zxis by pressing the  or  key.
3. The machine moves towards the machine zero point, it will be returned to the zero point after the inspectings of slow down and zero point signal, the zxis stops at the moment, the LED of letter **[X], [Z]or [U], [W]** start to glitter, then the LED of zero point return is on.



Indicator lamp for machine zero return ending

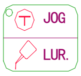
Note1: The machine zero point return can not be performed if the machine zero point is not installed.
Note2: The zero point return finish indicating LED will be off under below situations:
1) Moving from the zero point;
2) CNC power off;
Note3: The tool length compensation will be cancelled by CNC after the machine zero point return operation.
Note4: The related parameter of machine zero point return please refer to Chapter 4 <Connection>.

9.3 Other OPERATION under ZERO POINT RETURN

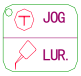
1. Spindle rotates forward by pressing the  key;
2. Spindle stops by pressing the  key;
3. Spindle rotates reverse by pressing the  key;
4. Coolant on/off can be switched by pressing the  key.
5. Lubrication control

1). Non-automatic lubrication

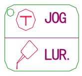
DT17 =0：Non-automatic lubrication.

When the status parameter NO.175.7 =1, the 

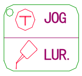
key in the panel is for spindle step feeding.

=0, the 

key in the panel is for lubricating.

When the No.112 of status parameter is set as 0, it is exchangeable output of lubrication, depress the 

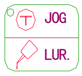
key on the panel, the lubrication will be output, press it again the lubrication will be cancelled. When the M32 is running, the lubrication is on, and then run the M33, it will be cancelled.

When the data parameter No.112>1, it is lubrication time-lapse output , depress the 

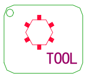
key on the panel, the lubrication is on, when the time which is set by the data parameter No.112 pass through, the lubrication will be cancelled; When the M32 is running, lubrication is on, when the time which is set by the data parameter No.112 pass through, the lubrication will be cancelled. If the time is not yet up, if the M33 is performed, then the lubrication will be cancelled.

2) Automatic lubrication

DT17>0： Automatic lubrication, the lubrication time and interval can be set by DT17 and DT16 separately.

The lubrication will be on during the time which was set by DT17 when the power on of GSK980TD,after that the lubrication output will be stopped, and then after the interval which was set by DT16, the lubrication will be on again, the rest may be deduced by analogy. During the automatic lubrication, the M32, M33 dictates and the 

key on the panel are all unavailable.

6. The tool can be changed by hand by pressing the 

key;

7. Spindle override adjusting

8. Rapid traverse override adjusting;

9. Feedrate override adjusting.

Chapter 10 SETTING, BACKUP and RESUMING of DATA

10.1 DATA SETTING

10.1.1 Switch Setting

Under the switch setting page, the on/off status of parameter switch, program switch and auto sequence can be displayed and set, the page are as follows:

```
SWITCH SETTING                         O0008
N0000

  Parameter switch  :    *off    on
  Program switch    :    *off    on
  Sequence number   :    *off    on



                               S 0000 T0100
                          MDI
```

1. Enter the setting interface by pressing the SETTING key, then enter the switch setting page by pressing the or key;

2. Move the cursor to the desired item by pressing the or key;

3. The swithch staus can be switched by pressing the D L or W key, the symbol "*" will be moved to left by pressing the W key,and the switch is turned off, the symbol "*" will be moved to right by pressing the D L key,and the switch is turned on,

The parameter modifying is only available based on the parameter switch is on; The program editing is only available based on the program switch is on; The auto sequence is only available based on the auto sequence switch is on;.

Note: CNC alarm will be occurred when the parameter switch is turning on, and it can be clear by pressing the // and CANCEL simultaneously, no alarm will be occurred if the parameter is switched again. For the sake of safety, please turn off the parameter switch after modifying.

10.1.2 Graph Setting

Enter the graph interface by pressing the SETTING key, then enter the graph parameter setting page by pressing the or key;

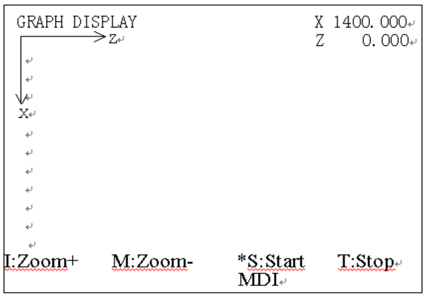
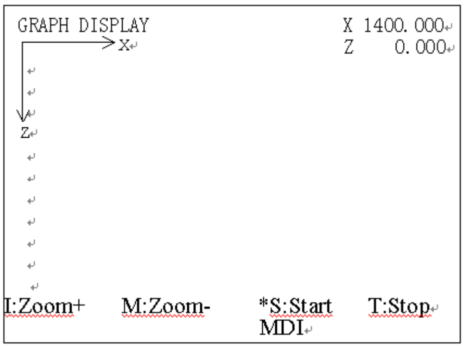
```
GRAPH SETTING                                  O0008
N0000
                     Graph parameter
  ■Coordinates=         1          (XZ:0     ZX:1)
   Scale       =     50.000
   Center      =    120.000        (X axis value)
   Center      =    160.000        (Z axis value)
   Max. X      =    240.000
   Max. Z      =    320.000
   Min. X      =      0.000
   Min. Z      =      0.000
   (Tool path display area: 320*240)
No.  001                             S 0000 T0100
                          MDI
```

A: Graph parameter setting

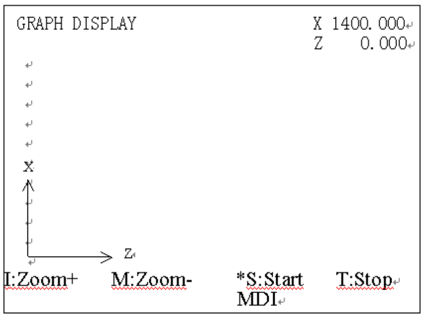
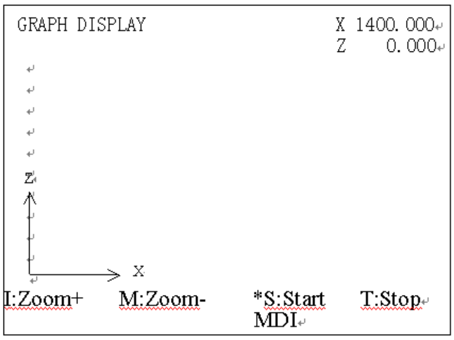
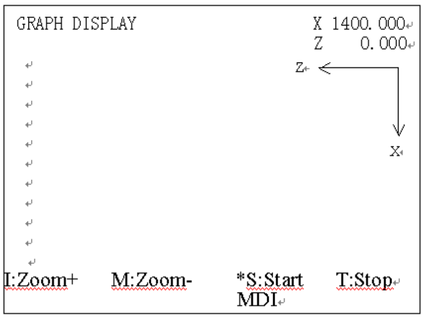
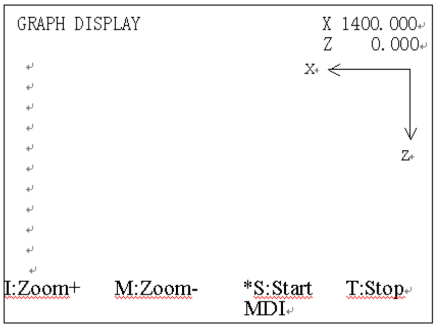
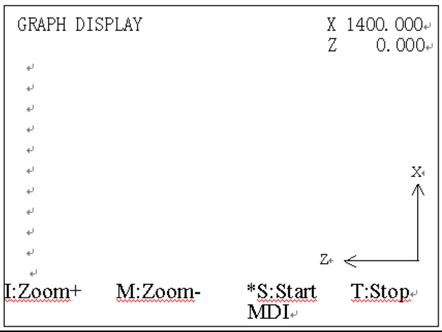
1. Under the MDI operation, Move the cursor to the desired item by pressing the ⇧ or ⇩ key;
2. Input the value;
3. The setting will be finished by pressing the INPUT key.

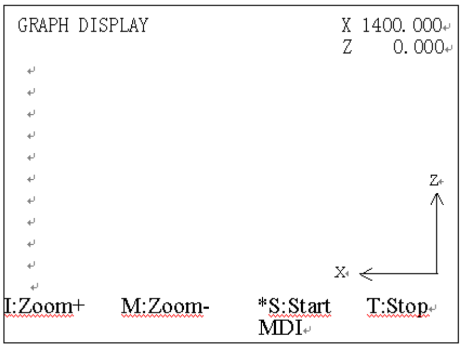
B: Meaning of the graph parameter

1. Setting of coordinates: There are 8 graph tracks of GSK980TD can be displayed according to the differences of front and back toolpost coordinates. Details are as follows:

Status parameter No.175		**Graph parameter**	**Graph of graph track coordinate**
Bit1	**Bit0**	**Coordinate selecting**	
0	0	0	
0	0	1	

Book2 operation

Status parameter No.175		Graph parameter	Graph of graph track coordinate
Bit1	Bit0	Coordinate selecting	
0	1	0	
0	1	1	
1	0	0	
1	0	1	
1	1	0	

Status parameter No.175		Graph parameter	Graph of graph track coordinate
Bit1	Bit0	Coordinate selecting	
1	1	1	

2. Scaling: Scaling setting
3. Graph center: the corresponding workpiece coordinate value setting under the LCD center.
4. The max. and min. values: CNC auto scaling and graph center auto setting are available after the max. and min. values of axises are set.

The max. value of X axis: The max. value of X axis during the process (unit:mm)
The min. value of X axis: The mIN. value of X axis during the process (unit:mm)
The max. value of Z axis: The max. value of Z axis during the process (unit:mm)
The min. value of Z axis: The mIN. value of Z axis during the process (unit:mm)

C: The zoom in and zoom out of the graph track

Under the graph displaying page, the real time aoom in or zoom out of graph tracks can be realized by pressing the I A or M .the graph will be zoom in at $\sqrt{2}$ times by pressing I A key once; and .the graph will be zoom out at $\sqrt{2}$ times by pressing M key once;

D: The starting, stop and clearing of graph track display

Under the graph displaying page, start the plot by pressing the S key, the symbol "*" will be moved in front of the character S; stop the plot by pressing the T key, the symbol "*" will be moved in front of the character T; the current graph track will be cleared by pressing the R V key.

10.1.3 Parameter Setting

The related characteristic of the driver, machine, etc, can be adjusted by parameter setting. Parameter details please refer to appendix 1.

Enter the parameter interface by pressing the PARAMETER key, the parameter pages can be switched by pressing the or key; details are as follows:

STATE PARAMETER O0008 N0000

No.	Data	No.	Data
001	00010001	009	00000010
002	11101000	010	00101111
003	01010100	011	10101000
_004	01000000	012	00110011
005	00010001	013	00000000
006	00000000	014	00000111
007	00000000	164	11100100
008	00000000	168	00000000

BIT0:1/0:reserved
*** RDRN DEC1 ORC TOC DCS PROD ***
No. 004 S0000 T0200
MDI

A、Status parameter modify setting

1. Character modifying

1). Turn on the parameter switch

2). Select the MDI mode;

3). Move the cursor to the desired parameter number:

Method1: Enter the the page of the parameter to be set by pressing the or key, Move the cursor to the desired parameter number by pressing the or key.

Method2: Press the address key ,parameter number and INPUT key.

4). Input new parameters;

5). The parameter value will be input and displayed by pressing the INPUT key;

6). For the sake of safety, the parameter switch needs to be turned off after all parameters are set.

For example:

Set the Bit5（DECI）of No.004 status parameter as 1, others remain.

Move the cursor to №004 according to above steps, input the 01100000 in sequence as follows:

STATE PARAMETER O0008 N0000

No.	Data	No.	Data
001	00010001	009	00000010
002	11101000	010	00101111
003	01010100	011	10101000
_004	01000000	012	00110011
005	00010001	013	00000000
006	00000000	014	00000111
007	00000000	164	11100100
008	00000000	168	00000000

BIT0:1/0:reserved
*** RDRN DEC1 ORC TOC DCS PROD ***
No. 004 =01100000 S0000 T0200
MDI

The parameter modifying will be finished by pressing the INPUT key. The page is as follows:

Book2 operation

STATE PARAMETER O0008 N0000

No.	Data	No.	Data
001	00010001	009	00000010
002	11101000	010	00101111
003	01010100	011	10101000
_004	01100000	012	00110011
005	00010001	013	00000000
006	00000000	014	00000111
007	00000000	164	11100100
008	00000000	168	00000000

BIT0:1/0:reserved

*** RDRN DEC1 ORC TOC DCS PROD ***

No. 004= S0000 T0200

MDI

2. Modifying by digits:

1). Turn on the parameter switch;

2). Select the MDI mode;

3). Move the cursor to the parameter number to be set;

Method1: Enter the the page of the parameter to be set by pressing the or key, Move the cursor to the desired parameter number by pressing the or key.

Method2: Press the address key P Q ,parameter number and INPUT key.

4). Press the hold the for 2 seconds or enter a parameter by pressing the CHANGE key, the cursor will located at the parameter, move the cursor to the one needs to be modified by pressing the or key, input 0 or 1 as requested;

5). For the sake of safety, the parameter switch needs to be turned off after all parameters are set.

Note: Enter a digit of a parameter, it is able to exit to the parameter number by pressing and holding key for 2 seconds or by pressing the CHANGE key.

For example:

Set the Bit5（DECI）of No.004 status parameter as 1, others remain.

Move the cursor to №004 according to above steps, Press the hold the for 2 seconds or enter a digit of parameter by pressing the CHANGE key, detaisl are as follows:

STATE PARAMETER O0008 N0000

No.	Data	No.	Data
001	00010001	009	00000010
002	11101000	010	00101111
003	01010100	011	10101000
004	01100000	012	00110011
005	00010001	013	00000000
006	00000000	014	00000111
007	00000000	164	11100100
008	00000000	168	00000000

BIT0:1/0:reserved

*** RDRN DEC1 ORC TOC DCS PROD ***

No. 004= S0000 T0200

MDI

Move the cursor to Bit5 by pressing the ⇨ or ⇦ key, details are as follows:

STATE PARAMETER O0008 N0000

No.	Data	No.	Data
001	00010001	009	00000010
002	11101000	010	00101111
003	01010100	011	10101000
004	01000000	012	00110011
005	00010001	013	00000000
006	00000000	014	00000111
007	00000000	164	11100100
008	00000000	168	00000000

BIT0:1/0:reserved

*** RDRN DEC1 ORC TOC DCS PROD ***

No. 004= S0000 T0200

MDI

Input 1, parameter modifying is done.

STATE PARAMETER O0008 N0000

No.	Data	No.	Data
001	00010001	009	00000010
002	11101000	010	00101111
003	01010100	011	10101000
004	01100000	012	00110011
005	00010001	013	00000000
006	00000000	014	00000111
007	00000000	164	11100100
008	00000000	168	00000000

BIT0:1/0:reserved

*** RDRN DEC1 ORC TOC DCS PROD ***

No. 004= S0000 T0200

MDI

B. Data parameter,worm offset data modifying setting

1. Data parameter modifying

1). Turn on the parameter switch;

2). Select the MDI mode;

3). Move the cursor to the parameter number to be set;

4). Input new parameter values

5). The parameter values will be input and displayed by pressing the INPUT key;

6). For the sake of safety, the parameter switch needs to be turned off after all parameters are set.

Example1: Set the №022 data parameter as 4000.

Move the cursor to №022 according to above steps, input 4000 in sequence as follows:

DATA PARAMETER O0008 N0000

No.	Data	No.	Data
015	1	023	7600
016	1	024	50
017	1	025	50
018	1	026	100
019	5	027	8000
020	2	028	500
021	645	029	100
_022	3800	030	10

Max. X rapid traverse speed(mm/min)

No. 022 =4000 S0000 T0200

MDI

The parameter modifying will be done by pressing the INPUT key. The page is as follows:

DATA PARAMETER O0008 N0000

No.	Data	No.	Data
015	1	023	7600
016	1	024	50
017	1	025	50
018	1	026	100
019	5	027	8000
020	2	028	500
021	645	029	100
_022	4000	030	10

Max. X rapid traverse speed(mm/min)

No. 022 = S0000 T0200

MDI

Example2: Set the X axis value of №000 worm offset data as 12, and 30 for Z axis.

Move the cursor to №000 worm offset data according to above steps, input X12 in sequence, detaisl are as follows:

PITCH ERROR COM. PARAMETER O0008 N0000

No.	X	Z
_000	55	32
001	-23	15
002	0	0
003	0	0
004	0	0
005	0	0
006	0	0
007	0	0

No. 002 X12 S0000 T0200

MDI

The data modifying is done by pressing the INPUT key, the page will be shown as follows:

PITCH ERROR COM. PARAMETER O0008 N0000

No.	X	Z
_000	12	32
001	-23	15
002	0	0
003	0	0
004	0	0
005	0	0
006	0	0
007	0	0

No. 002 S0000 T0200

MDI

The same, input Z30 in sequence and then press the INPUT key, the data modifying will be done, and the page will be shown as follows:

PITCH ERROR COM. PARAMETER O0008 N0000

No.	X	Z
_000	12	30
001	-23	15
002	0	0
003	0	0
004	0	0
005	0	0
006	0	0
007	0	0

No. 002 S0000 T0200

MDI

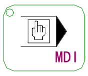
Book2 operation

10.2 RESUMING and BACKUP DATA

The user datas of GSK980TD (such as status parameter, data parameter, tool offset value and worm offset datas, etc.) can be backup(storage) and resumed(read). There is no impact for the CNC programs when the datas are backup and resume. The page is as follows:

```
PARAMETER OPERATION            O0008
N0000


    Backup par.    (machine factory)
    Resume par.    (machine factory)
    Resume default par.1 (test)
    Resume default par.2 (step)
   ■Resume default par.3 (servo)


Press [IN] + [P] to confirm(power on
again)

                          MDI
```

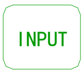
1. turn on the parameter switch;

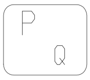
2. Enter the MDI operation by pressing the 

key, then enter the data backup page by pressing the 

key (press the  or  key if necessary).

3. Move the cursor to the item bo be operated;

4. Press the 

and 

simultaneously.

Note1: Avoid the power off during the data backup and resume, and donot proceed other operations before the operations are done;

Note2: The datas can be backuped and resumed are not the same among the current code level: user of 3,4,5 levels can proceed the status and data parameter backup and resume,but not change the worm offset parameter; User of 2 level can proceed the status,datas and worm offset parameter backup and resume.

For example: To resume the CNC parameter as servo standard parameter, the operation steps are as follows:

Turn on the parameter switch, and enter the MDI operation, data backup pages according to above steps, move the cursor to “Resume Default PAR.2 (step)”,details as follows:

```
PARAMETER OPRATION        O0008 N0000


    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).

    Successful to resume servo par.(power on
again)
                          MDI
```

Press the INPUT and P Q key simultaneously, "servo parameter resume successfully" will be displayed,(please restart).

10.3 PASSWORD SETTING and MODIFYING

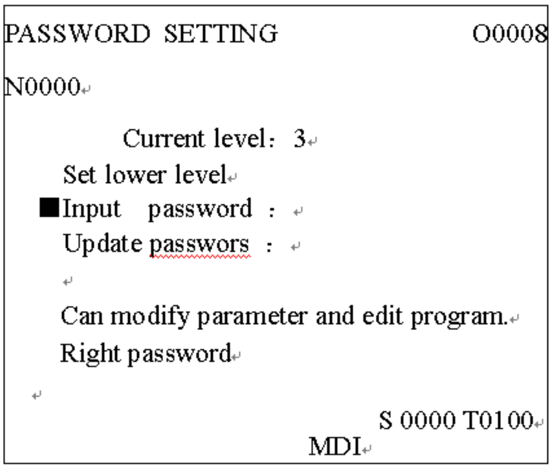
To prevent the programs and parameters be modified by others, GSK980TD offers password function, there are 4 levels falls as level 2 (machine builder),level 3 (equipment administrator), level 4 (technician) and level 5 (operator), the current level can be displayed on "current operation level__" under the password setting page.

Level 2: Machine builder, CNC status parameter,data parameter, worm offset data,tool offset data,program editing and PLC ladderlike chart transmission, etc, can be modified.

Level 3: The original password is 12345, CNC status parameter, data parameter, tool offset data and program editing can be modified.

Level 4: The original password is 1234, tool offset data (tool offset operation), macro veriable and program editing can be modified, CNC status parameter, data parameter and worm offset data cannot be modified.

Level 5: No password, only machine panel can be operated, tool offset data modifying, program selecting and program editing are unavailable, and CNC status parameter, data parameter and worm offset data cannot be modified.



Enter the password setting page, the cursor will stay at the row of "input password".Move the cursor to the related operation by pressing the ⇧ or ⇩ key.

a). The cursor will moves up a row by pressing ⇧ key once.If current cursor located at the row of "SET LOW LEVE" (The first row), By pressing the ⇧ key once, the cursor will be moved to "UPDATE PASSWORD" (last row).

b) The cursor will moves up a row by pressing ⇧ key once. If current cursor located at the last row, By pressing the ⇧ key once, the cursor will be moved to the first row.

10.3.1 Enter Operation Level

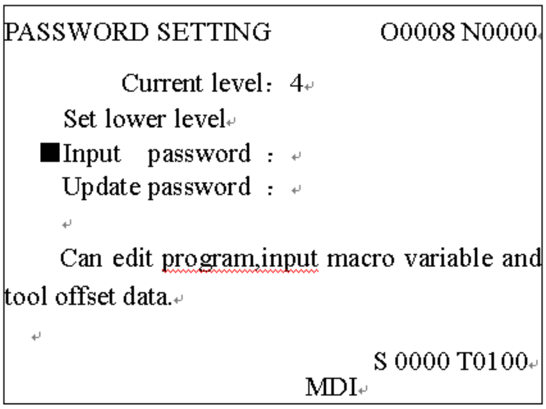
1. Enter the password setting page, move the cursor to the row of "input password";
2. Input password ("*" will be displayed for each input)

3. Press the INPUT key after all set, then enter the corresponding operation level.

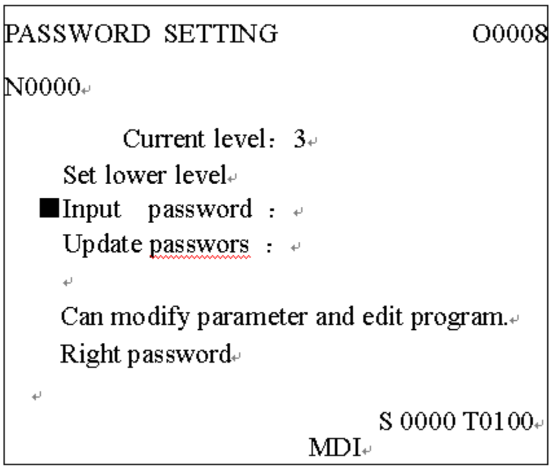
Note: The password digits of GSK980TD are corresponding to the operation level, users can not increase or reduce the digits, details please refer to below:

Operation level	Password digits	Original password
Level 3	5 digits	12345
Level 4	4 digits	1234
Level 5	None	None

For example: The current operation level is 4, the page is as follows. Password of level 3 is 12345, please change the operation level to 3.



Move the cursor to the row of “INPUT PASSWORD”, press the INPUT key after inputing 12345, “parameter modifying and program editing are available” “password is correct ” will be displayed, then the current will be change to level 3. the page will be displayed as follows:



Note: The level will remain when the power return on if the current operation level are lower than level 3 (including level 3, namely, level 3, 4, 5). If the level of last operation is higher than level 3 (level 0, 1, 2), then the default is level 3 when the power return on.

10.3.2 Password Change

The steps of password change are as follows:

1. Enter the password setting page, input the password according to section 10.3.2;
2. Move the cursor to the row of “UPDATE PASSWORD”;

3. Input new password, then press the INPUT key;
4. "Please input the new password again" will be displayed, details please find below:

```
PASSWORD SETTING              O0008 N0000

        Current level: 3
    Set lower level
  ■Input    password  :
    Update password  :

    Can edit program and modify parameter.
    Please input new password again.

                                S 0000 T0100
                           MDI
```

5. Press the INPUT key again after re-input the new password, if both input passwords are matched, the "password updated" will be displayed, the password is updated successfully.

```
PASSWORD SETTING              O0008 N0000

        Current level: 3
    Set lower level
  ■Input    password  :
    Update password  :

    Can edit program and modify parameter.
    Password has been updated.

                                S 0000 T0100
                           MDI
```

6. If the input passwords are not matched, "Password unmatched, please input again" will be displayed:

```
PASSWORD SETTING              O0008 N0000

        Current level: 3
    Set lower level
  ■Input    password  :
    Update password  :

    Can edit program and modify parameter.
    Password  verifying  is  incorrect,  please
input password again.
                                S 0000 T0100
                           MDI
```

10.3.3 Set Lower Level

The operation is for user to set one level lower than current level, operation steps are as follows:

1. Enter the password setting page, input the password according to section 10.3.2;
2. Move the cursor to the row of "SET LOWER LEVEL", if the current level is 3, the page is as follows:

```
PASSWORD SETTING                O0008 N0000
          Current level: 3
     Set lower level
  ■Input    password  :
     Update password  :

     Can edit program and modify parameter.


                                S 0000 T0100
                         MDI
```

3. "Set current level to level 4, confirm?" will be displayed by pressing the INPUT key, the page will be shown as follows:

```
PASSWORD SETTING                O0008 N0000
          Current level: 3
     Set lower level
  ■Input    password  :
     Update password  :

     Can edit program and modify parameter.
     Set current level to 4,sure?

                                S 0000 T0100
                         MDI
```

4. The set lower level setting will be done by pressing INPUT again, the page will be shown as follows:

```
PASSWORD SETTING                O0008 N0000
          Current level: 4
     Set lower level
  ■Input    password  :
     Update password  :

     Can edit program,input macro variable and
tool offset data.

                                S 0000 T0100
                         MDI
```

Note: Set lower level can not be done if level 5 is the current level.

Chapter 11 COMMUNICATION

11.1 Introdcution of the GSK980TD communication software, TDComm2a

TDComm2a can realize the file uploading and downloading between PC and CNC, it is easy to operate, and is efficacious in communication and relierable.

- **System (PC) requirement for TDComm2a**

Hardware: PC with RS232 serial-port interface, serial communication cable, (3 wires)

Operation System: Microsoft Windows 98/2000/XP/2003

- **Software Interface**

The software interface of TDComm2a is simple, the following Fig.ture is the interface of the software when running.

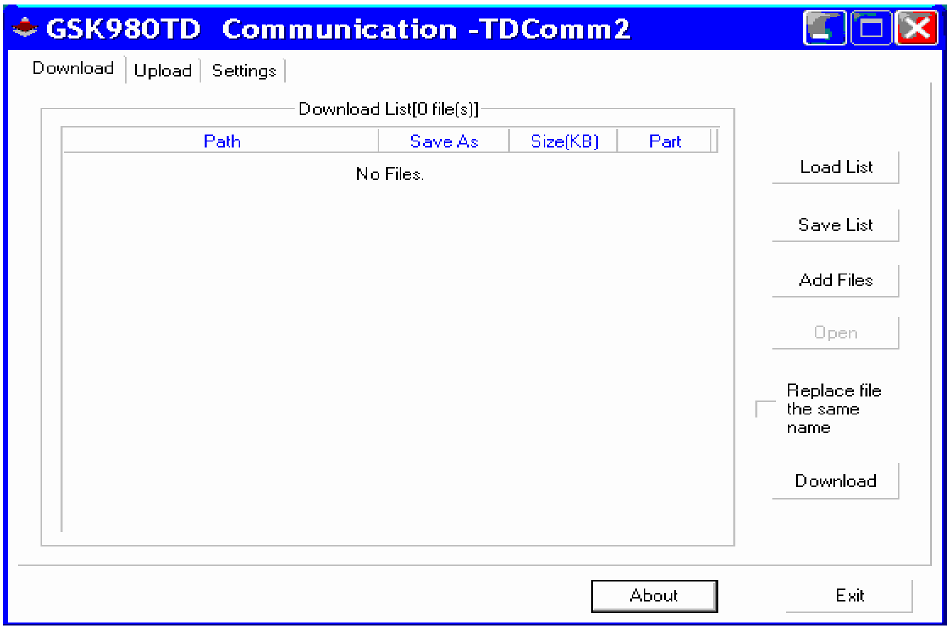


Fig. 3-1 File downloading interface (PC→CNC)

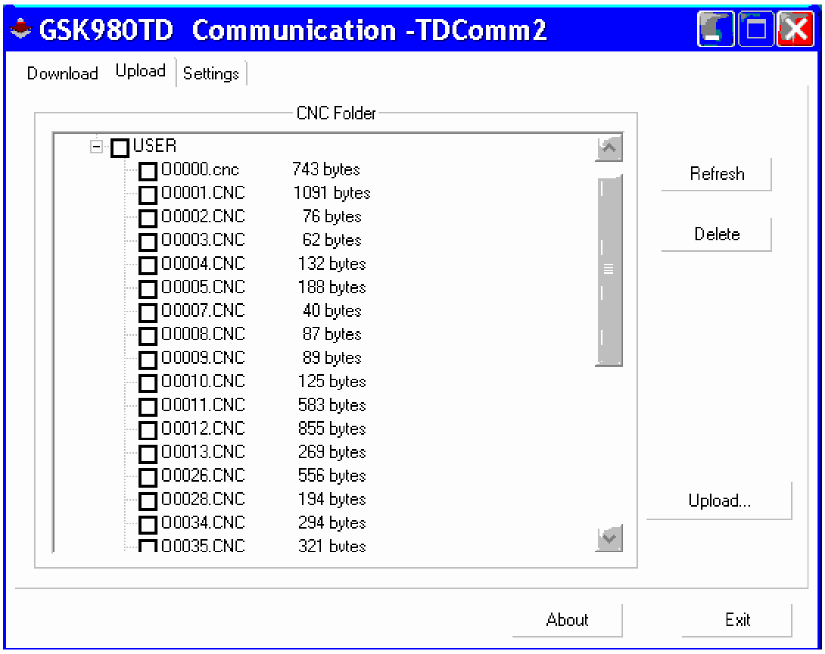


Fig. 3-2 File uploading interface (CNC→PC)

Book2 operation

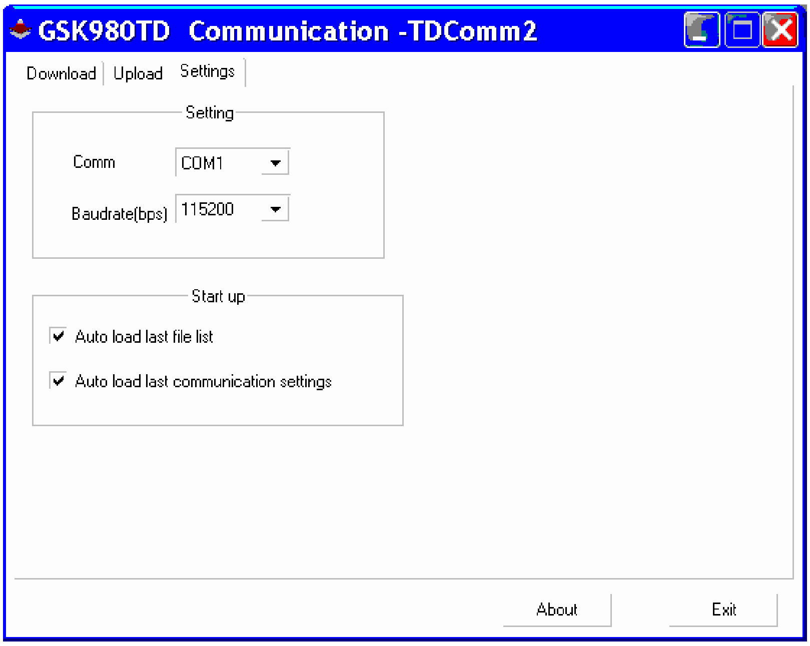


Fig. 3-2 Setting option interface

11.1.1 File Downloading（PC→CNC）

When downloading the files, press the button “Add File”, select to the list box the to-transfer Files to CNC, list the information such as file path, file name on CNC, file length, save area on CNC. You can save the list into files, so that the next time you use the programme to transfer the same file, you can open the list file, you don’t need to reselect the file.

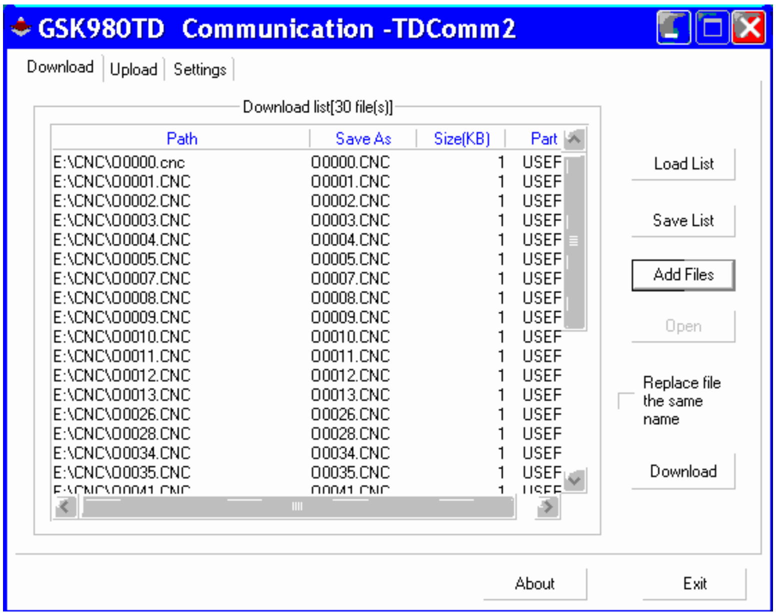


Fig. 3-4

As shown in Fig. 3-4, on the left is the disply box of the file list, on the right are the five buttons “Load to List”, “Save List”, “Add File”, “Check Source File”, “Start Downloading”, and the option for “overwrite the file with the same file name”

File list display box: When downloading files, it is the list box of the to-transfer file to CNC, listing the information such as file path, file name on CNC, file length, save area on CNC. You can save the list into files, so that the next time you use the programme to transfer the same file, you can open the list file, you don’t need to reselect the file.

Load to List: Load the download list file saved on the harddisk.

Save List: Save the current file list to harddisk as file.

Add File Add a file to the to-transfer file list from harddisk

Start Downloading: Start the file transferring after selecting the to-transfer files.

Check Source File: Open the selected file in text mood.

Overwrite the file with the same file name: Overwrite the file without asking if there are files with the same names on the CNC during the file transferring.

▲ **Select File Dialog Box**

Press the "Add File" botton as shown at the mouse pointer in Fig.3-4, appears the dialog box "Please select the file to send…", choose the to-transfer files in the dialog box, (can be multi-select) press botton "All CNC File", you can select all the CNC files under the directory to the file list at a time. The default file names saved to the CNC from the file list are the same as the original file names, the file names will cut to 8 characters automatically when the file name lengths are over 8 characters.

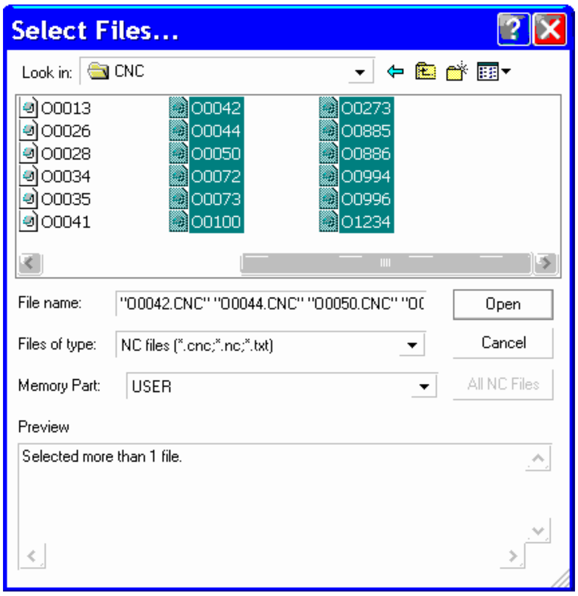


Fig. 3-5

Annotate: The file names saved in the CNC can't be Chinese Charactors, double click the file list item in the file list to modify and save the file name.

When selecting a single file, we can get the preview of the file content in the bottom of the dialog box, as shown in Fig. 3-6.

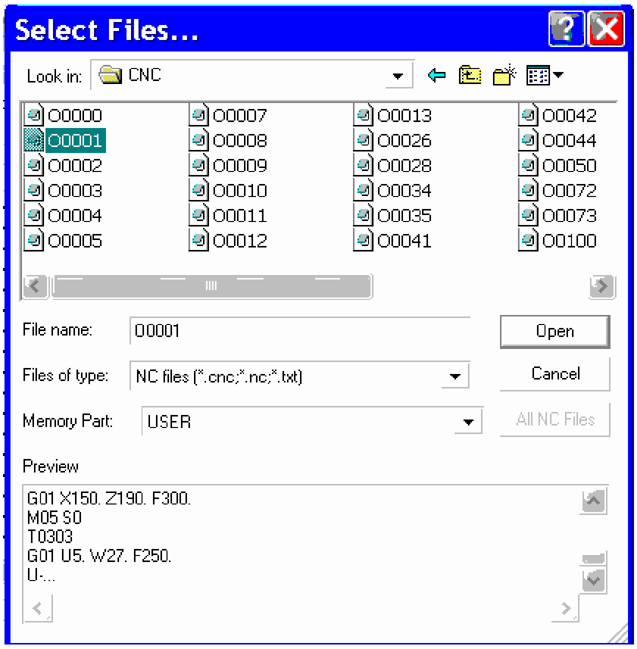


Fig. 3-6

▲ Modifying file list property

When we need to modify file list item property (file path, saved filed name and save area), double click the file list item to show the setting dialog box, as shown in Fig. 3-7, Fig. 3-8, Fig. 3-9.

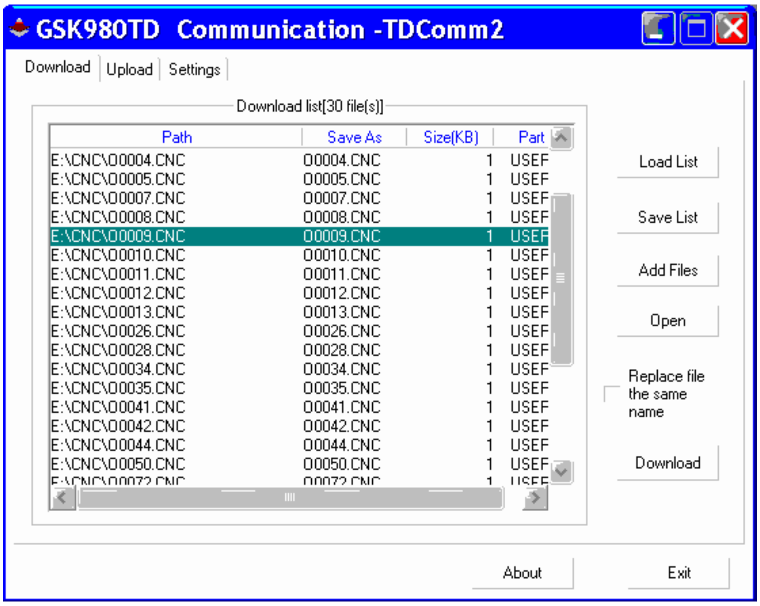


Fig. 3-7

Suppose we need to modify the saved file name of the highlighting item in the list to “O0001”, we can perform the following operation.

Move the mouse point to the row of the file list item, as shown in Fig. 3-7, double click left botton and appears the setting dialog box as Fig. 3-8, we can modify the file path and the file name (Fig. 3-9).

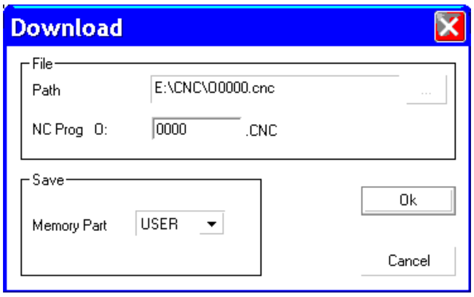


Fig. 3-8

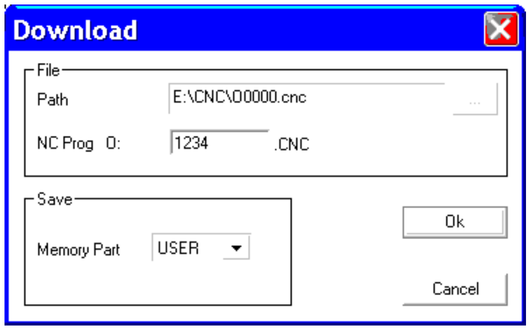


Fig. 3-9

Press OK and the file list is set OK. As shown in Fig. 3-10.

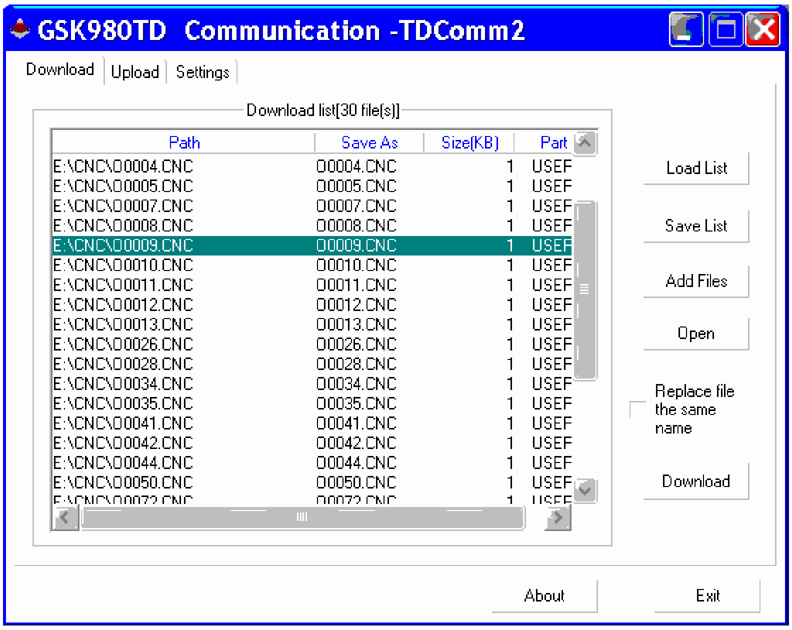


Fig. 3-10

We can add to-transfer file to the file list like this one by one.

Moreover, we can double click left mouse button on the item in the file list, appears the file setting dialog box as Fig. 3-7, change file name, saved area and such settings, select the list item and right click on the list tem, appears a menu, we can perform the delete the item or empty the list operation as the following Fig. 3-11.

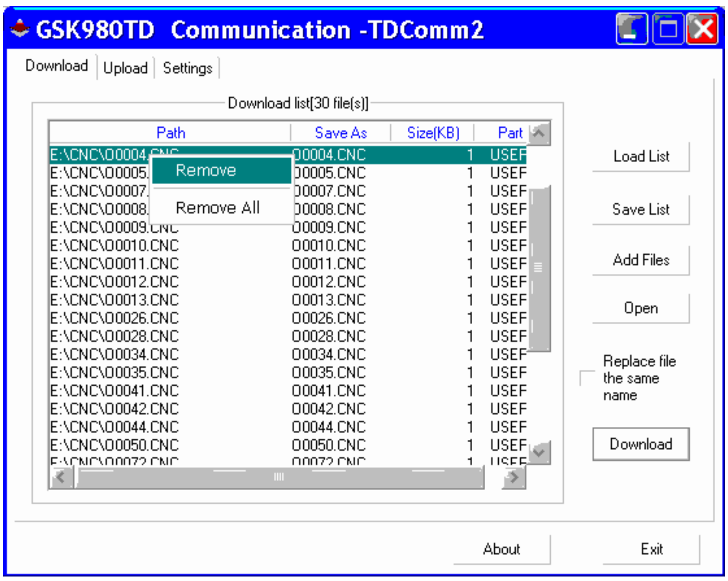


Fig. 3-11

When finished the file adding, we can press the “Save list” botton to save the list as a file, so that next time we can add to-transfer files by “Load list” directly, no need to set the file list one by one again, as Fig. 3-12.

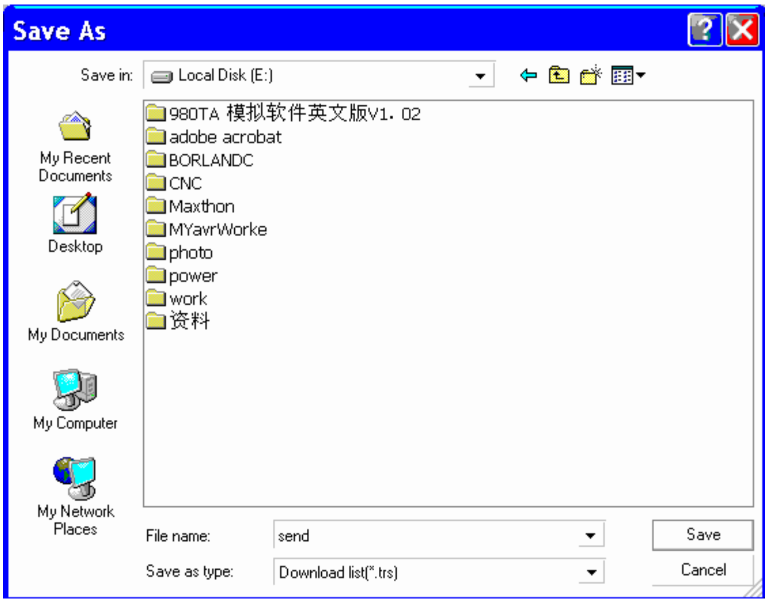


Fig. 3-12

Annotation: The file name in CNC can just be 8.3 format (8 letter or digit character as file name, 3 letter or digit character as extended name), can’t be Chinese charactors or other charactors. Obey this rule when downloading files, modifying the file names or setting CNC file names. If the file name is not accord with the rule, the item will be shown in red in the file list, modify it to this rule.

after the file list is done, we can press the “ Start Download” button, download the files and appears the dialog box of communication statues, in the dialog box, we can check the information of the transferring file, process and communication statue. (Fig. 3-13).

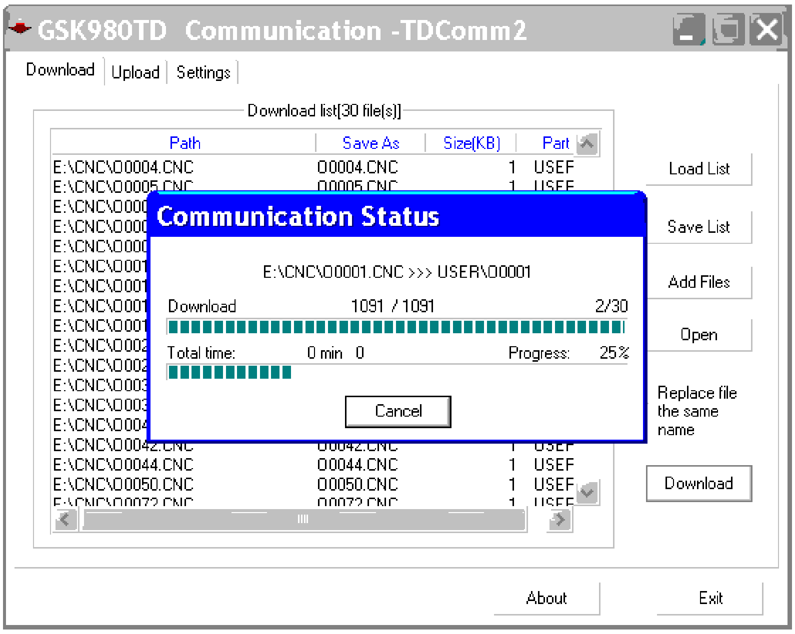


Fig. 3-13

If there are files with the same file names on the CNC, the dialog box will shows, we can choose to overwrite the files or skip the files and transfer, and perform corresponding operation. (Fig. 3-14).

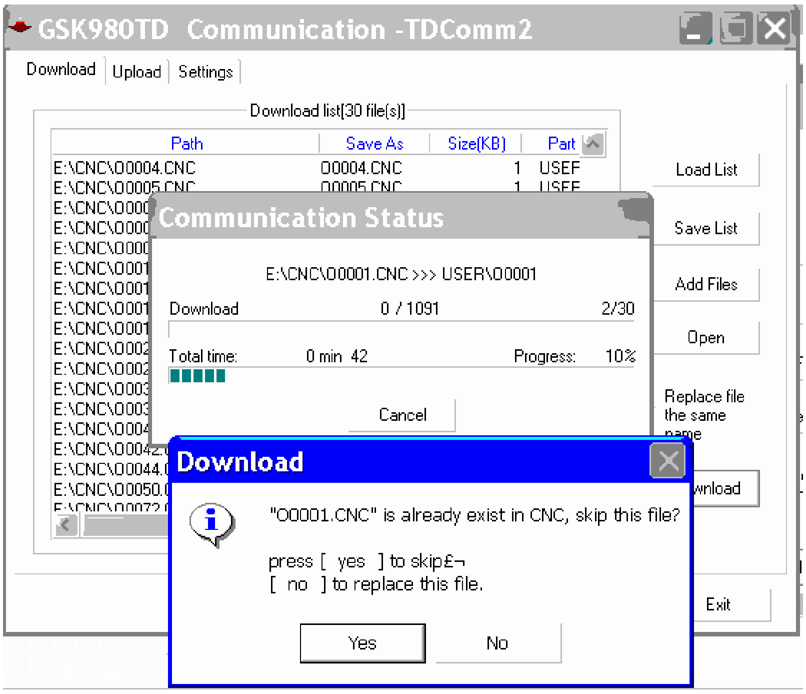


Fig. 3-14

11.1.2 File Uploading （CNC→PC）

Refresh Directory: In " File Upload" mode, show the file directories in all areas in CNC.

Delete File: In " File Upload" mode, delete the selected files in the file list from CNC.

Rename File: In " File Upload" mode, rename the files in user storage area in CNC.

1. Operation on PC

Click " File Upload" and select the following interface, click " Refresh Directory" Button, the CNC file directories show in the file list box in the main interface. Click the pane on the left of the directory item, the to-transfer files are selected, the red tick stands for selected.

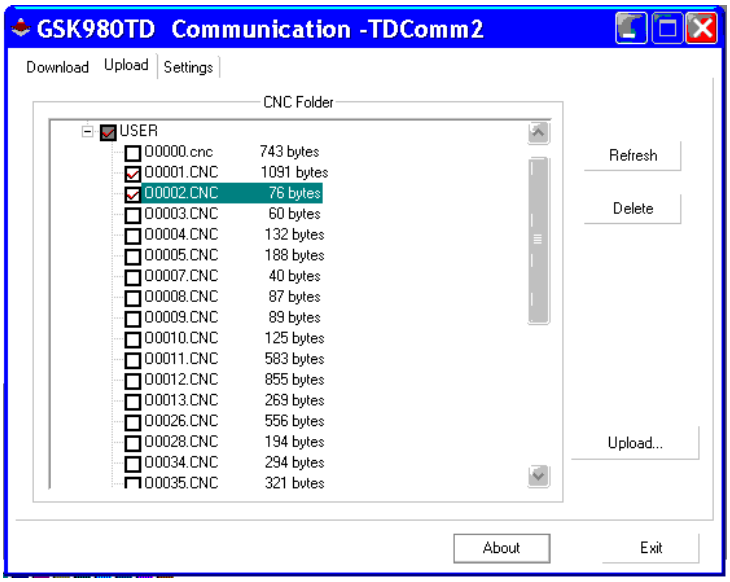


图 3-15

Click " Save to Directory" botton after selected, select the directory to store and start to receive the files transferred from CNC. The communication statue box as Fig. 3-13 will shows during the transferring process, and disappears after the transferring.

2. Operation on CNC

After connected to the CNC, in idle staute, the software can receive files sent initiatively by CNC. When the CNC starts to transfer files, the program starts to receive datas immediately, and ask the user to save the files after the receiving is finished.

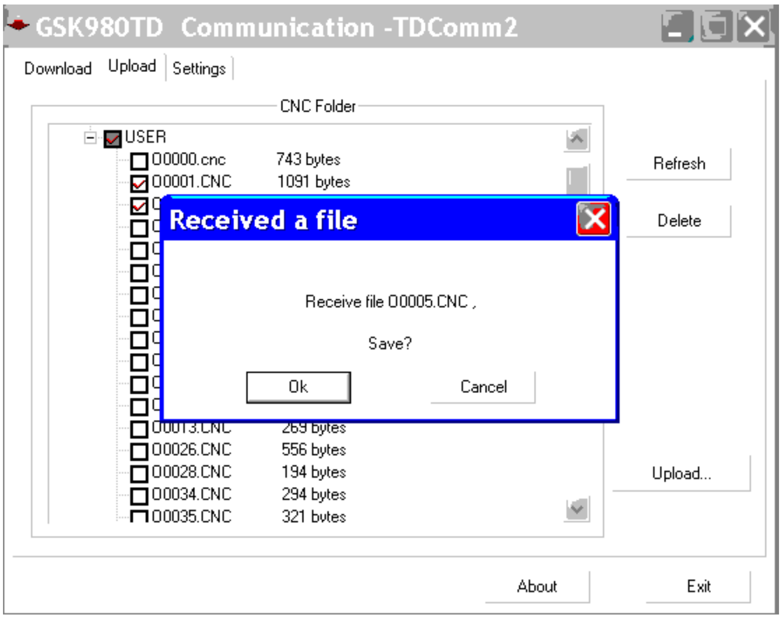


Fig. 3-16

3. Delete Files on CNC

When selecting " File upload" perperty page in Dialog box, after selecting the to-delete files in the file list, press the button " Delete File" to delete the selected files. (select more than one file to delete at a time is OK too).

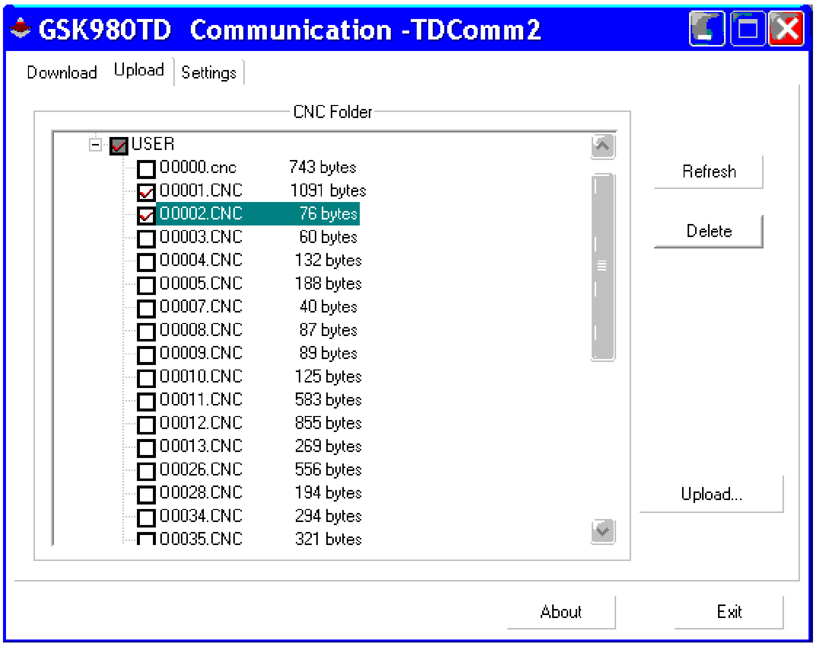
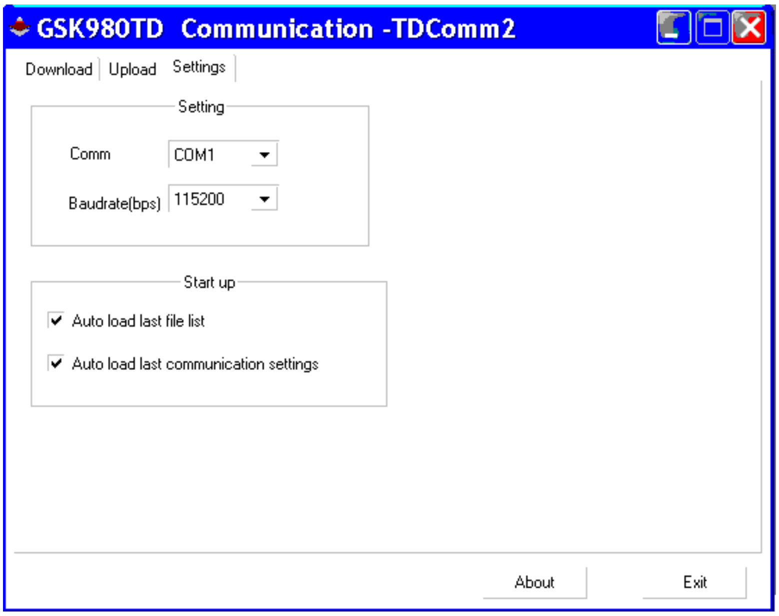


Fig. 3-17

11.1.3 Setting Option



1. **Communication setting box**

   Port select: choose PC COM port as the the communication port, COM1～COM4 for option.

   Baud rate: choose the baud rate of the communication, 4800～115200 for option.

2. **The auto load selection box when the program starts**

The previous file sending list: When the progrom starts again, load or not for the file list which have been loaded as the file load interface (Fig. 3-1) by last time.

The previous communication setting: When the progrom starts again, load or not for the communication settings as previous.

Preparations before communication

1. Connect the PC and CNC with the communication cable when both powers are off.

Connections between PC and CNC: Insert the DB9 pin connector plugs into the XS36 communication jack of CNC, insert the bore connectore plugs into serial port with 9 pins (COM orCOM1) of PC;

Connections between CNC and CNC: Insert the DB9 pin connector plugs into XS36 communication jacks of both CNC.

2. Set the Bit5（RS232）of CNC status parameter as 1 (details please see appendix one).

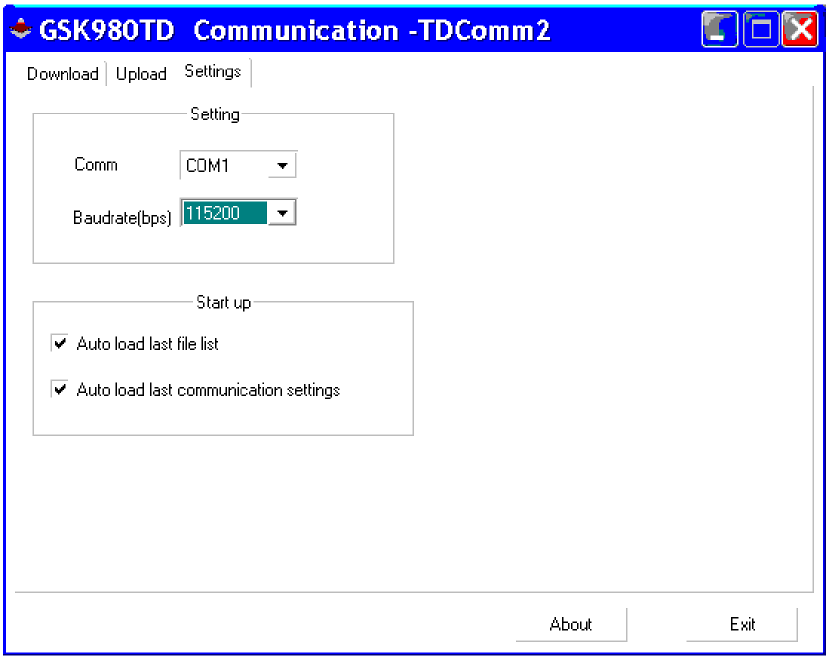
3. Set the baund rates of communication, to realize the baund rates are consistent between PC and CNC,or CNC and CNC.

● **Baund rate setting of CNC:**

The communication baund rate of CNC serial port for GSK980TD is set by No.044 data parameter, range from 50~115200 (unit: bps), the set value should not under 4800 during transmission between CNC and PC. And the default is 115200.

● **Baund rate setting in PC:**

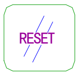
Run the communication software, the interface will be displayed as below by selecting the “option setting” and clicking the left key of mouse.



Port selecting: Select the port（COM1,COM2,COM3,COM4）for communication.

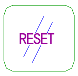
Baund rate selecting: Select the communication baund rate (4800,9600,19200,38400,57600,115200（unit：bps））

Note1: Turn on the program switch if program transmission is needed; Turn on the parameter switch if parameter and tool offset,etc, transmission is needed; If alarm occurs when turning on the switch,the alarm can be cleared by pressing the 

and 

key simultaneously.

Note2: To ensure the steady and reliable communication, please stop processing if it is running. If data sending from CNC is needed, please switch to the edit operation first.

Note3: It can be stopped by pressing the 

key;

Note4: Do not shut off the power during the data transmission, or the data transmission error will be occurred.

11.3 DATA INPUT（PC→CNC）

The appointed data file can be transferred to CNC by performing the input function, such as program, parameter,tool offset and worm offset,etc.

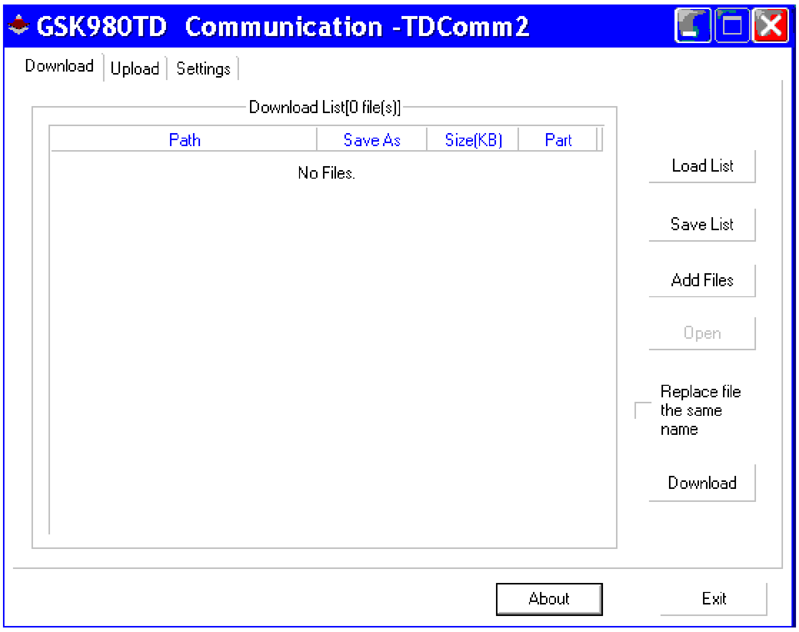
11.3.1 Program Input

1. Set the operation level (level 2 is required for macro setting) and then turn on the program switch;

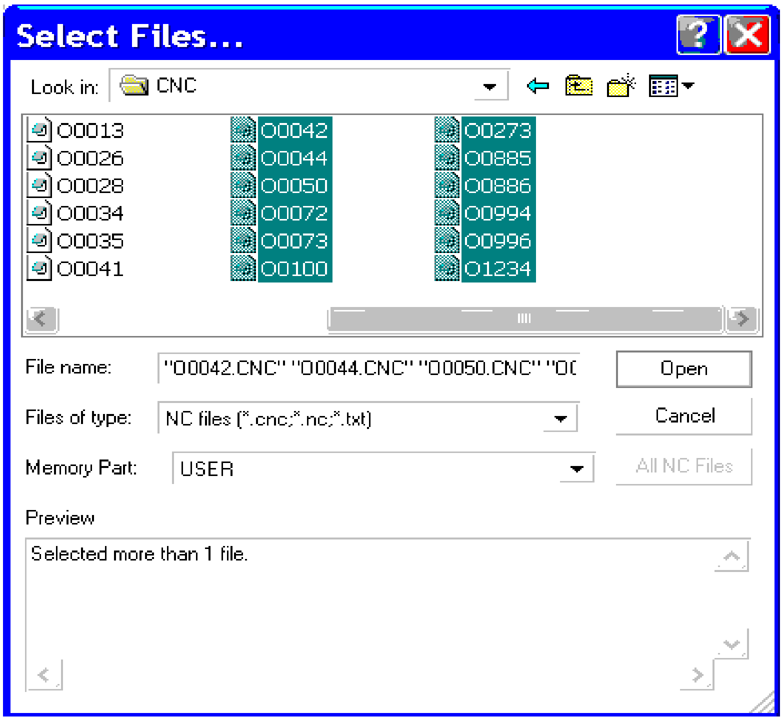
2. Edit the programs in PC (files with *.cnc,*.nc,*.txt postfix are supported), and then save them in the harddisk;

3. Click the option of “file download” when the communication software is running in PC, it will be displayed as below:

Book2 operation

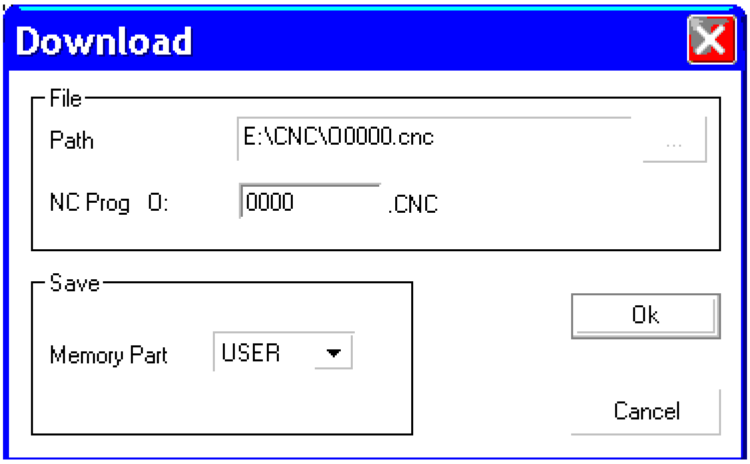


4. Click the "Add file" button under the software interface as shown in above chart, A dialogue box for adding file will appears, the screen will be shown as below after selecting the edited program.

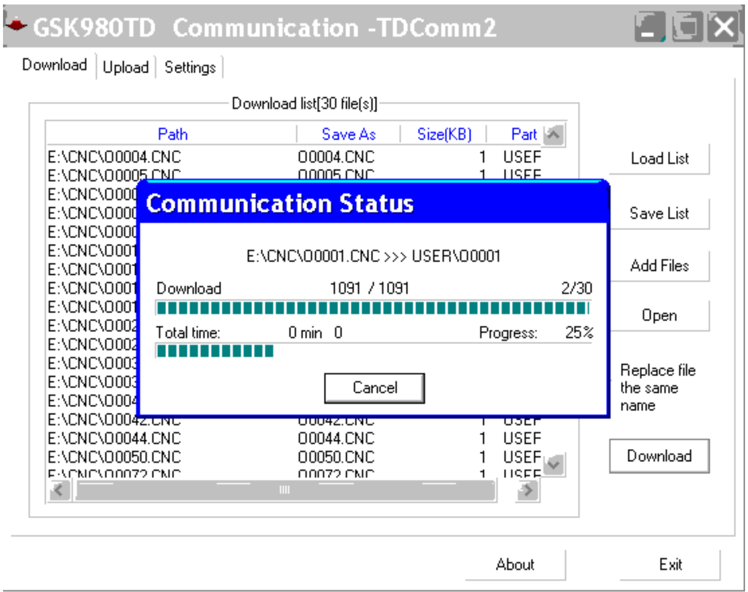


5. Under the software interface as shown in above chart, the program name can be changed by double clicking it:

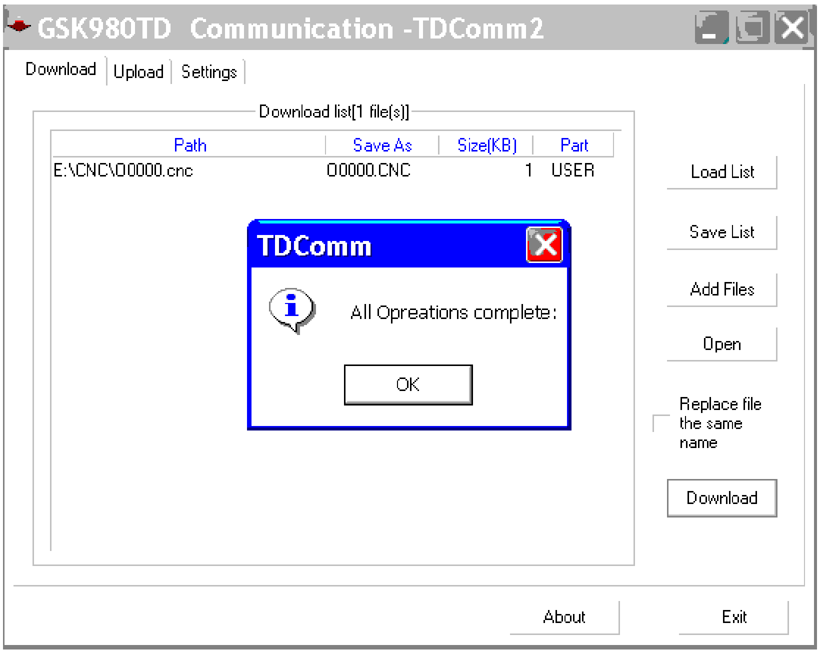
Book2 operation



6. It will be displayed as below by clicking the “start to download” button:



7. After the transmission, it will be displayed as below by clicking the “confirm” button on the arisen box.



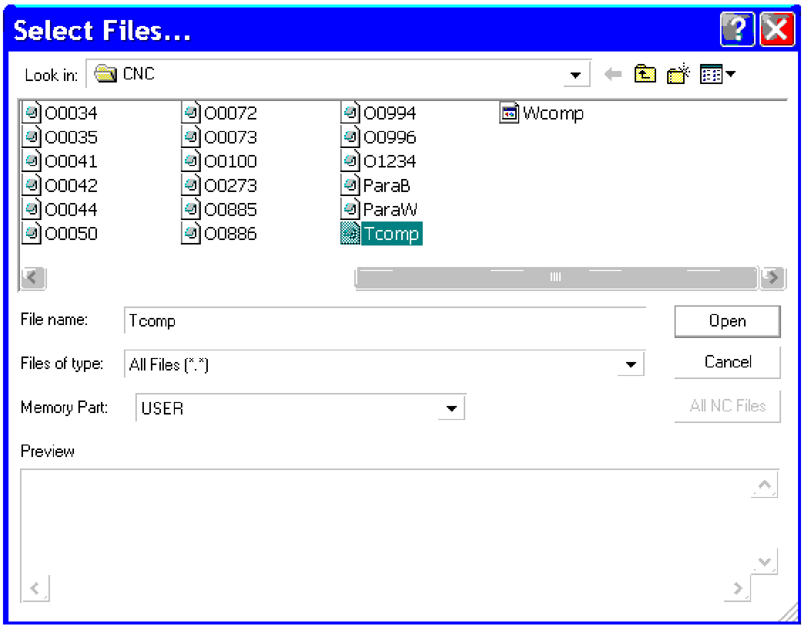
8. Single program or more can be transferred according to step 1 to step 7.

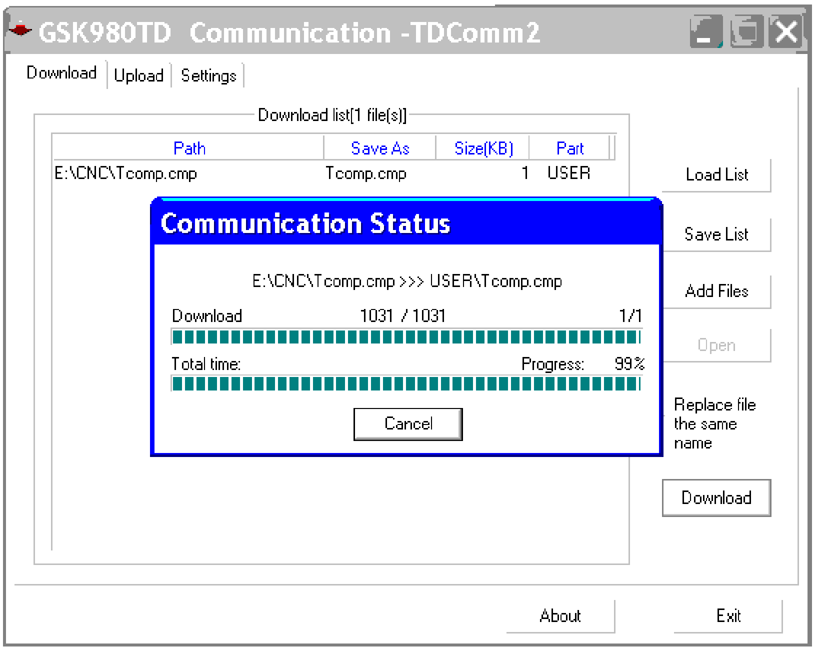
Note: The operation at CNC side requires level 3 or above.

11.3.2 Tool Offset Input

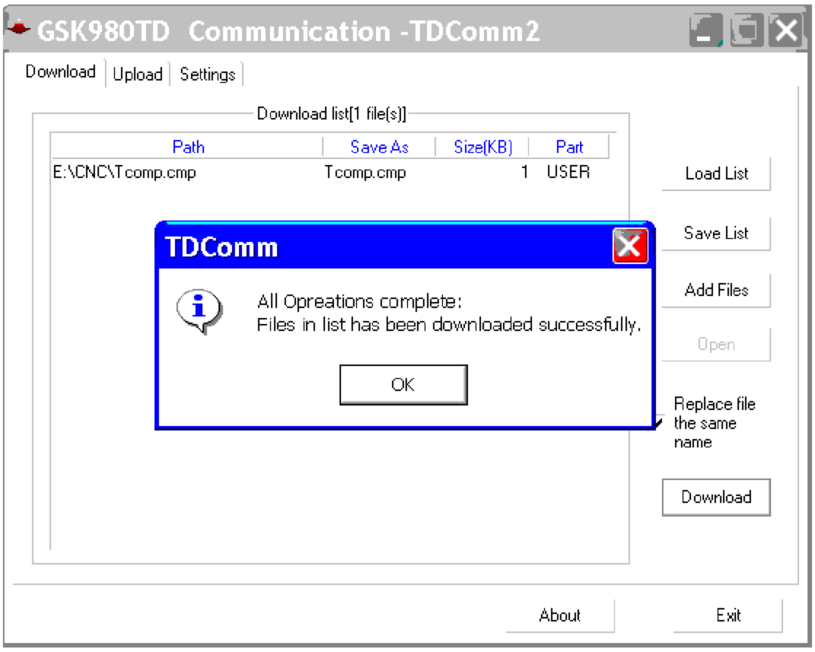
1. Set a corresponding level in CNC, turn on the parameter switch and then select the edit operation;

2. Run the communication software in PC, select the “file download” option and then click the “add file” button to add the tool offset file which is to be transferred (the file postfix should be .cmp, such tool offset file can be transferred from CNC if there isn’t), it will be displayed as below by clicking the “start to download”:





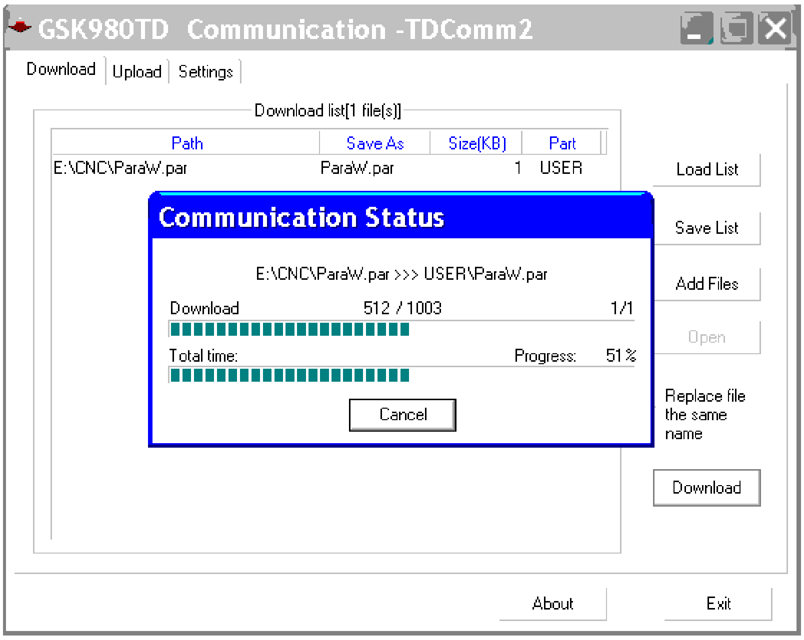
3. Other operations are avalilabe by clicking the “confirm” after the transmission.



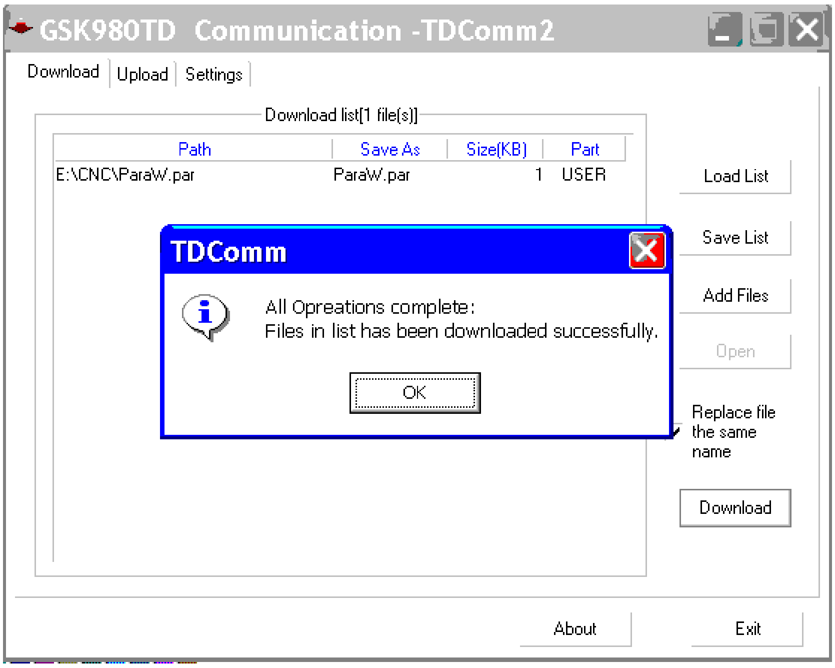
11.3.3 Parameter Input

1. Set a corresponding level in CNC (level 2 is required for worm offset setting), turn on the parameter switch and then select the edit operation;

2. Run the communication software in PC, select the “file download” option and then click the “add file” button to add the parameter file which is to be transferred (the file postfix should be .par, such tool offset file can be transferred from CNC if there isn’t), it will be displayed as below by clicking the “start to download”:



4. Other operations are avalilabe by clicking the “confirm” after the transmission.



Note1: Parameter files including status parameter,data parameter and worm offset parameter, users can select the operation based on the requirements.

Note2: The displaying sequence numbers in PC are counted from 0 for the status and data parameter, corresponding to the parameter sequence in CNC.

Note3: User want to transfer the status and data parameter from PC requires CNC operation level 3 or above.

Note4: Transferring the worm offset parameter requires CNC operation level 2 or above

11.4 DATA OUTPUT（CNC→PC）

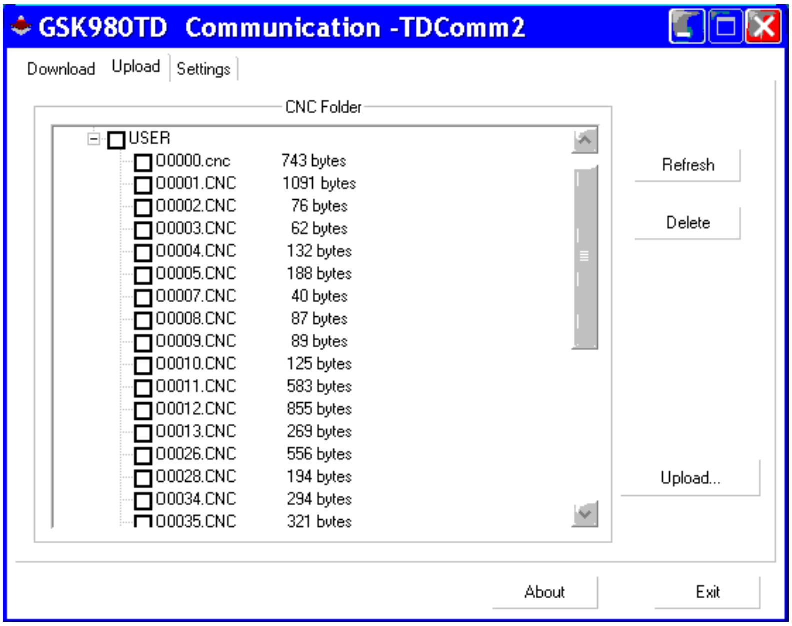
The CNC datas of GSK980TD can be transferred to PC by performing the output function, the datas can be received by PC including program, parameter, tool offset or worm offset, etc.

11.4.1 Single Program Output

Detail operations of transferring single program to PC are as follows:

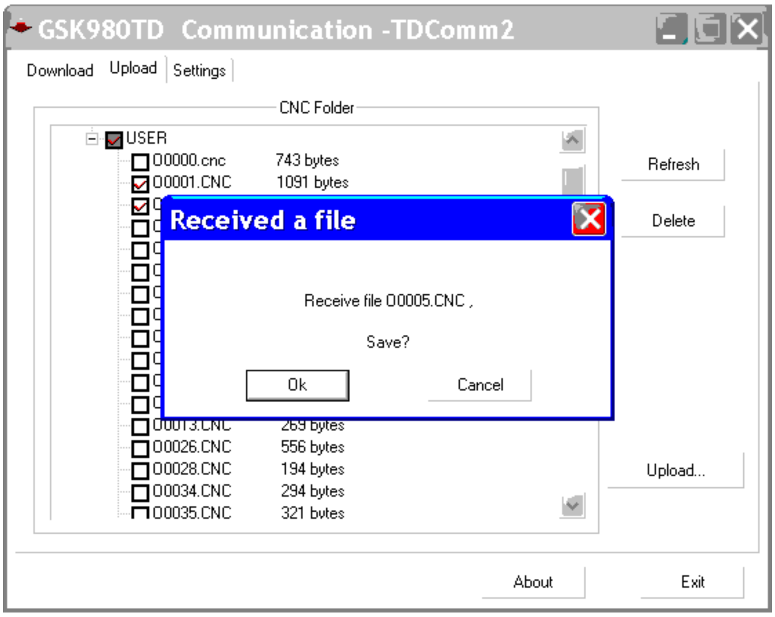
Method 1: Operation at CNC side

1. Select the edit operation and enter the program contents page;
2. Run the communication software at PC side, switch to 【file upload】page;

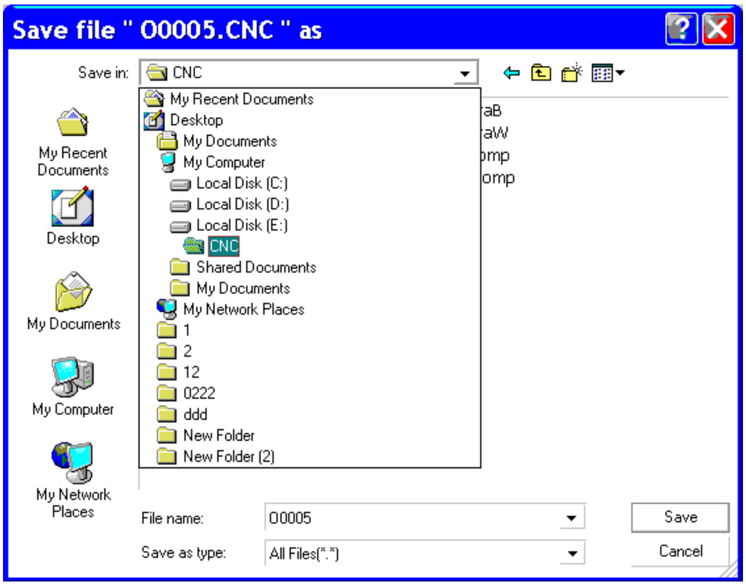


3. Press the O key and input the program name at CNC side (this step can be omitted if transferring current program);

4. Output starts by pressing the OUTPUT key, The character "output" under the CNC screen will be glitter, and the screen at PC will be displayed as follows after the transmission:

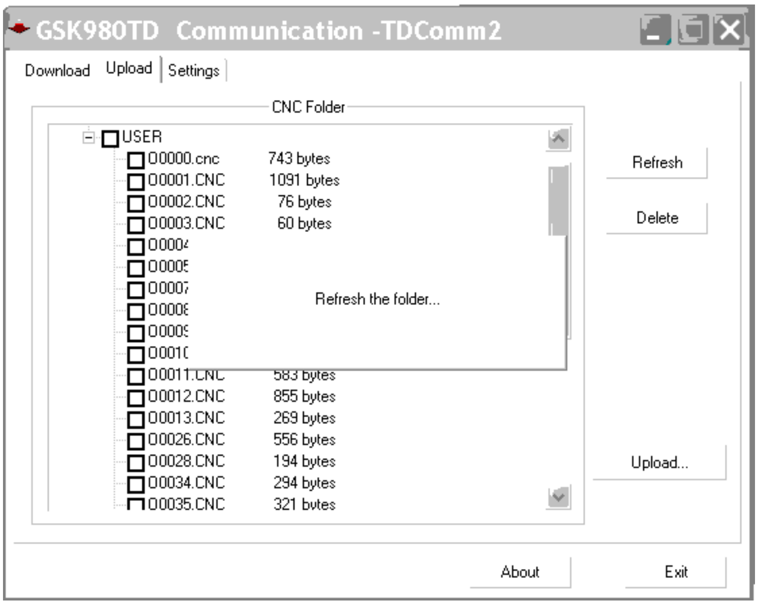


5. Click the【no save】button if no need to save; if need to save it, please click the【save】button and then select the saving route on the save route dialogue box, the file will be saved by click the 【save】 button:

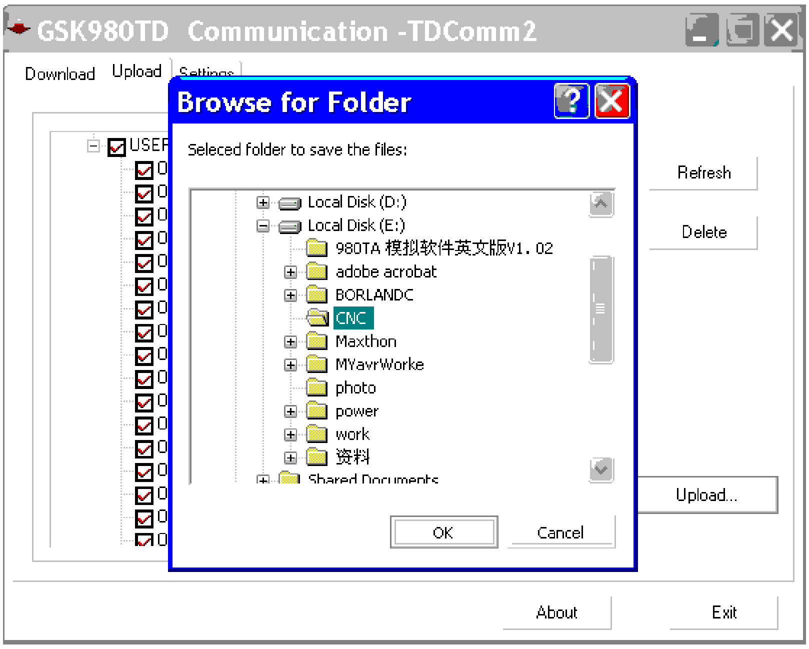


Method 2: Operations at PC side:

1. Select the edit operation and enter the program contents page;
2. Run the communication software at PC side, switch to【file upload】page and click the【contents refurbish】;



3. Select the program to be save, and then click the 【save to folder】, details are as follows (select the tenth program to save):



4. Select the saving route and then click the "confirm".

11.4.2 All Program Output

All programs in CNC can be outputted to PC by user, operation steps are as follows:

1. Enter the program contents page under the edit operation mode;

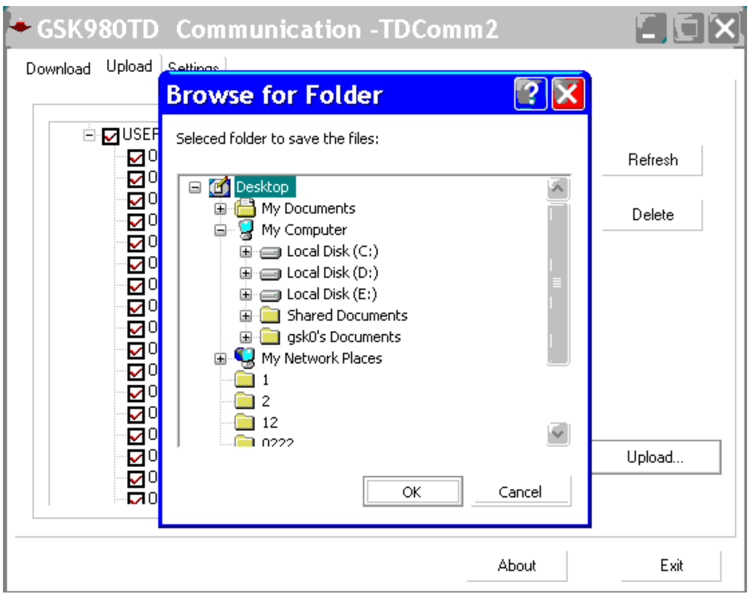
2,. Process the communication software in PC, switch to【Document upload】page;

3. Input the address key O ,symbol key – and other address keys 9 , 9 , 9 ;

4. Press the OUTPUT key, the output begins, the character "Output" under CNC screen will be displayed and glitter, when the output is done, the screen of PC will be displayed as follows:

5. Save the programs one by one according to Section 11.5.2.

Note: It also can be done in PC according to "method 2" described in section 11.5.1, select all programs, and save them at the desired route. The page will be displayed as below:



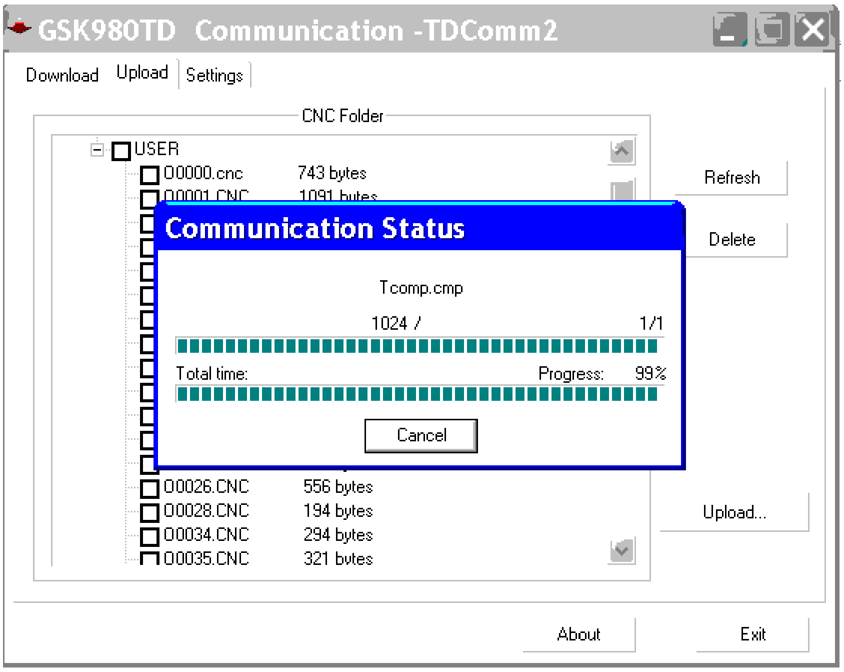
11.4.3 Tool Offset Output

1. Enter the tool offset page under the edit operation mode;

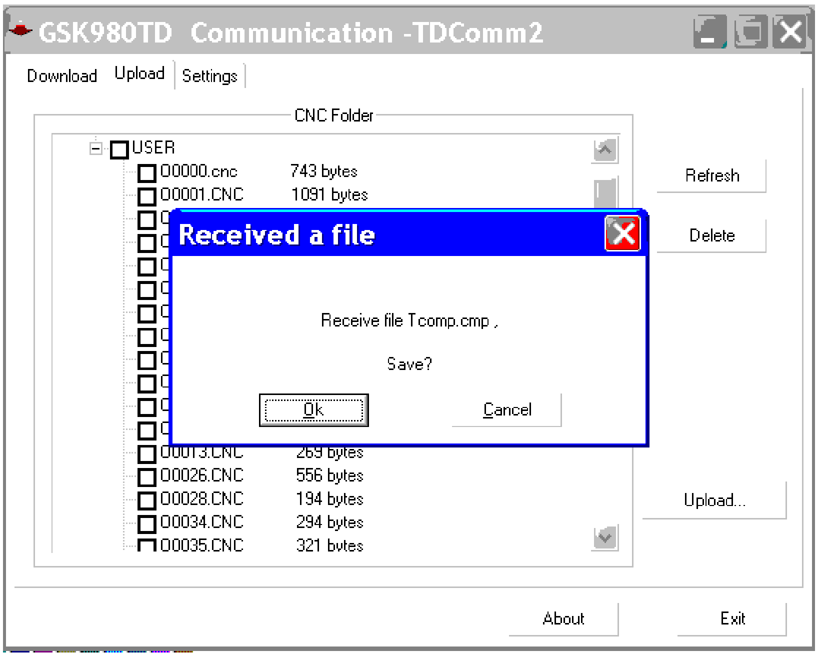
2. Process the communication software in PC, switch to【Document upload】page;

3. Press the 

key on CNC, the output begins, the character “Output” under CNC screen will be displayed, the screen of PC will be displayed as follows:



4. When the output is done, the screen of PC will be displayed as follows (default name: Tcomp.cmp):



5. Save the tool offset in desired route according to Section 11.5.2.

11.4.4 Parameter Output

1. Enter the related page of parameter interface under edit operation mode:

2. Enter the status parameter page if status parameter has to be transferred

3. Enter the data parameter page if data parameter has to be transferred

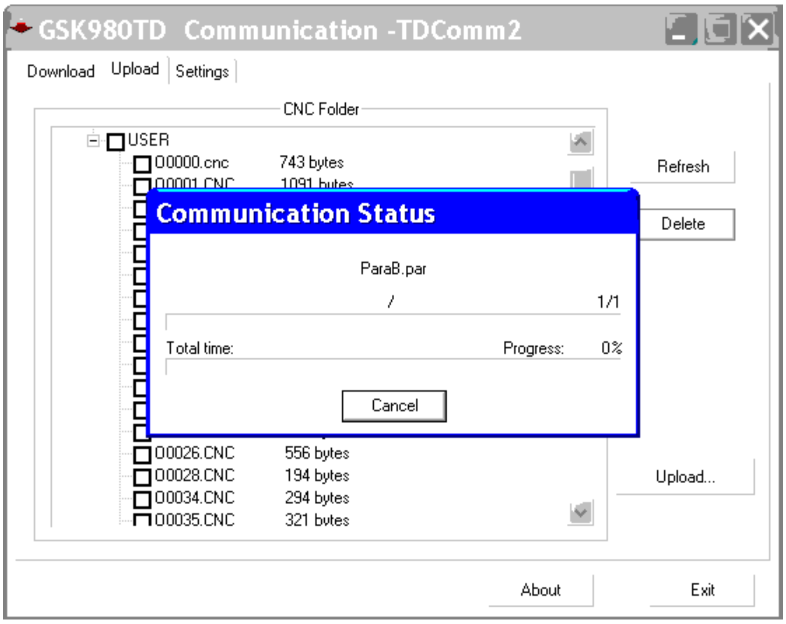
4. Enter the worm offset data page if worm offset data has to be transferred;

5. Process the communication software in PC, switch to【Document upload】page;

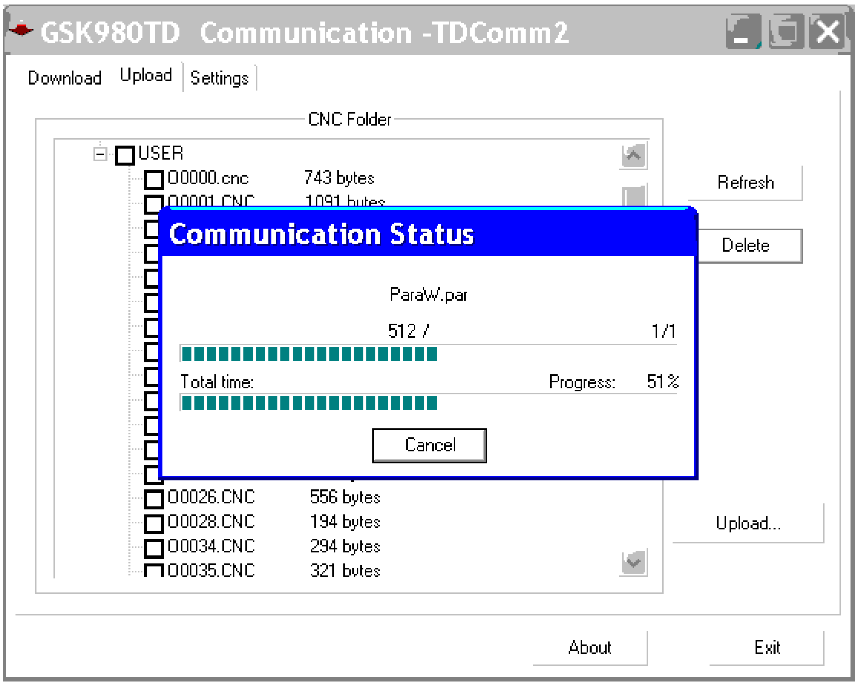
6. Press the OUTPUT key on CNC, the output begins, the character "Output" under CNC screen will be displayed, the screen of PC will be displayed as follows:

7. During the transmission, the file names dispayed in PC are different for different types of datas, detaisl are as follows (The default names are marked in ellipses):

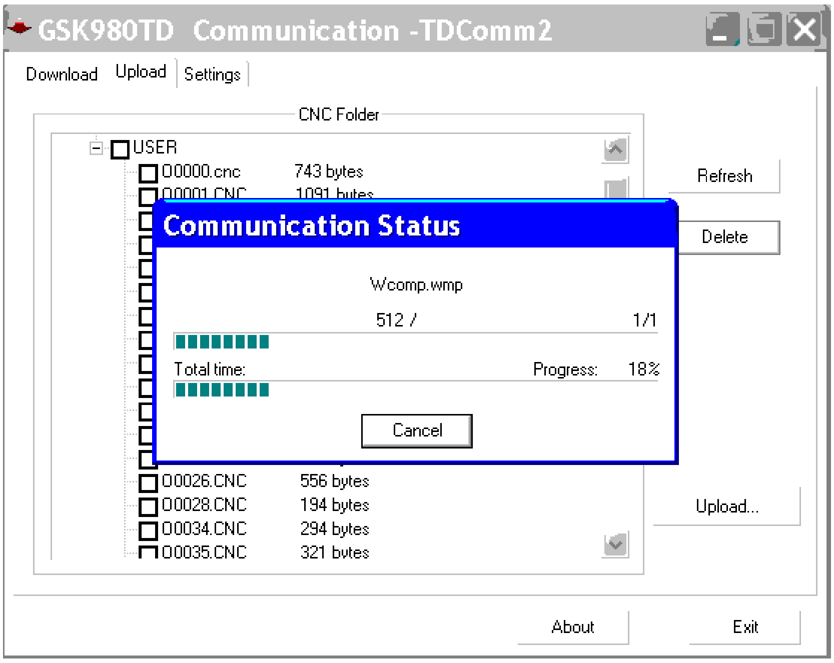
Status parameter, displaying as follows (default file name: ParaB.par)



Data parameter, displaying as follows (default file name: ParaW.par)



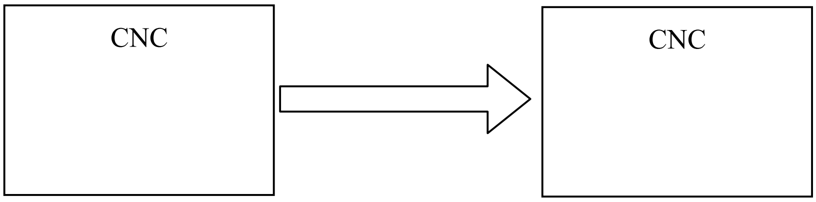
Worm offset data, displaying as follows (default file name: Wcomp.wmp)



5, Save the parameter datas to the desired rote according to section 11.5.2.

11.5 COMMUNICATION betweent CNC and CNC

To simplify the operation, transmission between two CNCs is allowed, the sending CNC is so-called sender; and the receiving CNC is so-called receiver.Detaisl are as follows:



Sender Receiver

Please take note of below when transferring datas:

1. The communication functions are both available for sender and receiver, namely, the Bit5（RS232）of No.002 of CNC status parameters are both set as 1;
2. The baud rates are the same between two sides, namely the No.044 CNC data parameters are the same;
3. Both side must under the edit operation;
4. The sender must under the page which is ready for data transferring (For example, it must under status parameter page if status parameter is to be transferred)
5. The receiver must under the related levels and all related switches must have been turned on (parameter switch, program switch)

Received datas	Level	Remark
Program (Program names under 9000)	Level 4,3,2	Turn on the program switch
Macro program (Program above or equal to 9000)	Level 2	Turn on the program switch
Tool offset value	Level 4,3,2	

Status parameter	Level 3,2	Turn on the parameter switch
Data parameter	Level 3,2	Turn on the parameter switch
Worm offset data	Level 2	Turn on the parameter switch

6. The operation steps are the same with “Data output (CNC→PC)” on CNC which is described in Section 11.5.

Chapter 12 PROCESS EXAMPLE

To fabricate a workpiece as below , the dimension of roughcast is Φ136×180mm.

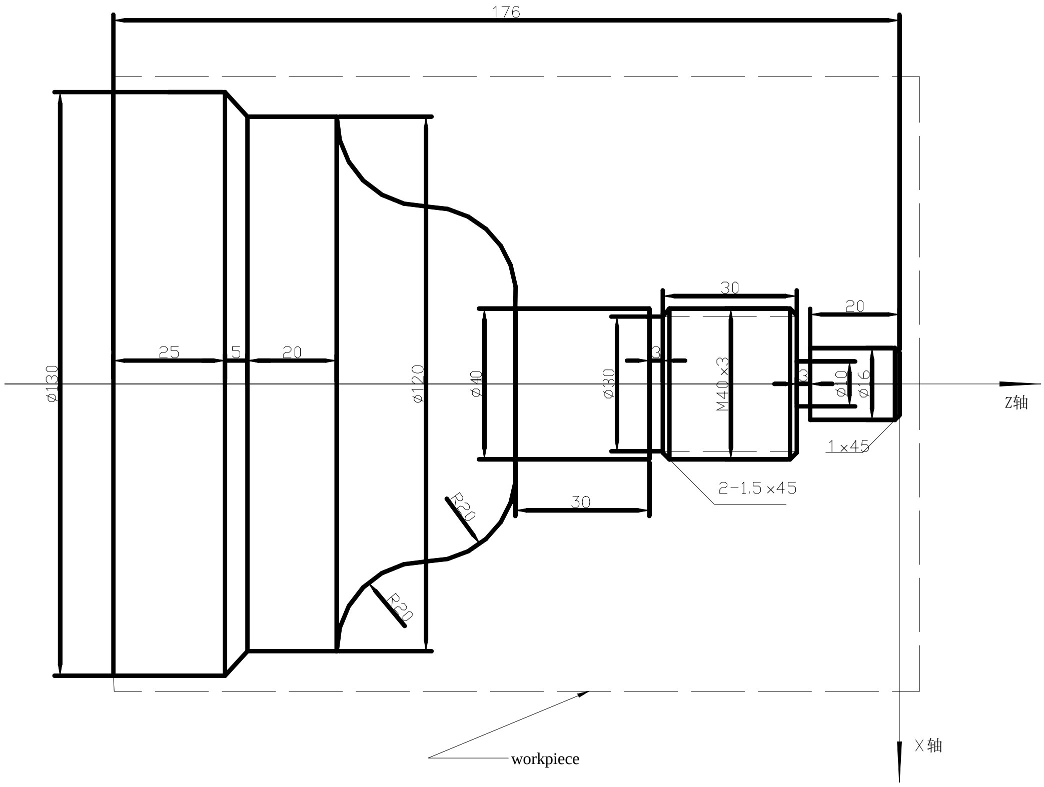
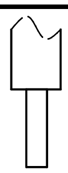


Chart 12-1

To fabricate with below four tools:

Number	Tool shape	Description
Tool 1		Outer rough tool
Tool 2		Outer smooth tool
Tool 3		Slot cutting tool (width:3mm)
Tool 4		Worm cutting tool (angle of knifepoint: 60°)

12.1 PROGRAM EDIT

According to the machining technics and the operation manual, set up a workpiece coordinate as chart 12-1, the program are as follows:

O0001;		Program name
N0000	G0 X150 Z50;	Move to a safe position for tool change
N0005	M12;	Clamp the chuck
N0010	M3 S800;	Turn on the motor with 800 r/min
N0020	M8;	Coolant on
N0030	T0101;	Change the first toll
N0040	G0 X136 Z2;	Move the tool close to the workpiece
N0050	G71 U0.5 R0.5 F200;	Cut the workpiece with 1 mm deep and drow it back with 1 mm
N0055	G71 P0060 Q0150 U0.25 W0.5;	Reserve 0.5 mm for both X and Z axises
N0060	G0 X16;	Get close to the section of workpiece
N0070	G1 Z-23;	Cut for Φ16
N0080	X39.98;	Cut for the section
N0090	W-33;	Cut forΦ39.98
N0100	X40;	Cut for the section
N0105	W-30;	Cut forΦ40
N0110	G3 X80 W-20 R20;	Cut for a gibbosity
N0120	G2 X120 W-20 R20;	Cut for a concave
N0130	G1 W-20;	Cut for aΦ120
N0140	G1 X130 W-5;	Cut for a taper
N0150	G1 W-25;	Cut for aΦ130
N0160	G0 X150 Z185;	Return the tool change position
N0170	T0202;	Change to tool 2 and run with No.2 tool offset
N0180	G70 P0060 Q0150;	Smooth cutting cycle
N0190	G0 X150 Z185;	Return the tool change position
N0200	T0303;	Change to tool 3 and run with No.3 tool offset
N0210	G0 Z-56 X42;	Get close to the workpiece
N0220	G1 X30 F100;	Cut for aΦ30 taper
N0230	G1 X37 F300;	Return
N0240	G1 X40 W1.5;	Cut for an angle
N0250	G0 X42 W30;	Cut the slot for drawing back
N0260	G1 X40 ;	
N0262	G1 X37 W1.5;	Cut for an angle
N0264	G1 X10;	Cut for aΦ10 slot
N0266	G0 X17 Z-1;	
N0268	G1 X16;	
N0270	G1 X14 Z0 F200;	Cut for an angle
N0280	G0 X150 Z50;	Return to the tool change position
N0290	T0404 S100;	Change to No.4 tool and change the speed to 200 r/min
N0300	G0 X42 Z-54;	Get close to the workpiece
N0310	G92 X39 W-34 F3;	Worm cutting cycle
N0320	X38;	Second cut with 1mm deep
N0330	X36.4;	Third cutting with 0.6 mm deep
N0332	X36;	Fourth cutting with 0.4 mm deep
N0340	G0 X150 Z50;	Return to the tool change position

N0350	T0100;	Change back to No. 1 tool
N0360	M5;	Turn off the spindle
N0370	M9;	Coolant off
N0380	M13;	Release the chuck
N0390	M30;	Program over

12.2 PROGRAM INPUT

12.2.1 Look over the Preserve Programs

Under non-edit operation, enter the program interface by pressing the PROGRAM key, select the program contents page by pressing the  or  key, details are as follows:

```
PROGRAM LIST                               O0009 N0000
   Version: GSK-980TD                        V05.10.20
   Part-prog No.: Most 384;          Used:          20
   Memory capacity: 6144 KB;         Used:     5310 KB
   Program list:
    O0000  O0002  O0003  O0004  O0005  O0006
    O0007  O0008  O0009  O0010  O0011  O0012
    O0014  O0023  O0088  O0089  O1000  O0044
    O0100  O0101

Program capacity: 16KB     Note: QIU TOU GAN
                                  S 0000 T0100
                               MDI
```

The preserve program names can be looked over on above page, then it is ready to set up a new program name.

12.2.2 Set up a New Program

Under the edit operation, enter the program contents page by pressing the PROGRAM key, the page will be displayed as follows:

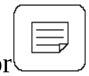
```
PRG.CONTENT LINE2 COLUMN1 O0101 N0000
O0101; (CNC PRPGRAM.20051125)
G50 X100 Z100;
G0 X0 Z2;
G01 W-100 F200;
G0 X100 Z100;
M30;
%





                          S 0000 T0100
                     EDIT
```

Press the address key O, select a nonexistent program (such as 0001), input the numeric keys 0, 0, 0, 1, set up a new program by pressing the EOB key, the page will be displayed as follows:

```
PRG.CONTENT LINE2 COLUMN1  O0001  N0000
O0001; (O0001)
;_
%

                                S 0000 T0100
                          EDIT
```

Input the program as above mentioned, the program edit will be done and after that the first page will be displayed as below:

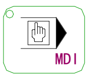
```
PRG.CONTENT LINE2 COLUMN1  O0001  N0000
O0001; (O0001)
N0000 G0 X150 Z185;
N0005 M12;
N0010 M03 S300;
N0015 M08;
N0020 T0101;
N0025 G0 X136 Z180;
N0030 G72 R1 U2 F200;
N0035 G71 P0040 Q0180 U1 W1;
%
                                S 0000 T0100
                          EDIT
```

To display other contents by pressing the or key.

12.3 PROGRAM TEST

12.3.1 Graph Setting

Enter the graph interface by pressing the 

key, select the graph setting page by pressing the or key, then enter the MDI operation by pressing the 

key, the page will be displayed as below:

```
GRAPH SETTING                              O0008
N0000
                  Graph parameter
■Coordinates=          1          (XZ:0    ZX:1)
 Scale       =     50.000
 Center      =    120.000         (X axis value)
 Center      =    160.000         (Z axis value)
 Max. X      =    240.000
 Max. Z      =    320.000
 Min. X      =      0.000
 Min. Z      =      0.000
 (Tool path display area:  320*240)

No. 001                           S 0000 T0100
                          MDI
```

Book2 operation

Move the cursor by pressing the or key to set the “coordinate selecting”, “max. value of X”, “max. value of Z”, “min. value of X” and “min. value of Z” (the “scaling” and “graph center” in the graph setting will be adjusted automatically according to the “max. value of X”, “max. value of Z”, “min. value of X” and “min. value of Z” .) Here is the setting of “max. value of X” will be set as an example of graph parameter setting :

1. Move the cursor to the parameter of“max. value of X” by pressing the or key.
2. The dimension of the roughcast is 135 mm, then the input value should be bigger than 136 mm, here 150 mm is set, input 1 , 5 , 0 in sequence;

3, The page will be displayed after setting and pressing the INPUT key:

```
GRAPH SETTING                      O0008 N0000
                 Graph parameter
   Coordinates=         1        (XZ:0    ZX:1)
   Scale      =      0.000
   Center     =      0.000       (X axis value)
   Center     =      0.000       (Z axis value)
 ■X max.      =    150.000
   Z max.     =      0.000
   X min.     =      0.000
   Z min.     =      0.000
   (Tool path display area: 320*240)

No. 005                          S 0000 T0100
                            MDI
```

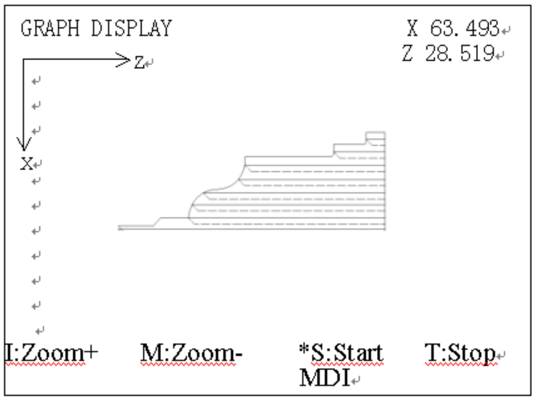
To set other datas according above method, in this example the page will be displayed as below after setting:

```
GRAPH SETTING                      O0008 N0000
                 Graph parameter
 ■Coordinates=          1        (XZ:0    ZX:1)
   Scale      =     32.812
   Center     =     75.000       (X axis value)
   Center     =     95.000       (Z axis value)
   X max.     =    150.000
   Z max.     =    200.000
   X min.     =    -10.000
   Z min.     =    -10.000
   (Tool path display area: 320*240)

No. 001                          S 0000 T0100
                            MDI
```

12.3.2 Program Test

Enter the graph displaying page by pressing the or key, enter the automatic operation by pressing the AUTO key, Enter the auxiliary lock, machine lock and dry running by pressing the MST M. S. T. LOCK , MACHINE LOCK , DRY keys, at the moment the LEDs of MST , and are turned on.Start to plot by pressing the S key, and then press the RUN key to run program automatically,the tool moving track will be displayed and the correctness of program can be tested,the page will be displayed as follows after all set.

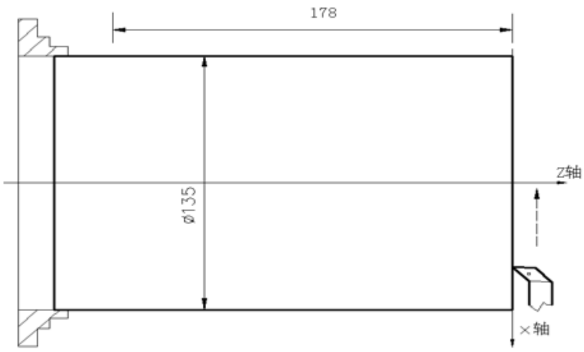
Book2 operation



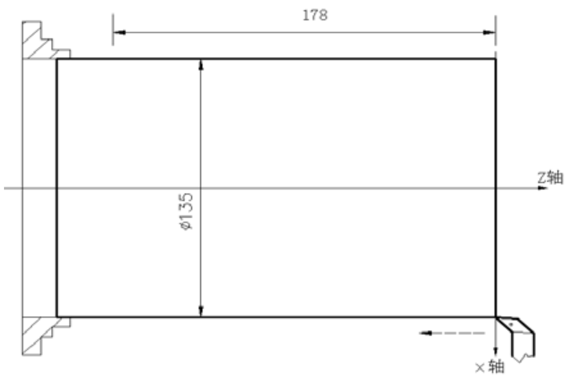
If there is something wrong, please analyse the error and modify the program, then test it again until there is nothing unusual.

12.4 TOOL ADJUSTING and RUN

1. Move the tool to a safe positon, run the T0100 U0 W0 under the MDI mode, program status page, and then cancel the tool offset;
2. Move the tool close to the workpiece and let it cut along the section.



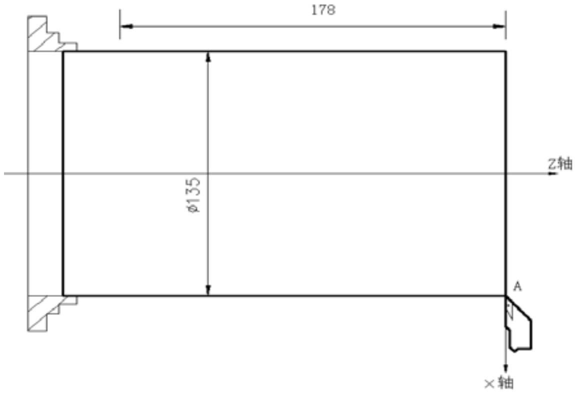
3. Release the tool along the X axis and remain the Z axis, stop the spindle, perform the G50 Z0 under the program status page of MDI operation, then set the coordinate of Z axis;
4. Switch to tool offset page and input Z0 into No.001 offset;
5. Move the tool close to the workpiece and cut along the outer surface;



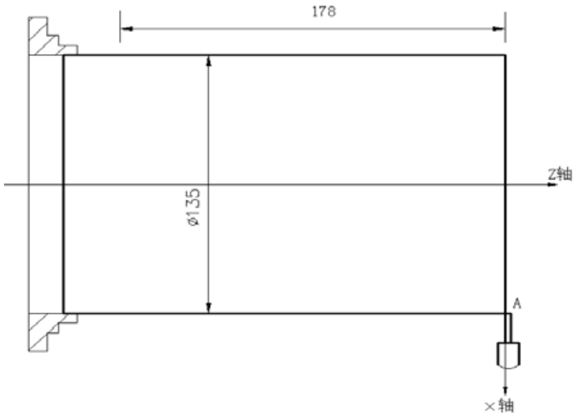
6. Release the tool along the Z axis and remain the X axis, stop the spindle, then measure the dimension of the outer column (suppose the value is 135 mm)
7. Perform the G50 X135 under the program status page of MDI operation, then set the coordinate of X axis
8. Switch to tool offset page and input X135 into No.001 offset;
9. Move the tool to a safe positon,select the No.2 tool by pressing the tool change key under the manual

operation;

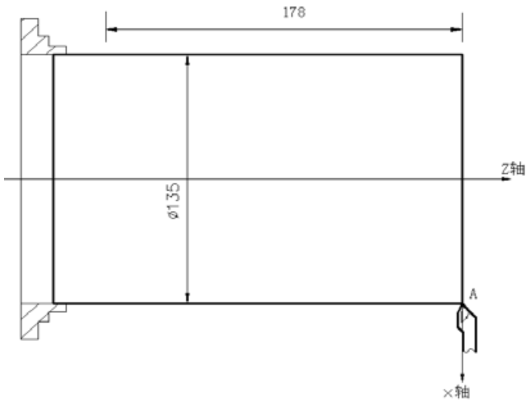
10. Start the spindle, move the tool to the fixed point, A point as follows;

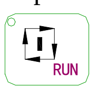


11. Switch to tool offset page, move the cursor to No.2 offset and then input X135,Z0；
12. Move the tool to a safe positon, select the No.3 tool by pressing the tool change key under the manual operation;
13. Start the spindle, move the tool to the fixed point, A point as follows;



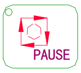
14. Switch to tool offset page, move the cursor to No.3 offset and then input X135,Z0；
15. Move the tool to a safe positon, select the No.4 tool by pressing the tool change key under the manual operation;
16. Move the tool to the fixed point, A point as follows;

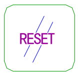


17. Switch to tool offset page, move the cursor to No.4 offset and then input X135, Z0；
18. Tool adjusting is done, move the tool to a safe position;
19. Start the automatic process by pressing the 

key under the automatic operation;
20. If there is any error between the designed and actual dimension, the tool offset can be modified until the dimension under the tolerance.

Book2 operation

Note: Press the 

to pause the automatic running if needed. If any emergency occurs, the 

key, emergency button can be pressed or the power can be shut off to terminate the program running.

BOOK 3

CONNECTION

Chapter 1 FIXING and LAYOUT

1.1 CONNECTION of GSK980TD

1.1.1 Interface Layout of GSK980TD

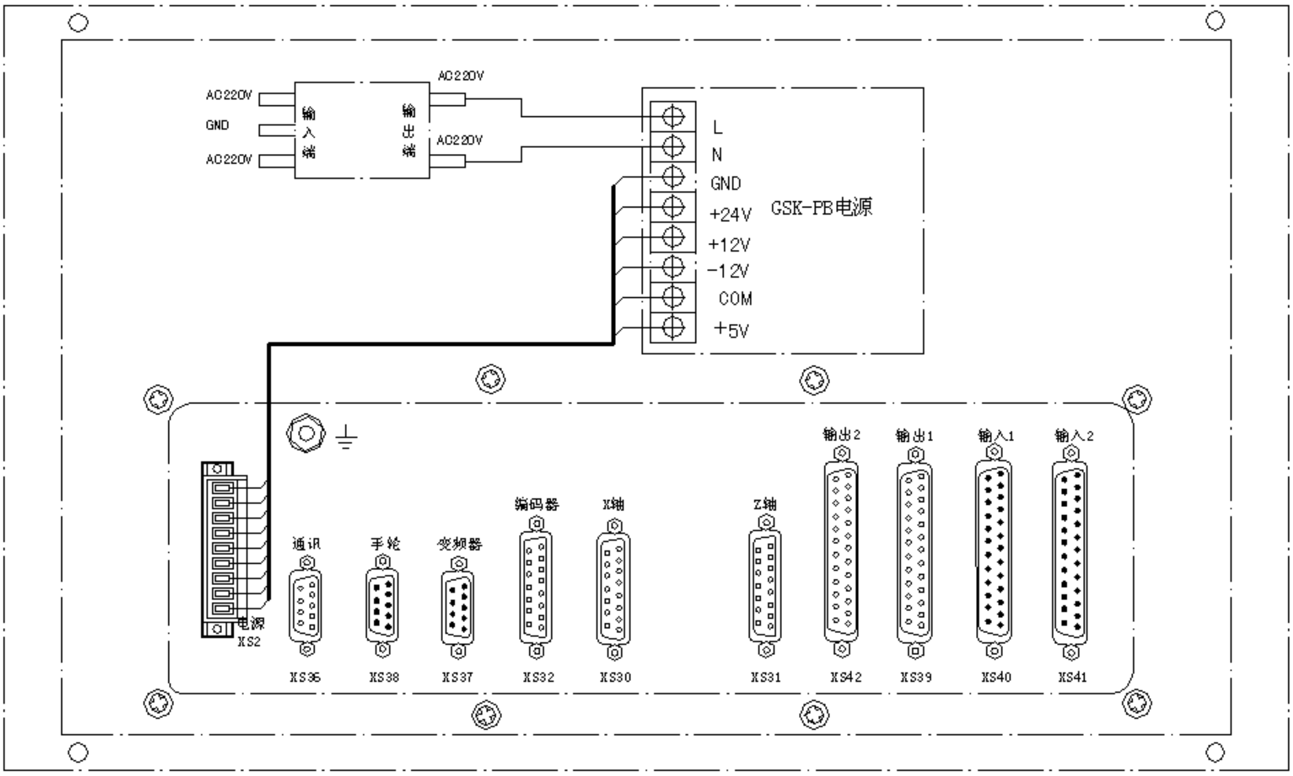


Fig.1-1 interface layout of GSK980TD

NOTE: XS41 and XS42 are optional interfaces

1.1.2 Interface Description

- Power box：Adopt GSK-PB power box and provide the voltage of +5V、+24V、+12V、-12V、GND
- Filter（optional）：Input port connects to 220VAC power，PE port grounds and output port connects to L，N port of GSK-PB power box
- XS30：15-slot D-type female connector，connect to X driver
- XS31：15-slot D-type female connector，connect to Z driver
- XS32：15-slot D-type female connector，connect to spindle encoder
- XS36：9-slot D-type female connector，connect to RS232 interface of PC
- XS37：9-pin D-type male connector，connect to frequency changer.
- XS38：9-pin D-type male connector，connect to manual pulse generator（MPG）
- XS39：25-slot D-type female connector，the interface from CNC to machine
- XS40：25-pin D-type male connector，the interface from machine to CNC
- XS41：25-pin D-type male connector，the expanding input interface
- XS42：25-slot D-type female connector，the expanding output interface

1.1.3 Total Connection Diagram

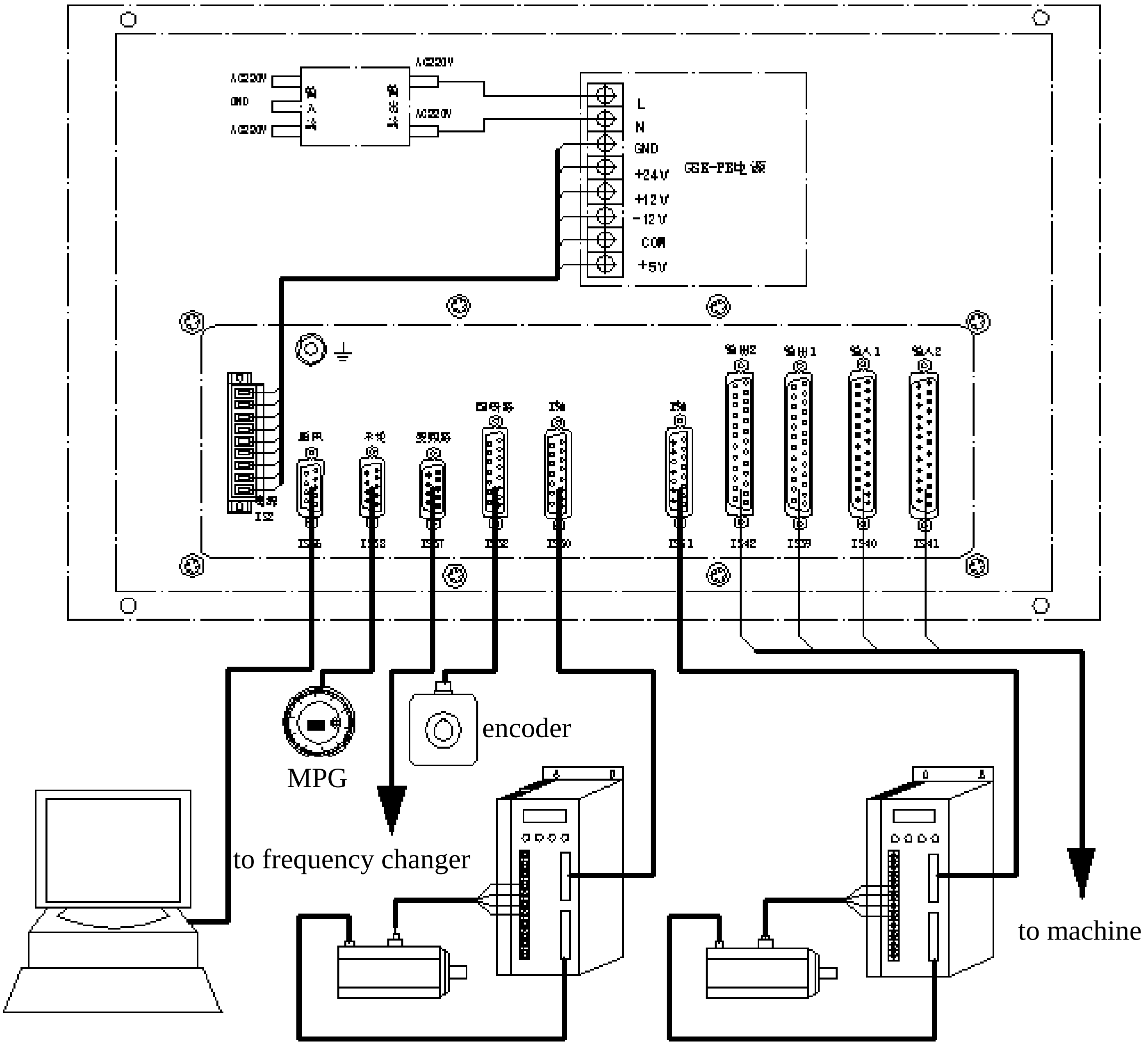


Fig.1-2 total connection diagram

1.2 INSTALLATION of GSK980TD

1.2.1 Outline Dimensions of GSK980TD

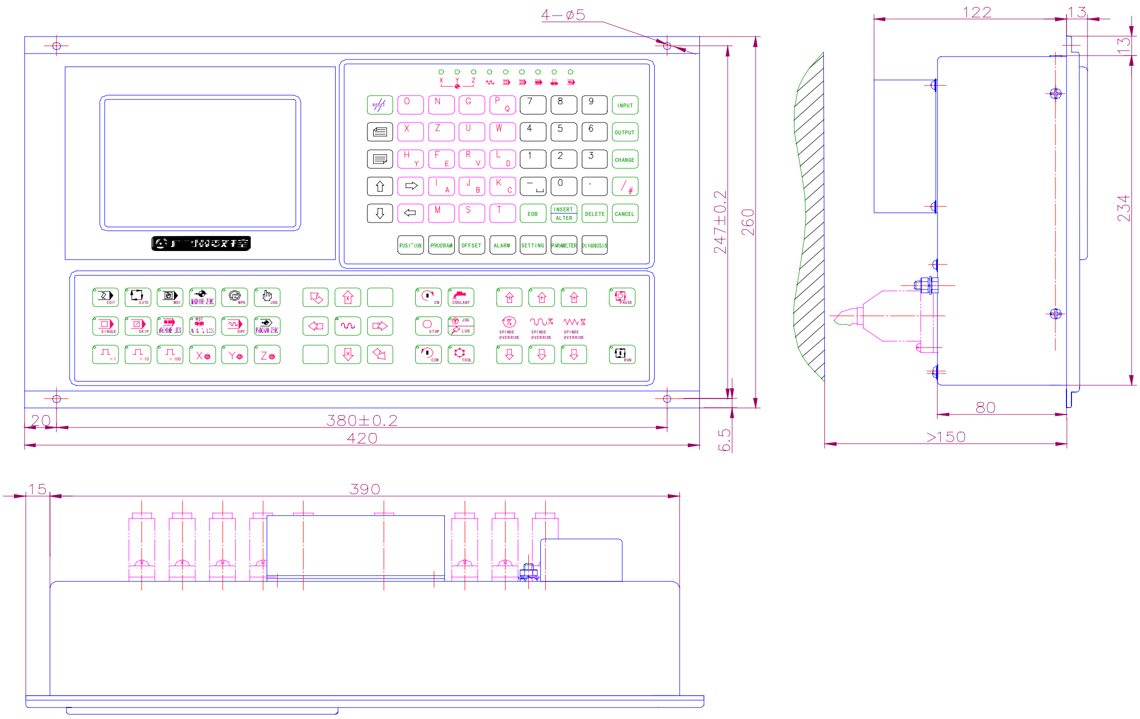


Fig.1-3 outline dimensions of GSK980TD

1.2.2 Outline Dimensions of GSK980TD-B

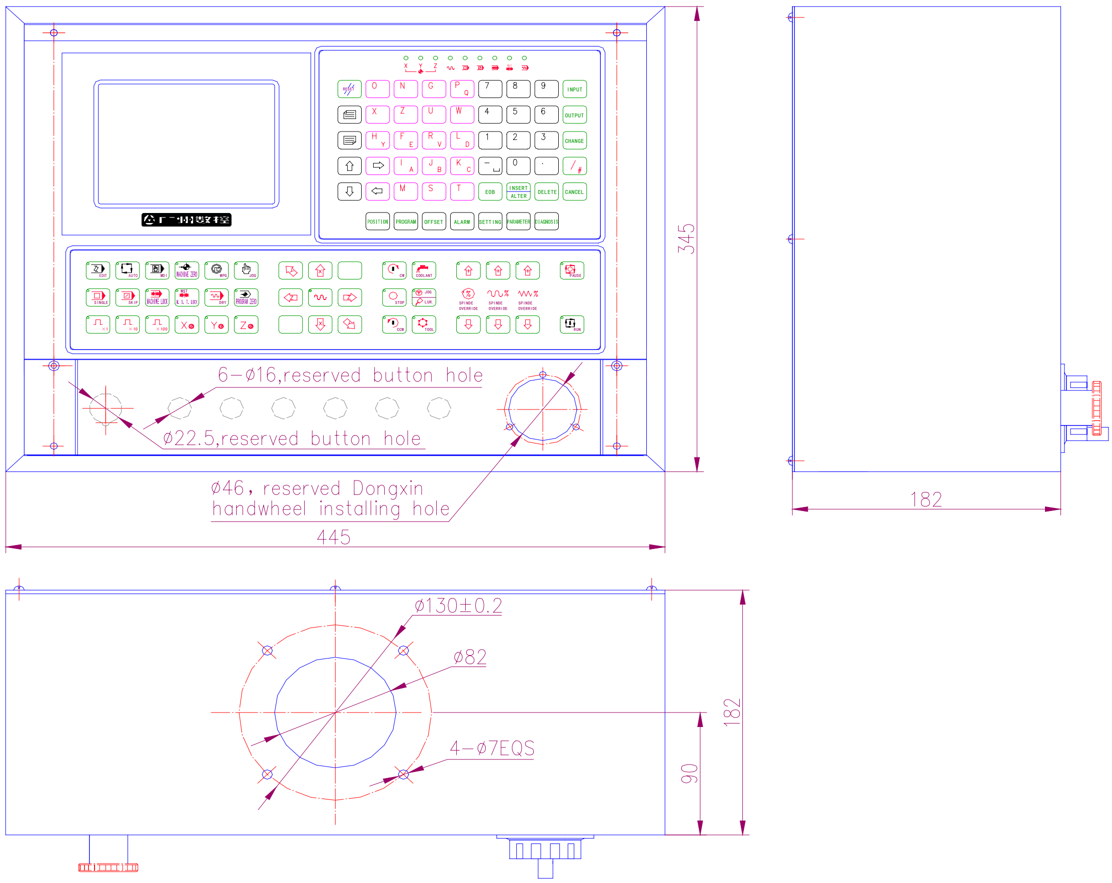


Fig.1-4 outline dimensions of GSK980TD-B

1.2.3 Installation Requirement of the Machine Electromagnetism Cabinet

- The cabinet must be fully closed and designed to prevent the entry of airborne dust、coolant、and organic solvent.
- The cabinet must be designed to maintain a difference in temperature of 10℃ or less between the air in the cabinet and the outside air when the temperature in the cabinet increases. The distance between back cover of CNC and surface of cabinet isn't less than 20cm.
- A closed cabinet must be equipped with a fan to circulate the air within.
- The display panel must be installed in a location where coolant cannot be poured directly on it.
- Cabinet design to minimize noise generation and to prevent it from being transmitted to the CNC is necessary.

1.2.4 Action Against Noise

The CNC has been designed to be protected from external noise. However，it is difficult to measure the lever and frequency of noise quantitatively，and noise has many uncertain factors. In order to improve the stability of the CNC，guard against noise in the machine as following：

1：CNC should be apart from the units which generating noise，such as frequency inverter、AC contactor、static generator、high voltage generator etc.

2：Provide the CNC with power supply by isolated transformer，machine tool fixed with CNC must be grounded and the independent grounding cables are required for CNC and servo driver.

3：Noise suppressor：Parallel connect RC-type loop with AC coil（as fig.1-5）and RC-type loop must be as near as possible to inductance load. Conversely parallel connect diode with DC coil（as fig.1-6）.Parallel connect surge absorber with coil of AC motor.

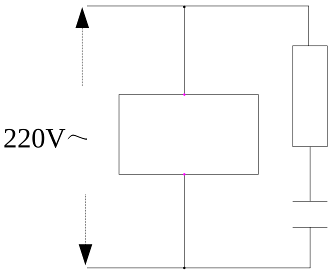


fig.1-5

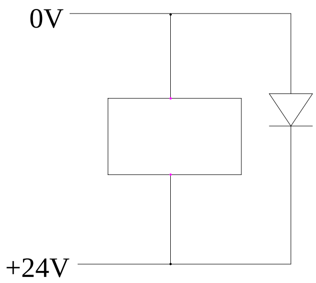


fig.1-6

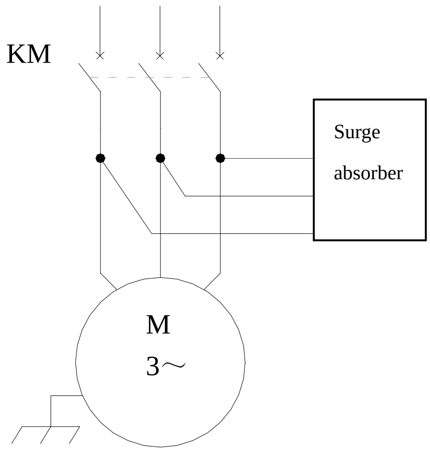


fig.1-7

4：The intertwist shielding or shielding cables as short as possible are absolutely necessary for the CNC，shielding layer of which is single-port grounding in CNC side.

5：Process the cables in each group as described in the action column.

Group	Signal line	Action
A	AC power lines	Bind the cables in group A separately from groups B and C，(the groups must be 10cm or more apart from one another） or cover group A with an electromagnetic shield
	AC solenoid	
	AC relay	
B	DC solenoid（24VDC）	Bind the cables in group B separately from group A，or cover group B with an electromagnetic shield. Separate group B as far from group C as possible
	DC relay（24VDC）	
	Cable between the CNC and power magnetics cabinet	
	Cable between the CNC and machine	
C	Cable between the CNC and servo driver	Bind the cables in group C separately from group A，or cover group C with an electromagnetic shield. Separate group C as far from group B as possible. cables apply twisted-pair
	Cable for position feedback	
	Cable for the position encoder	
	Cable for the manual pulse generator	
	Other cables to be covered with the shield	

C	Cable between the CNC and servo driver	Bind the cables in group C separately from group A，or cover group C with an electromagnetic shield.Separate group C as far from group B as possible.cables apply twisted-pair
	Cable for position feedback	
	Cable for the position encoder	
	Cable for the manual pulse generator	
	Other cables to be covered with the shield	

Chapter 2　DEFINITION and CONNECTION of INTERFACE

2.1　CONNECTION to DRIVER

2.1.1　Driver Interface

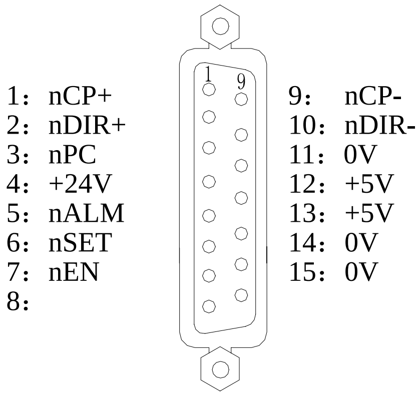


Fig.2-1　interface of XS30、XS31
（15-slot D-type female connector）

Signal Name	Description
nCP+, nCP-	Pulse signal
nDIR+, nDIR-	Direction signal
nPC	Zero signal
nALM	Driver alarm signal
nEN	Axis enabling signal
nSET	Pulse forbidden signal

Note：n means X or Z，the same as following.

2.1.2　Pulse Signal and Direction Signal

nCP+，nCP- are pulse signal and nDIR+，nDIR- are direction signal，which output in the difference mode（AM26LS31）.Advise to receive these signal outside by AM26LS32，and the inside circuit of which as fig.2-2：

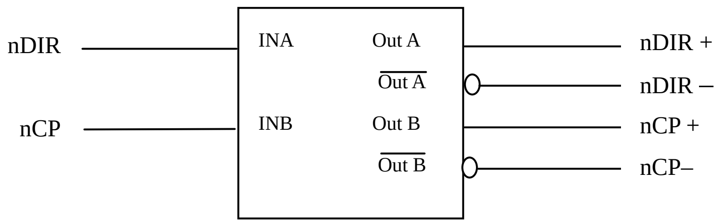


Fig.2-2　the inside circuit of nCP＋/－,nDIR+/－

2.1.3　Driver Alarm Signal　nALM

Driver alarm voltage can be specified to high or low level by Bit0 and Bit1 of No.009 parameter of the CNC. The inside circuit of nALM as fig.2-3：

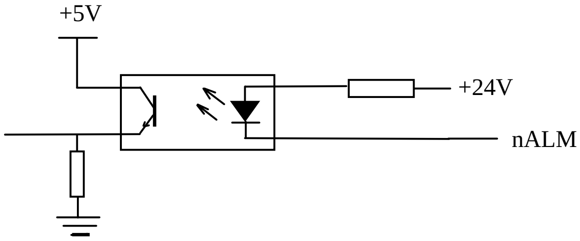


Fig.2-3 the inside circuit of nALM

In such input circuit, recommend to provide driver with the signal as fig.2-4

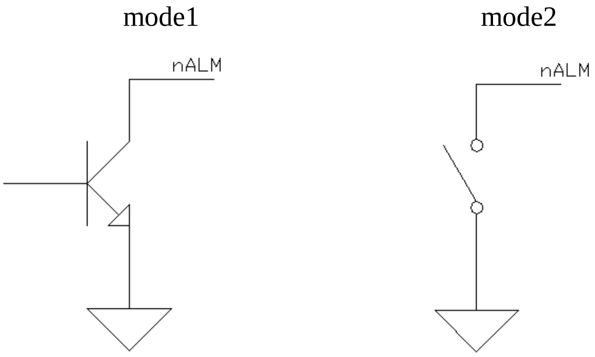


Fig.2-4 the mode of driver providing signal

2.1.4 Axis Enabling Signal nEN

nEN is valid (being 0V) in normal operation and is invalid (not being 0V) at driver alarm or emergency stop. The inside circuit as fig.2-5:

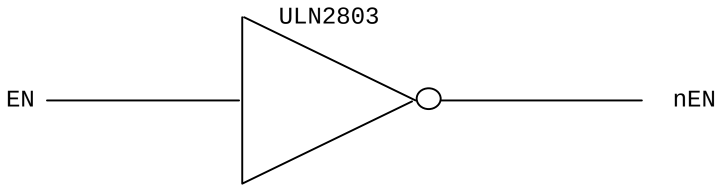


Fig.2-5 the inside circuit of nEN

2.1.5 Pulse Forbidden Signal nSET

nSET is to control servo driver input and increase the ability of eliminating noise between the CNC and driver, which is low voltage when outputting the pulse signal and is high impedance otherwise. The inside circuit as fig.2-6:

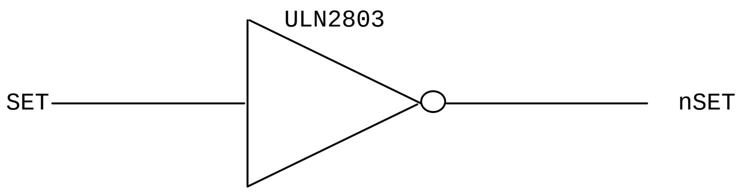


Fig.2-6 the inside circuit of nSET

2.1.6　Zero Signal　　nPC

In reference point return，zero signal is the encoder phase C signal or is the approach switch signal. The inside circuit as fig.2-7：

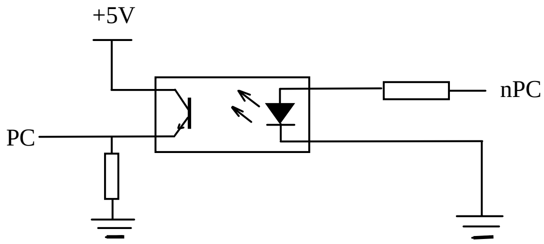


Fig.2-7　the inside circuit of nPC

Note：nPC is valid at ＋24V

a） User should provide nPC as fig.2-8：

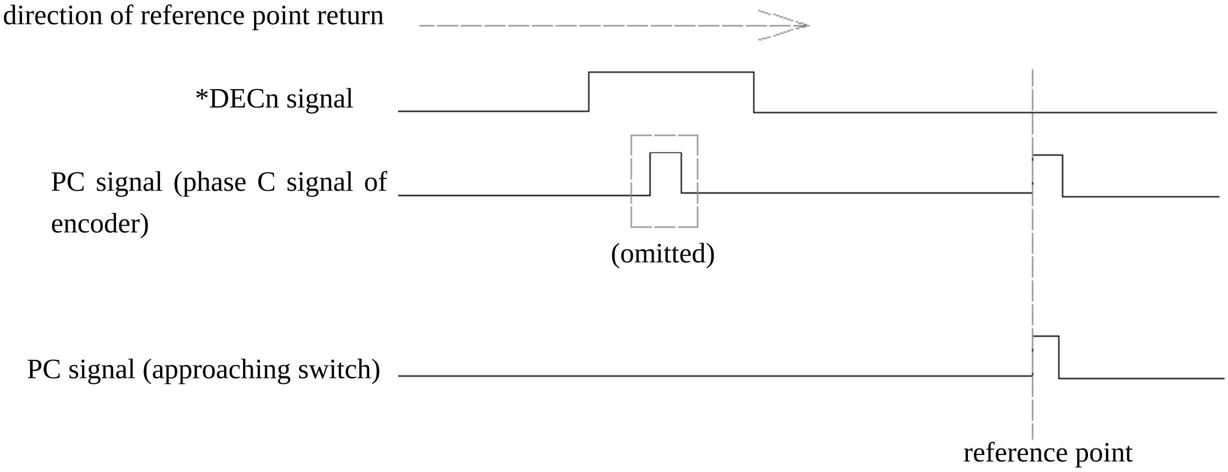


Fig.2-8

Note：In reference point return，the reference point position is specified by detecting skip of nPC（rising or descending edge is valid）after decelerating switch coming away.

b） Connection mode as fig.2-9 when both decelerating signal and zero signal are rooted in one NPN type HALL component.

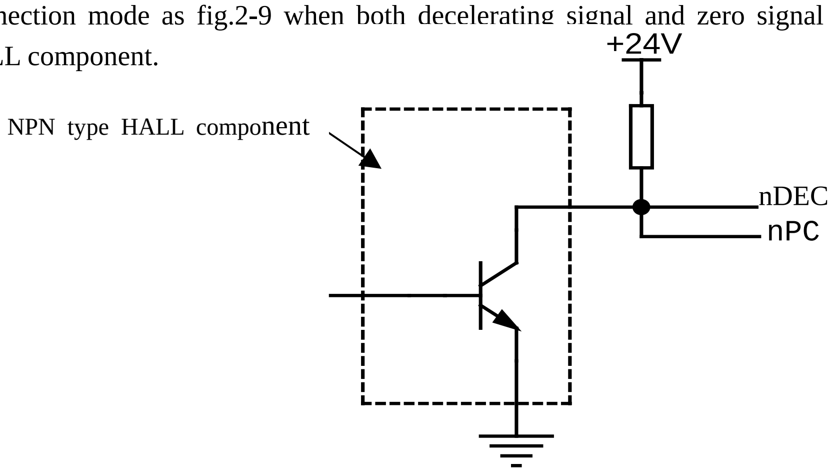


Fig.2-9　connection of NPN type HALL component c） Connection mode as fig.2-9 when both decelerating signal and zero signal are rooted in one PNP type HALL component .

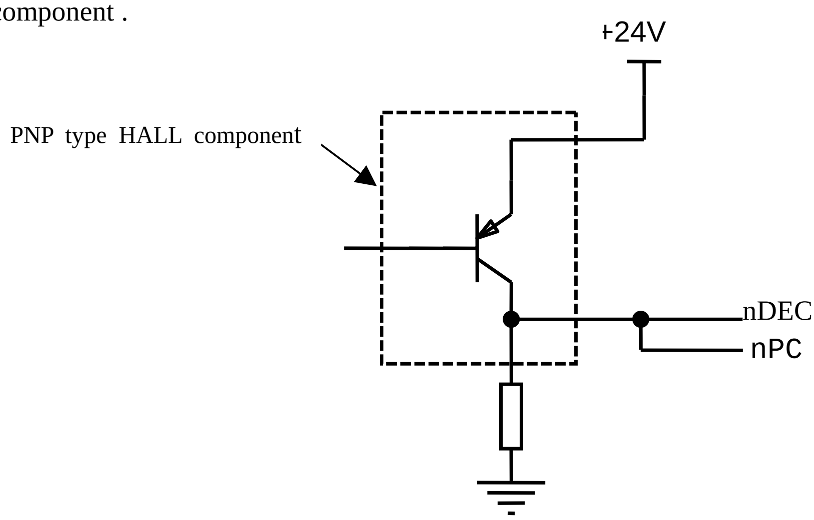


Fig.2-10 connection of PNP type HALL component

2.1.7 Connection to Driver

Connection between GSK980TD and the driver as fig.2-11

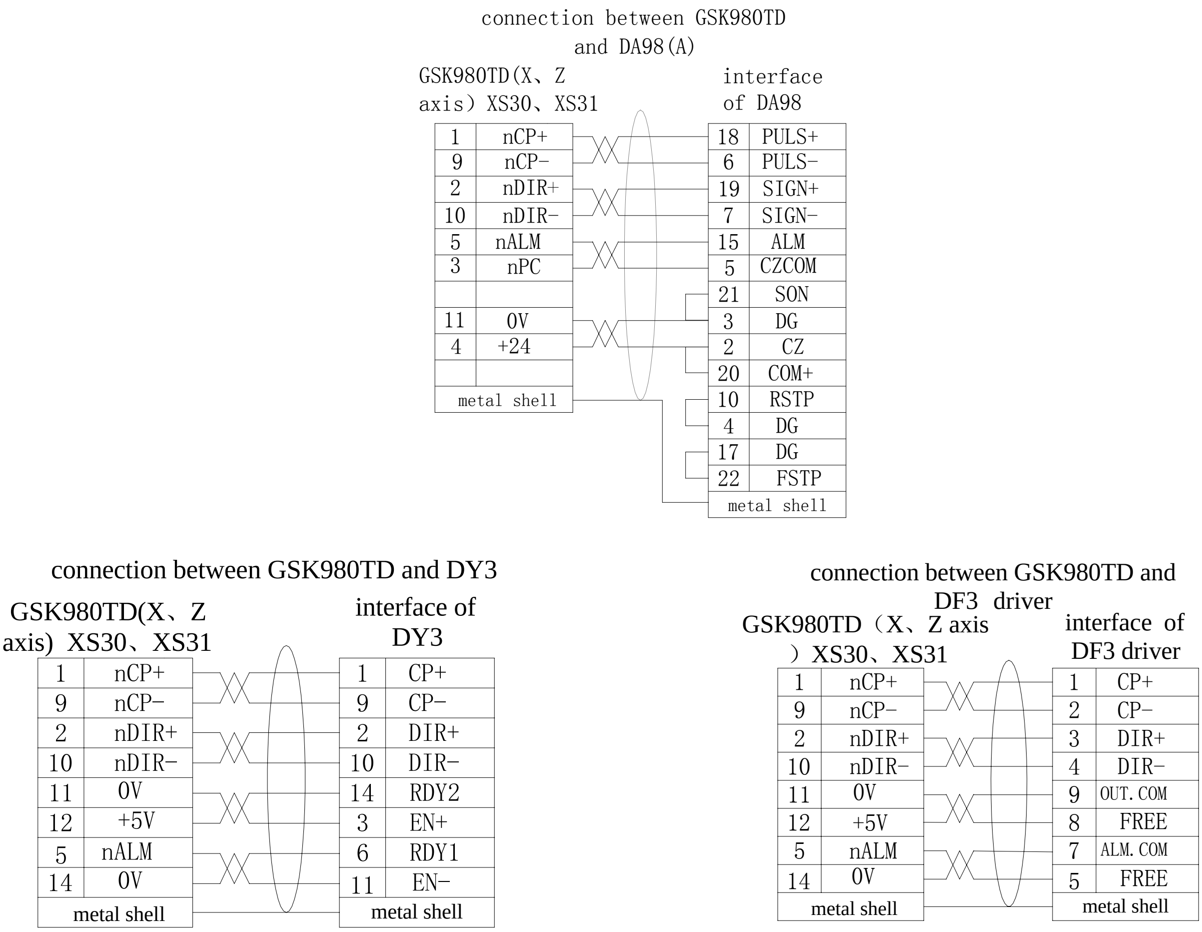
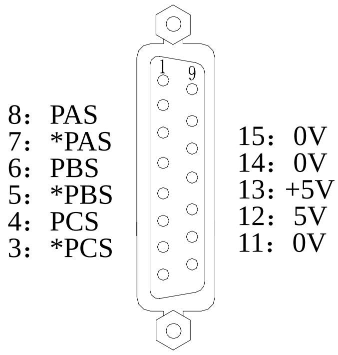


Fig.2-11 connection between GSK980TD and drivers

2.2 CONNECTION to SPINDLE ENCODER

2.2.1 Spindle Encoder Interface



Signal Name	Description
*PAS/PAS	Encoder phase A signals
*PBS/PBS	Encoder phase B signals
*PBS/PBS	Encoder phase C signals

Fig.2-12 encoder interface of XS32
（15-slot D-type female connector）

2.2.2 Signal Description

*PCS/PCS,*PBS/PBS and *PAS/PAS are the encoder phase C,B and A difference signals separately，which are received by 26LS32. *PAS/PAS,*PBS/PBS are square wave with 90° phase difference，the highest frequency of which is less than 1MHz.Encoder lines of GSK980TD can be specified at will from 100～5000 by the parameter.

The inside circuit as fig.2-13（n＝A,B,C）

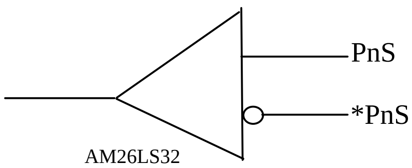


Fig.2-13 the circuit of encoder signal

2.2.3 Connection of Spindle Encoder Interface

Connection between GSK980TD and spindle encoder as fig.2-14 and the twisted-pair is in need（take Changchunyiguang ZLF-12-102.4BM-C05D encoder as example ）：

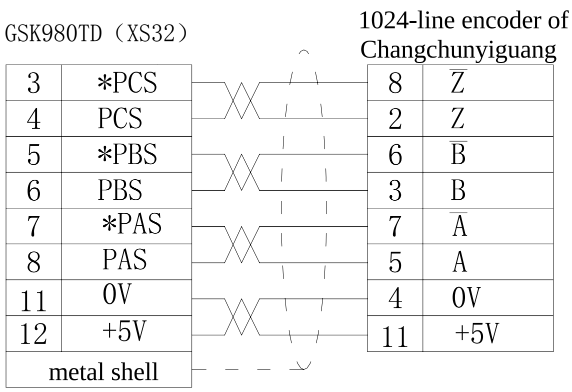


Fig.2-14 connection between GSK980TD and spindle encoder

2.3 CONNECTION to MANUAL PULSE GENERATOR

2.3.1 Manual Pulse Generator Interface

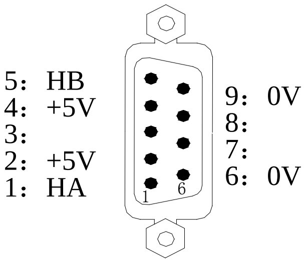


Fig.2-15 manual pulse generator interface of XS38
（9-pin D-type male connector）

Signal Name	Description
HA	Manual pulse generator phase A signal
HB	Manual pulse generator phase B signal
+5V,0V	DC power

2.3.2 Signal Description

HA and HB are the phase A,B input signal of manual pulse generator separately.

The inside circuit as fig.2-16：

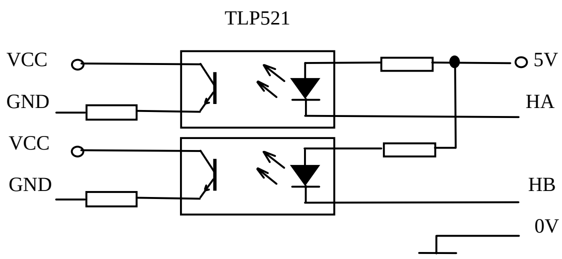


Fig.2-16 the circuit of manual pulse generator signal

Connection between GSK980TD and the manual pulse generator as fig.2-17：

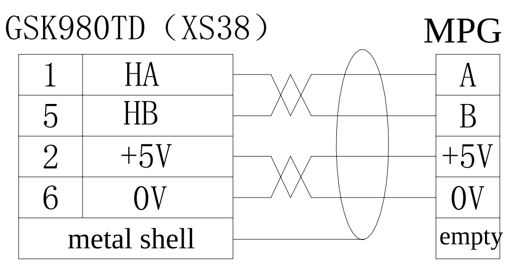
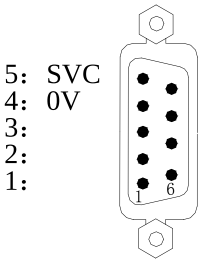


Fig.2-17 connection between GSK980TD and the manual pulse generator

Book3 connection

2.4 CONNECTION to FREQUENCY CHANGER

2.4.1 Analog Spindle Interface



Signal Name	Description
SVC	0～10V analog voltage
0V	Signal ground

Fig.2-18 analog spindle interface of XS37

（9-pin D-type male connector）

2.4.2 Signal Description

SVC port can output 0～10V voltage. The inside circuit as fig.2-19：

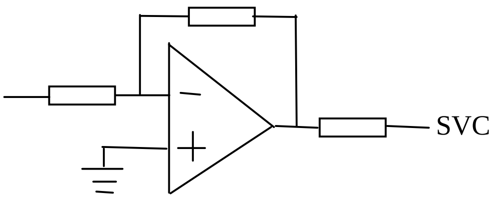


Fig.2-19 the inside circuit of SVC

2.4.3 Connection of Frequency Changer

Connection between GSK980TD and frequency changer as fig.2-20：

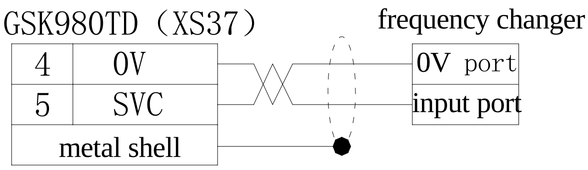
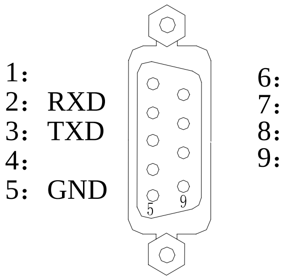


Fig.2-20 connection between GSK980TD and frequency changer

2.5 CONNECTION between GSK980TD and PC

2.5.1 Communication Interface



Signal Name	Description
RXD	Receive data
TXD	Send data
GND	Signal ground

Fig.2-21 communication interface of XS36
（9-slot D-type female connector）

2.5.2 Connection of Communication Interface

GSK980TD may communicate with PC by RS232 interface（GSK980TD communication software is in need）.Connection between GSK980TD and PC as fig.2-22：

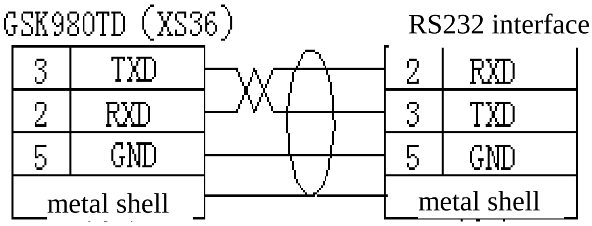


Fig.2-22 connection between GSK980TD and PC

2.6　POWER SUPPLY CONNECTION

GSK980TD adopts GSK-PB power box，which outputs +5V（3A）,+12V（1A）,-12V（0.5A）,+24V（0.5A），4 groups voltage and has one common port（COM）.What's more，connection between GSK-PB power box and XS2 interface has been finished， users only connect the power box to 220VAC power.

Connection between GSK-PB power box and XS2 interface of GSK980TD as fig.2-23：

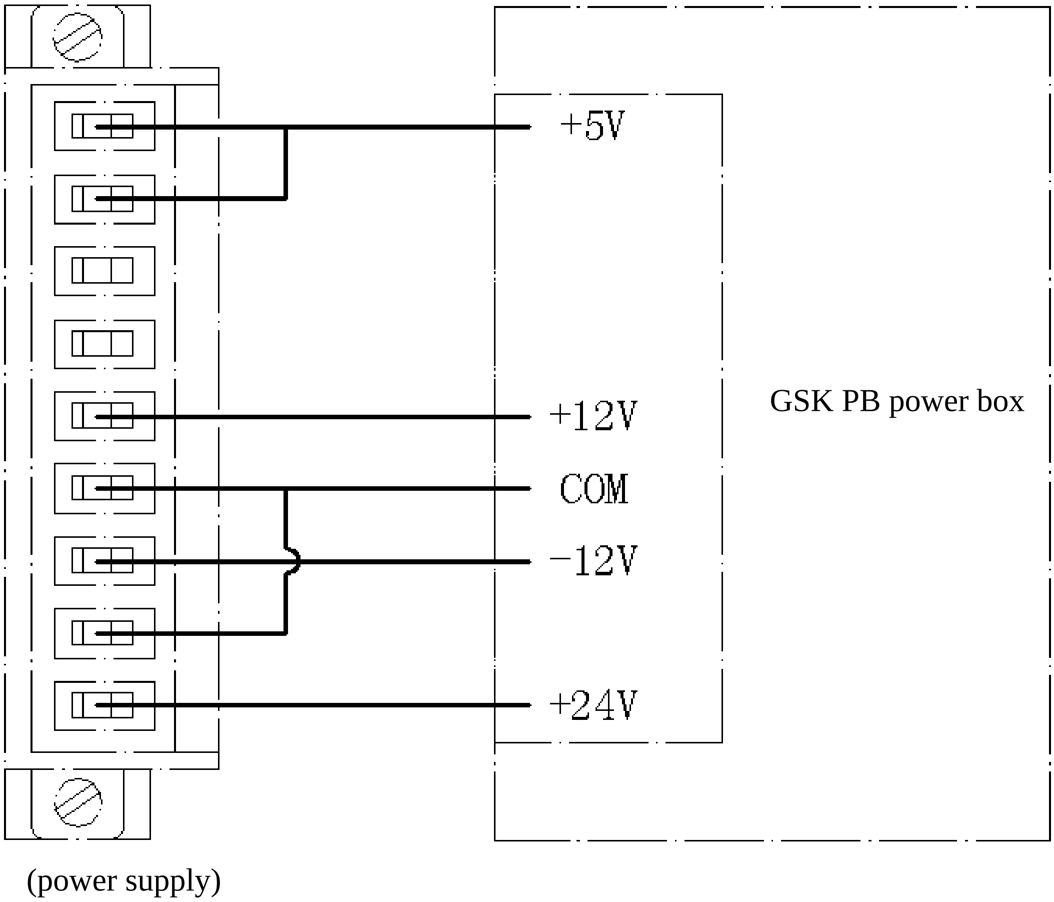


Fig.2-23

2.7 STANDARD and EXPANDED I/O INTERFACE

Note

Generally，the I/O interface of GSK980TD can be defined by PLC program（ladder）in assembling machine，which is decided by machine manufacturer，please read the manual of machine manufacturer for details.

This section describes the I/O interface according to the standard PLC program of GSK980TD.

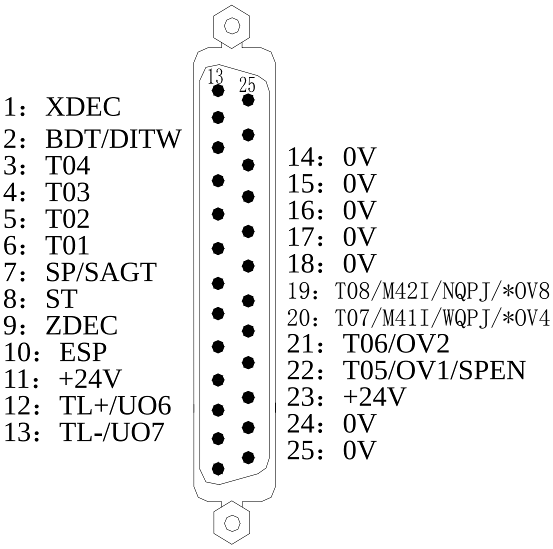
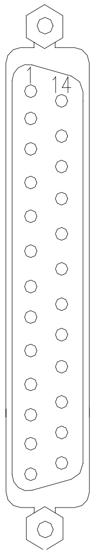


Fig.2-24 machine input interface of XS40
（25-pin D-type male connector）

Signal name	Description
XDEC,ZDEC	Decelerating signal in reference point return
BDT/DITW	Optional skip/tailstock control signal
T01～T08	Tool position signal
SP/SAGT	Feed keep/defence gate detecting signal
ST	Cycle start signal
ESP	Emergency stop signal
TL+,TL-	Positive rotation, negative rotation signal for toolpost
M42I,M41I	In-position signal for spindle auto gearing
NQPJ,WQPJ	In-position signal for chuck clamping or chuck unclamping
SPEN	Spindle rotation permission signal
OV1～OV8	Override signal

1: S2/M42/UO1
2: DOTWJ
3: M04
4: DOQPJ
5: S1/M41/UO0
6: M32
7: M03
8: S4/M44/UO3
9: UO4/DOTWS
10: DOQPS/UO5
11: DIQP
12: PRES
13: +24V



14: S3/M43/UO2
15: M08
16: M05
17: SPZD
18: 0V
19: 0V
20: 0V
21: 0V
22: 0V
23: 0V
24: 0V
25: +24V

Fig.2-25 machine output interface of XS39
（25-slot D-type female connector）

Signal name	Description
S1~S4	Spindle gear signal
M03, M04, M05	Spindle positive rotation,negative rotation signal and spindle stop signal
M08	Coolant on signal
DOTWJ, DOTWS	Tailstock advancing,withdrawing signal
DOQPJ, DOQPS	Chuck clamping,unclamping signal
M32	Lubrication on signal
PRES	Pressure detecting signal
SPZD	Spindle brake signal
M41~M44	Spindle auto gearing signal
UO1～UO4	User output
24V	24VDC output
0V	Common
DIQP	Chuck control input signal

Note 1: Some I/O interface, which is figured by "/", can be defined to multiplexer function.

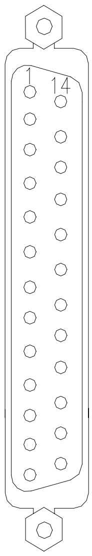
Note 2: Output signal is 0V when its function is valid, otherwise, output signal is high impedance.

Note 3: Input signal is 24V when its function is valid, otherwise, input signal is 0V.

Note 4: ＋24V and 0V are equivalent to the homonymy ports of power box.

XS41（extended input）and XS42（extended output）are optional interface，which contains 16 ports severally.

1: (X2.0)
2: (X2.7)
3: (X3.0)
4: (X3.7)
5: (X3.5)
6: (X3.6)
7: (X3.3)
8: (X3.1)
9: (X2.5)
10: (X2.3)
11: +24V
12: (X2.2)
13: (X2.1)



14: COM
15: COM
16: COM
17: COM
18: COM
19: (X3.4)
20: (X3.2)
21: (X2.6)
22: (X2.4)
23: +24V
24: COM
25: COM

Fig.2-26 extended input interface of XS41
（25-pin D-type male connector）

Signal Name	Description
24V	24VDC output
COM	Common
X2.0	Extended input
X2.1	Extended input
X2.2	Extended input
X2.3	Extended input
X2.4	Extended input
X2.5	Extended input
X2.6	Extended input
X2.7	Extended input
X3.0	Extended input
X3.1	Extended input
X3.2	Extended input
X3.3	Extended input
X3.4	Extended input
X3.5	Extended input
X3.6	Extended input
X3.7	Extended input

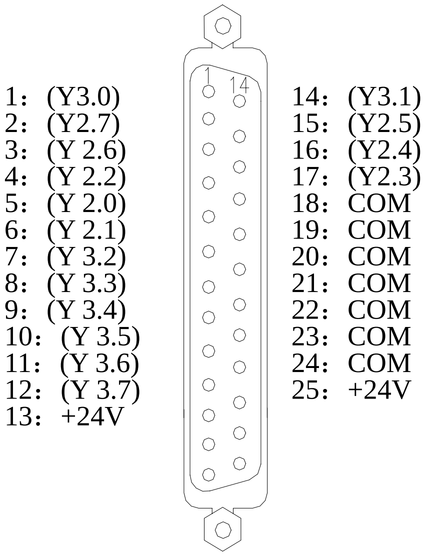


Fig.2-27 extended output interface of XS42
（25-slot D-type female connector）

Signal Name	Description
24V	24VDC output
COM	Common
Y2.0	Extended output
Y2.1	Extended output
Y2.2	Extended output
Y2.3	Extended output
Y2.4	Extended output
Y2.5	Extended output
Y2.6	Extended output
Y2.7	Extended output
Y3.0	Extended output
Y3.1	Extended output
Y3.2	Extended output
Y3.3	Extended output
Y3.4	Extended output
Y3.5	Extended output
Y3.6	Extended output
Y3.7	Extended output

2.7.1 Input Signal

Input signal is the signal from machine to CNC，which is valid when connecting to＋24V，otherwise，is invalid. The signal conditions as following is necessary：

Contact capability：voltage more than 30VDC and current more than 16mA

Leakage current between contacts in open circuit：less than 1mA

Drop voltage between contacts at short：less than 2V（current is 8.5mA，including the drop voltage of cables）

Two input modes are suitable for external input signal：one is to adopt the contact switch，in which the input signal comes from the key, travel-limit switch and relay contact etc.

The connection as fig.2-28：

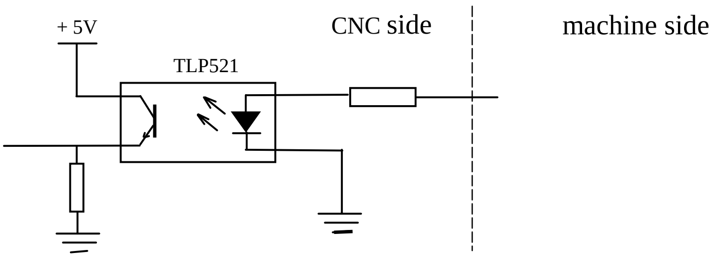


Fig.2-28

The other is to adopt the non-contact switch（transistor）.The connection as fig.2-29,fig.2-30：

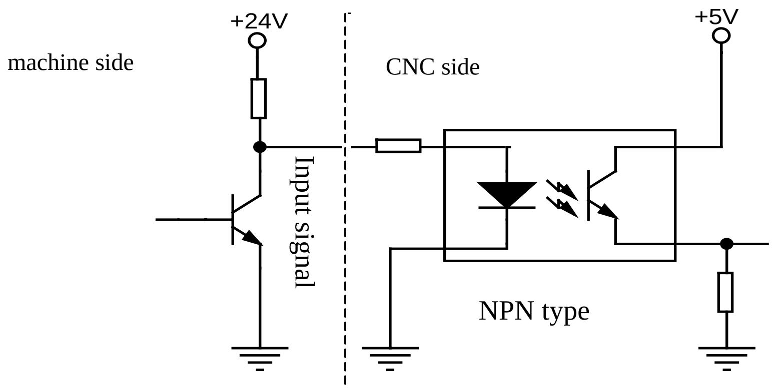


Fig.2-29　NPN connection

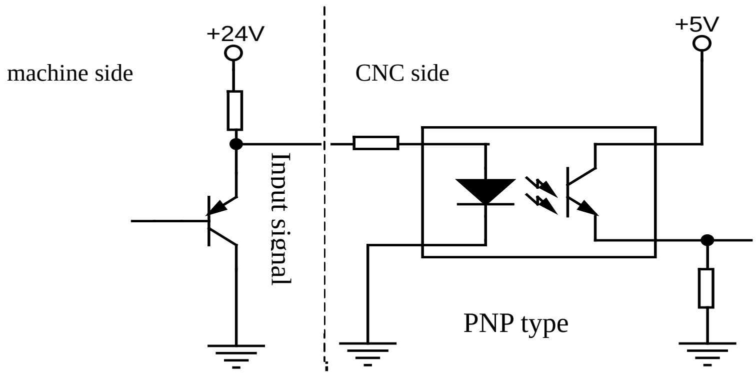


Fig.2-30　PNP connection

In the standard function，input interface includes XDEC,ZDEC,ESP,ST,SP/SAGT,BDT/DITW,DIQP,OV1～OV8,T01～T08 etc.

2.7.2　Output Signal

Output signal is to drive the relay or indicator lamp from machine side，which is valid when connecting to 0V，otherwise，is invalid. I/O interface contains 36 digital output signal with the same circuit as fig.2-31.

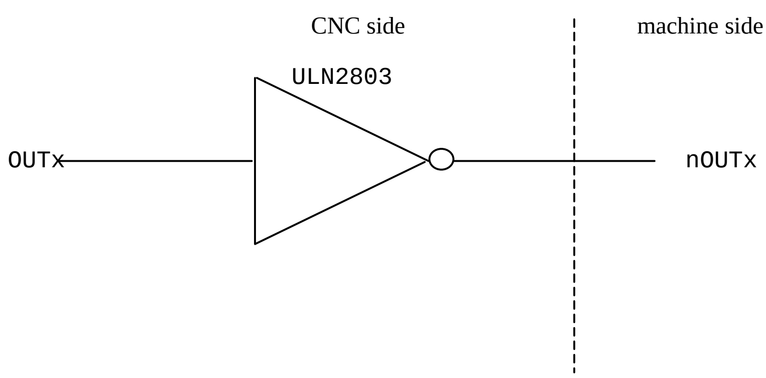


Fig.2-31　the circuit structure of digital output

OUTx from the main board streams into the input port of inverter（ULN2803）and nOUTx has two state of 0V and high impedance，with the typical application as fig.2-32：

- Drive LBD

A resistor in series is in need to drive LBD by ULN2803 as fig.2-32，which limits the current of LBD（general being about 10 mA）.

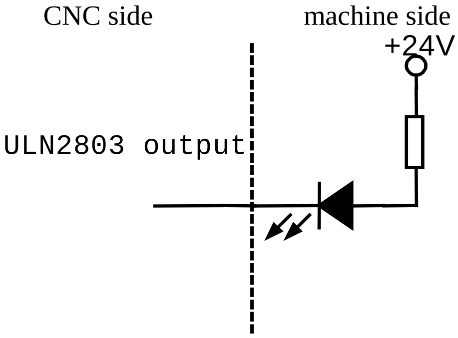


Fig.2-32

- Drive filament-type indicator lamp

When driving the filament indicator lamp by ULN2803，a warm-up resistor is in need to reduce the current concussion as fig.2-33，the impedance of which is on principle that the indicator lamp isn’t light.

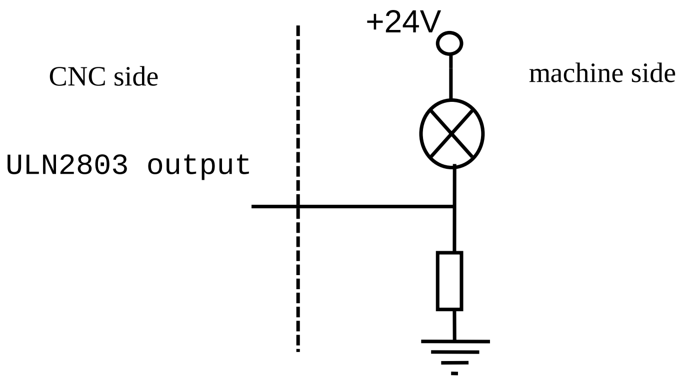


Fig.2-33

- Drive inductance load

To protect the circuit and reduce noise，a diode near the coil is needed to drive the inductance load by ULN2803 as fig.2-34：

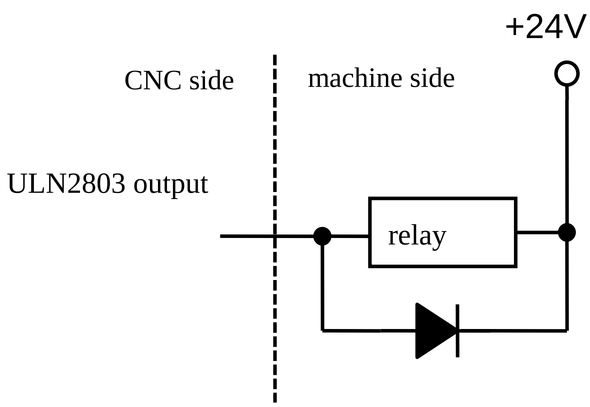


Fig.2-34

The output signals，defined by standard PLC program，includes S1～S4(M41～M44),M3,M4,M5,M8,M10,M11,M32,TL-,TL+,UO0～UO5,DOQPJ,DOQPS,SPZD，and the common port is ＋24V.

2.8　FUNCTION and CONNECTION of I/O INTERFACE

Note

Generally，the I/O interface of GSK980TD can be defined by PLC program（ladder）in assembling machine，which is decided by machine manufacturer，please read the manual of machine manufacturer for details.

This section describes the I/O interface according to the standard PLC program of GSK980TD.

2.8.1　Overtravel and Emergency Stop

- Relative signal

ESP：emergency stop signal，ESP alarm is given when disconnecting with ＋24V .

- Diagnosis data

0	0	1								
Interface pins							**XS40.10**			

			ESP				
			XS40.10				

- Control parameter

State parameter

1	7	2

				ESP			

ESP＝0　　Emergency stop function is valid

=1　　Emergency stop function is invalid

- Signal connection

The circuit of ESP as fig.2-35：

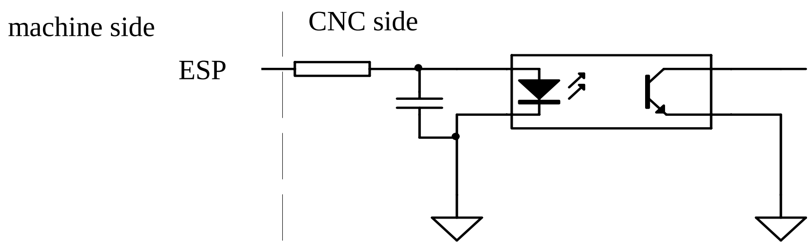


Fig.2-35

- Machine connection

The connection of emergency stop switch and overtravel canceling switch as fig.2-36

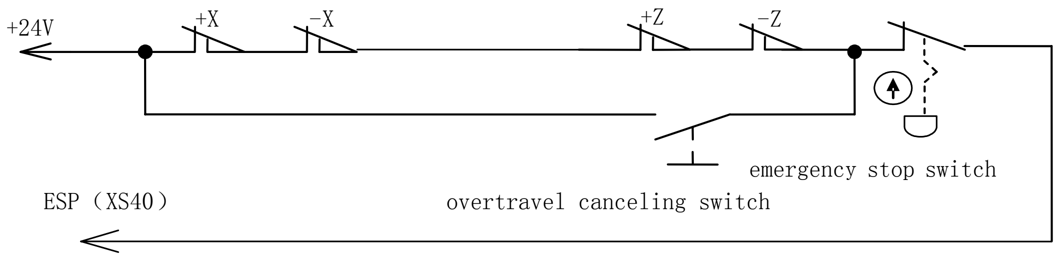


Fig.2-36

- Control logic

In the standard PLC program，M03 or M04,M08,enabling signal（EN）and pulse signal are closed，and M05 is outputted when ESP alarm， which caused by that ESP signal disconnects with ＋24V，is given.

2.8.2 Tool Changing Control

- Relative signal（defined by the standard PLC program）

T01~T04：1 to 4 tool signals from machine to CNC

T05~T08：5 to 8 tool signals from machine to CNC，in which T05,T07,T08 are the multiple ports separately with SPEN,M41I/WQPJ,M42I/NQPJ.

TCP：toolpost locking-up signal，which is the multiple port with PRES（pressure detecting signal）in the standard PLC program.

TL+,TL-：positive rotation,negative rotation signals.

- Diagnosis data

1、 input signal：T01~T08 signal

0 0 0					T04	T03	T02	T01
interface pins					XS40.3	XS40.4	XS40.5	XS40.6

0 0 2	T08	T07	T06	T05				
interface pins	**XS40.19**	**XS40.20**	**XS40.21**	**XS40.22**				

2、 output signal：TL+ positive rotation signal；TL- negative rotation signal

0 0 5	TL-	TL+						
interface pins	**XS40.13**	**XS40.12**						

- Control parameter

State parameter

0 1 1							TSGN	TCPS

TSGN＝0：T01～T08 are valid when connecting to +24V

=1：T01～T08 are valid when disconnecting with ＋24V

TCPS＝0：TCP signal is valid when connecting to ＋24V

＝1：TCP signal is valid when disconnecting with ＋24V

1 8 2						PB6		PB5

PB5＝0：tool changing mode A

＝1：tool changing mode B

PB6=0：don’t detect T01～T08 signal when tool changing is over

＝1：detect T01～T08 signal when tool changing is over

Data parameter

0	7	6	TIMAXT

Time upper limit for changing one tool

0	7	8	TLMAXT

Time upper limit for changing the most tools

0	**8**	**2**	**T1TIME**

Tool changing time 1：delay time from closing TL+ to outputting TL-

0	**8**	**4**	**TMAX**

Total tool number

0	**8**	**5**	**TCPTIME**

Tool changing time 2：time from outputting TL- to receiving TCP

● Signal connection

1. The optocoupler is used for T01～T08 and TCP，the inside circuit as fig.2-37：

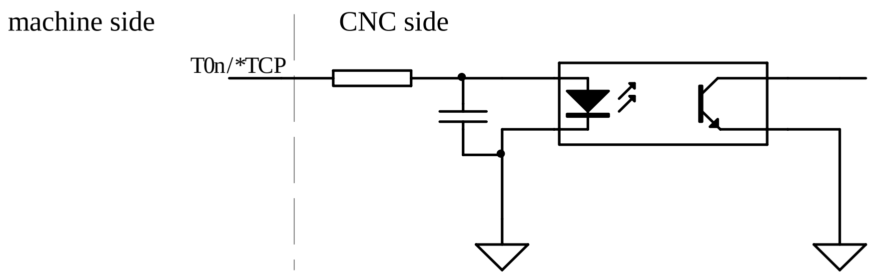


Fig.2-37

2. TL+ and TL- are positive/negative rotation signals，the inside circuit as fig.2-38：

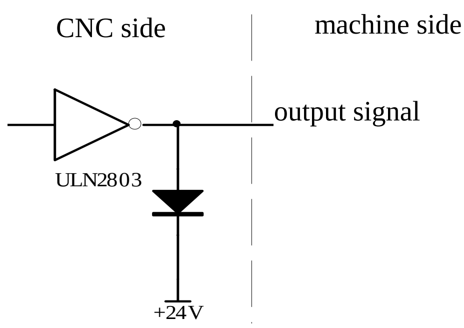


Fig.2-38

3. The connection of T01～T08 as fig.2-39，in which a pull-up resistor is needed when valid T01～T08 in low voltage.

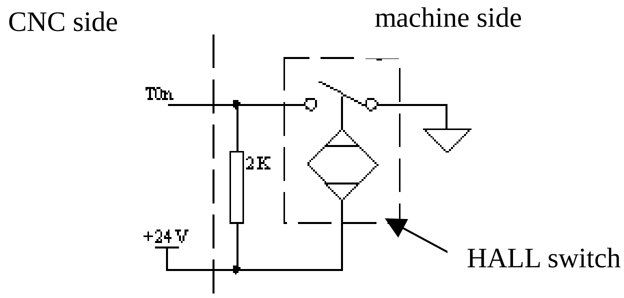


Fig.2-39

- Function description（defined by standard PLC program）

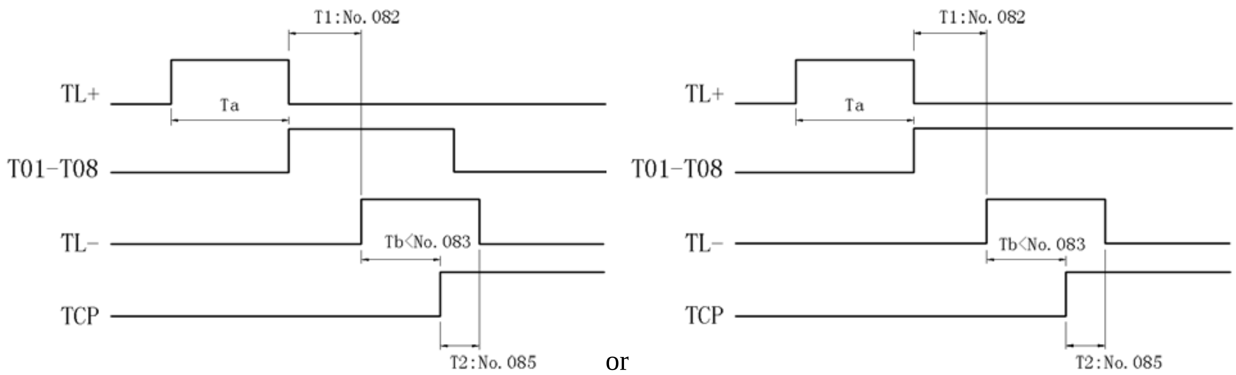
  In the standard PLC program，four modes for tool changing are defined as following.

  1、 PB5=0，PB6=0：mode B

  ① During the tool changing process，CNC outputs TL+ signal firstly until the tool signal（T01~T08）being detected，then CNC outputs TL- signal after the time specified by No.82 parameter from TL+ being closed，afterwards CNC detects TCP signal and closes TL- signal after the time specified by No.85 parameter from TCP being detected，and tool changing process is over.

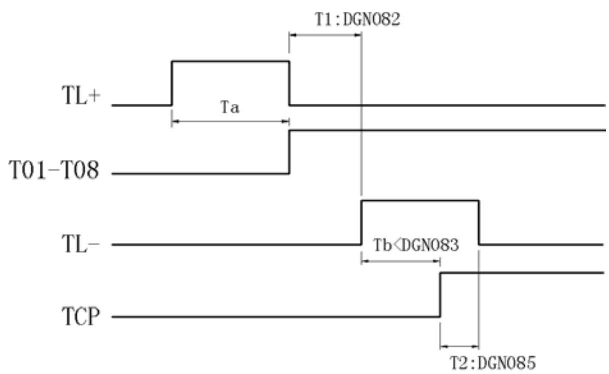
  ② CNC will give alarms and close TL- signal when not detecting TCP signal during the time specified by No.83 parameter after outputting TL- signal.

  ③ Set bit0（TCPS）of No.11 parameter to 0 if TCP signal doesn’t exist.



2. PB5=0，PB6=1：mode B（with in-position detecting）

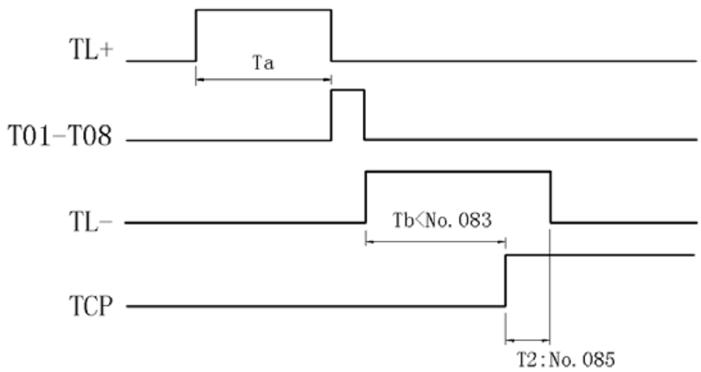
  The tool changing process is same as the mode B，except for the added step of confirming tool position，in which CNC will detect the tool signal at the moment of closing TL-，if the current tool position is in accordance with the current tool number，the tool changing process is over，otherwise，CNC will give “unfinished tool changing”alarm.



Book3 connection

2、 PB5=1，PB6=0：mode A

During the tool changing process，CNC outputs TL+ signal firstly until the tool signal（T01~T08）being detected，then CNC outputs TL- signal when skip of the tool signal being detected，afterwards CNC detects TCP signal and closes TL- signal after the time specified by No.85 parameter from TCP being detected，and tool changing process is over.

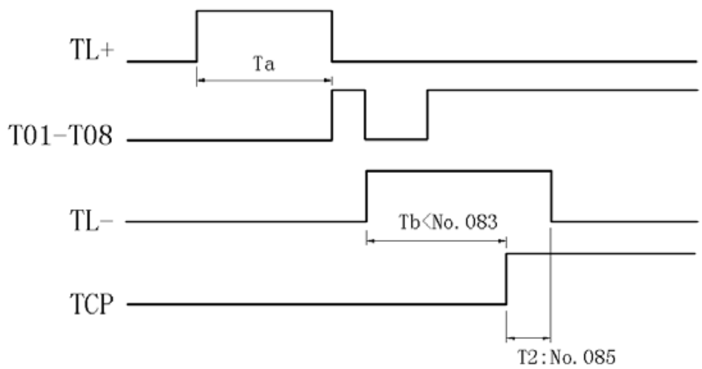


Note 1：No.82 parameter is invalid，that is to say，CNC don't check the delay time from closing TL+ to outputting TL-.

Note 2：Except No.82 parameter，the parameters and function for toolpost control are still valid.

3、 PB5=1，PB6=1：mode A（with in-position detecting）

The tool changing process is same as the mode A，except for the added step of confirming tool position，in which CNC will detect the tool signal at the moment of closing TL-，if the current tool position is in accordance with the current tool number，the tool changing process is over，otherwise CNC will give "unfinished tool changing" alarm.



Note:"tool changing time is too long"alarm will be given when Ta is more than the time specified by No.78 parameter.

2.8.3 Reference Point Return

● Relative signal

XDEC：decelerating signal of X axis

ZDEC：decelerating signal of Z axis

XPC：zero signal of X axis

ZPC：zero signal of Z axis

- Diagnosis data

0	0	0			XDEC					
interface pins					XS40.1					

0	0	1			ZDEC					
interface pins					XS40.9					

0	0	8							PCZ	PCX
interface pins									XS31.3	XS30.3

- Control parameter

State parameter

0	0	4			DECI					

DECI＝1：nDEC signal is valid when connecting to＋24V

＝0：nDEC signal is valid when disconnecting with ＋24V

0	0	5							PPD	

PPD＝1：G50 can set the relative coordinate

＝0：G50 can't set the relative coordinate

0	0	6							ZMZ	ZMX

ZMZ=1：reference point return mode C of Z axis

＝0：reference point return mode B of Z axis

ZMX=1：reference point return mode C of X axis

＝0：reference point return mode B of X axis

0	0	7							ZCZ	ZCX

ZCZ＝1：In reference point return，ZDEC signal is in parallel connection with PCZ signal（ZDEC and PCZ all come from one approach switch）

＝0：In reference point return，ZDEC signal and PCZ signal are connected independently

ZCX＝1：In reference point return，XDEC signal is in parallel connection with PCX signal（XDEC and PCX all come from one approach switch）

＝0：In reference point return，XDEC signal and PCX signal are connected independently

0	1	1						ZNLK		

ZNLK＝1：In reference point return，once the corresponding direction key is pressed，the axis keeps moving until the reference point return completed. To halt the movement，press "RESET" key

＝0：In reference point return，the axis movement will stop once the corresponding direction key is released

0	1	2	APRS							ISOT

APRS＝1：Automatically set the absolute coordinate system after reference point return completed，the coordinate value is specified by No.49 and No.50 parameter

＝0：Don't set the absolute coordinate system automatically after reference point return completed

0	1	4								ZRSZ	ZRSX

ZRSZ,ZRSX＝1：Reference point exists for Z,X axis，decelerating signal and zero signal need to be detected in reference point return

＝0：Reference point doesn't exist for Z,X axis，decelerating signal and zero signal don't need to be detected in reference point return

1	8	3								MZRZ	MZRX

MZRX,MZRZ =1：The direction of reference point return is negative for X,Z axis

＝0：The direction of reference point return is positive for X,Z axis

Data parameter

0	3	3	ZRNFL

ZRNFL：The low speed in reference point return for X,Z axis

0	4	9	PRSX
0	5	0	PRSZ

PRSX,PRSZ：The absolute coordinate setting value of X,Z axis after reference point return being completed

- Signal connection

The inside circuit of nDEC signal as fig.2-40

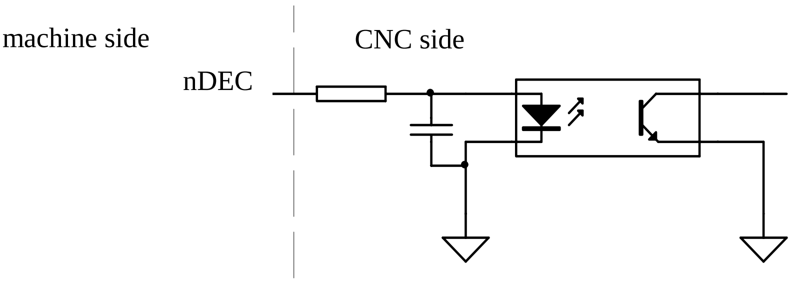
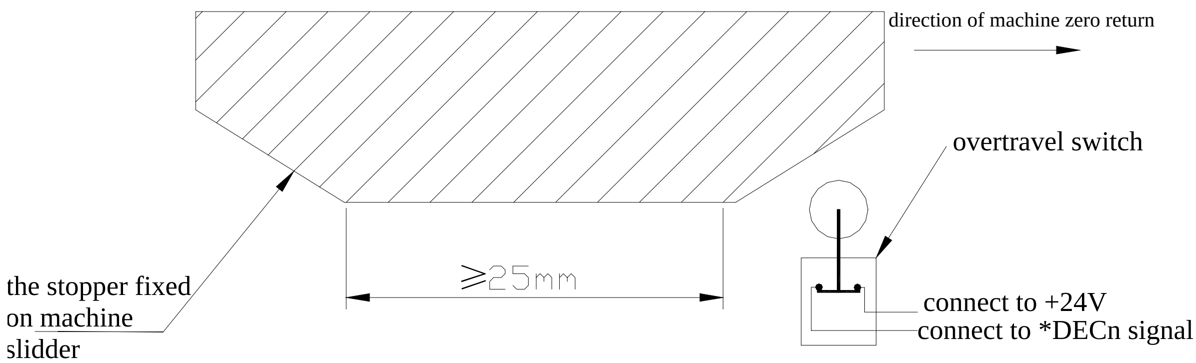


Fig.2-41

- Reference point return of taking the phase C signal of servo motor encoder as the zero signal

① The sketch map as following：



② The connection circuit of nDEC signals

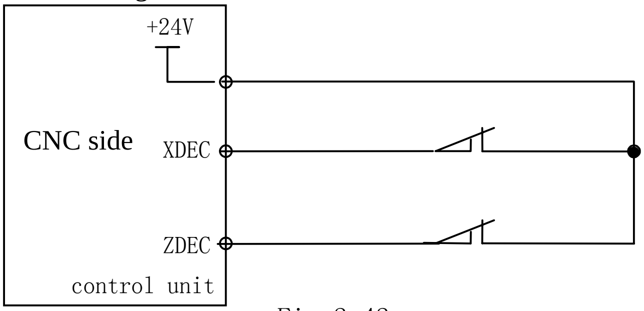


Fig. 2-42

③ The action logic for reference point return

The action logic for reference point return as fig.2-43，under the condition of setting BIT0（ZMX）, BIT1（ZMZ）of No.6 parameter and BIT5（DECI）of No.4 parameter to“0”，in which initial backlash direction is positive and nDEC signal is valid in low voltage.

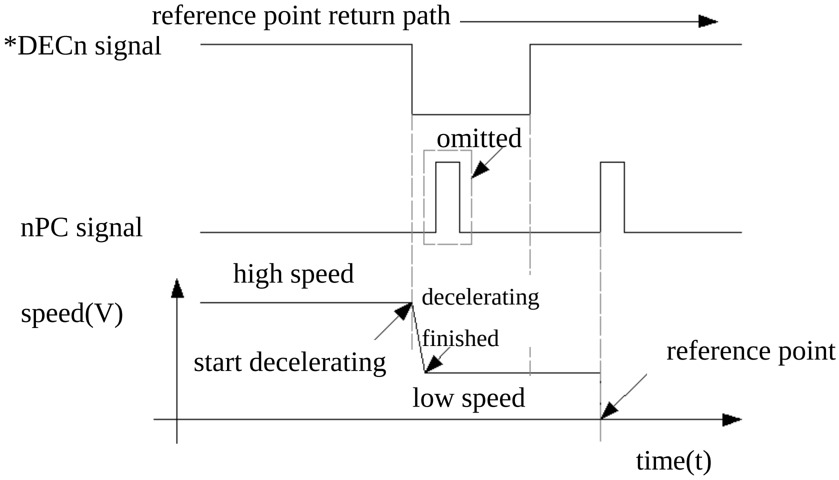


Fig.2-43

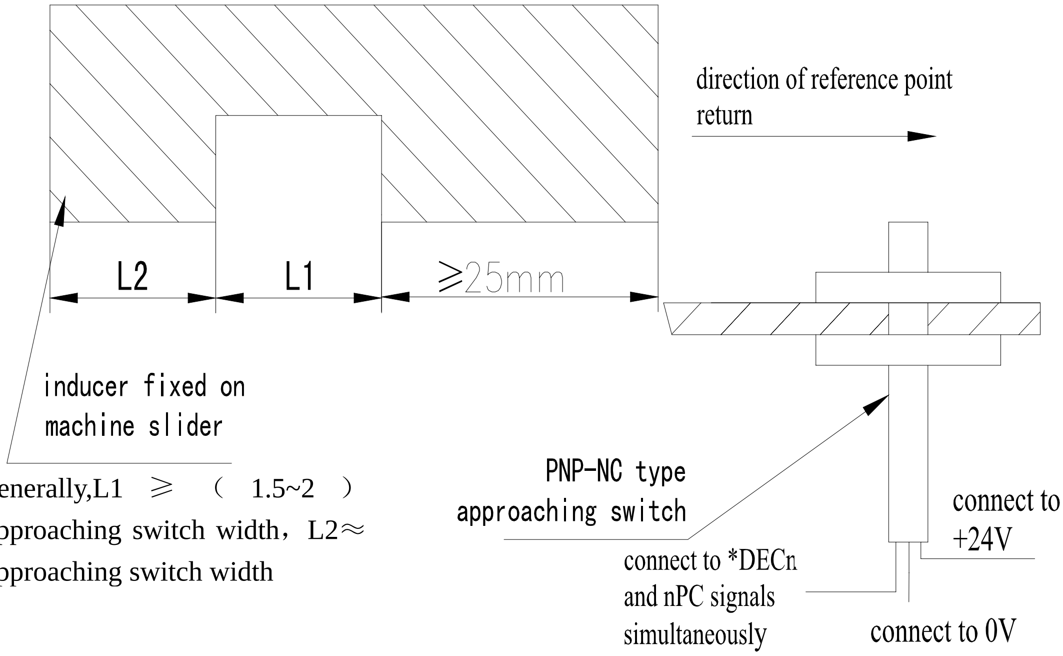
④ Reference point return process

A In REF mode，when pressing the corresponding move key（the direction specified by No.183 parameter），the axis moves to the reference point with the rapid traverse speed until pressing the decelerating switch，then keeps moving with the definite low feed speed.

B After releasing the decelerating switch，CNC detects the encoder phase C signal（PC）until PC signal skipping， then the axis stops moving and the corresponding indicator lamp for reference point return ending lights，and the reference point return process is over.

- Reference point return when taking a approach switch as decelerating signal and zero signal simultaneously

① The sketch map as following：



generally,L1 ≥ （ 1.5~2 ） approaching switch width，L2≈ approaching switch width

② The connection circuit of nDEC signals

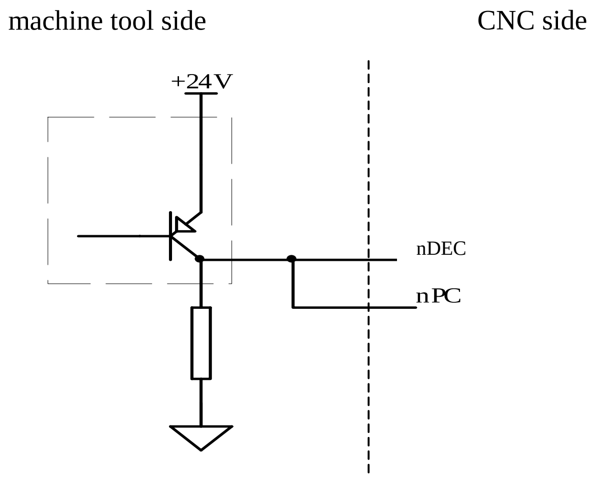


Fig.2-45

③ The action logic for reference point return

The action logic for reference point return as fig.2-46，under the condition of setting BIT0（ZMX）, BIT1（ZMZ）of No.6 parameter and BIT5（DECI）of No.4 parameter to“0”，in which initial backlash direction is positive and nDEC signal is valid in low voltage.

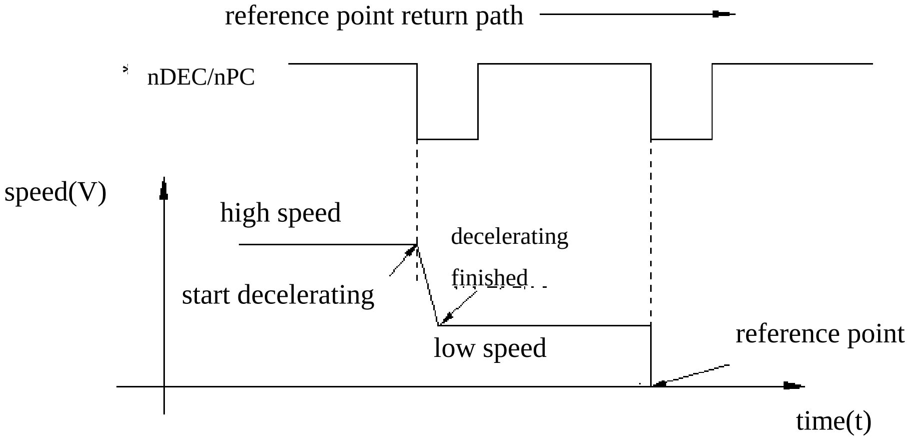


Fig.2-46

④ Reference point return process

A In REF mode, the axis moves to the reference point with the rapid traverse speed when pressing the corresponding move key (the direction specified by No.183 parameter).

B The axis moves with the definited low speed and nDEC signal is valid when the approach switch inducing the stopper firstly.

C The axis keeps moving and CNC begins to detect nPC signal when the approach switch departing away from the stopper.

D The axis stops moving and the indicator lamp for reference point return ending lights when the approach switch inducing the stopper secondly, and the reference point return process is over.

2.8.4 Spindle Positive or Negative Rotation Control

● Relative signal (defined by the standard PLC program)

M03: spindle positive rotation
M04: spindle negative rotation
M05: spindle stopping
SPZD: spindle braking

● Diagnosis data

diagnosis data

0	0	4	SPZD		M05				M04	M03
interface pins			XS39.17		XS39.16				XS39.3	XS39.7

● Control parameter

control parameter

0	0	9					RSJG			

RSJG=1: not close M03,M04,M08,M32 signals when pressing "RESET" key
= 0: close M03,M04,M08,M32 signals when pressing "RESET" key

data parameter

0	8	0	MTIME

Duration for M code

0	8	7	SPDDLT

Delay time for from executing M05 to outputting SPZD

Book3 connection

0	8	9	SPZDTIME

Duration for SPZD signal

- Signal connection

The output circuit of M03,M04,M05,SPZD as fig.2-47：

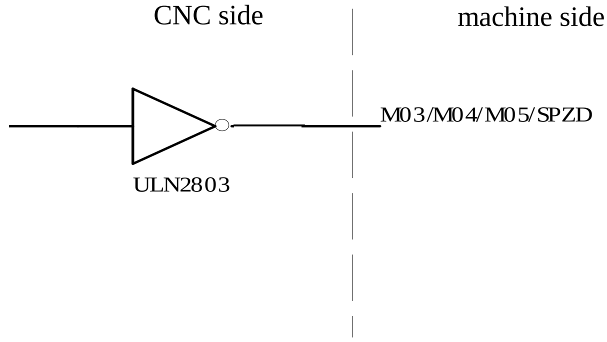


Fig.2-47

- Action logic（defined by the standard PLC program）

Spindle action logic as fig.2-48：

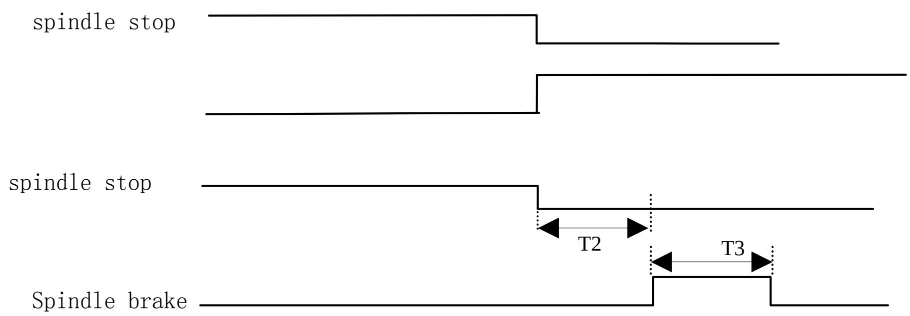


Fig.2-48

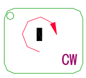
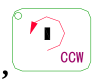
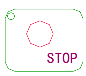
Note：T2 is the delay time for from executing M05 to outputting SPZD and T3 is duration for SPZD signal

- Control logic（defined by the standard PLC program）

M05 is valid when powering on until executing M03 or M04，then M03 or M04 is valid until executing M05.The corresponding time is specified by No.87 and No.89 parameters. M03 and M04 can't be valid simultaneously.

Note1：In emergency stop，CNC closes M03 or M04,M08 and outputs M05.

Note2：CNC closes M03 or M04 at reset when bit3 of No.9 parameter is 0，otherwise，keeps M03 or M04 at reset.

Note3：Spindle rotation can be controlled by  **,**  **,**  **keys，please read book2 of the manual for details.**

2.8.5 Spindle Speed Controlled by On-off Variable

- Relative signal（defined by the standard PLC program）

S01～S04：on-off signal of spindle speed control. The standard PLC program defined that S01~S04 have the same interfaces with M41~M44,UO0~UO3.

- Diagnosis data

0	0	5					S4	S3	S2	S1
interface pins							XS39.8	XS39.14	XS39.1	XS39.5

- Control parameter

State parameter

0	0	1				analog spindle				

Bit4＝1：spindle speed controlled by analog voltage

＝0：spindle speed controlled by on-off variable

1	7	3								SOUS

SOUS= 0：S1,S2,S3,S4 are valid when spindle speed controlled by on-off variable

＝1：only S1,S2 are valid when spindle speed controlled by on-off variable ，S3 and S4 are corresponding to UO2 and UO3.

Note：S1～S4 are invalid when spindle speed controlled by analog voltage，and M41~M44 or UO0~UO3 are valid

- Control logic

when powering on or executing S0，S1～S4 are invalid until executing any one of them，what's more，only one of S1～S4 is valid at one time.

2.8.6 Automatic Gearing Control for Spindle

- Relative signal（defined by the standard PLC program）

M41～M44：output signal for spindle automatic gearing

M41I,M42I：in-position signal of 1,2 gear

M41I has the same port with T07,WQPJ and OV4，and M42I has the same port with T08,NQPJ and OV8.4-gear spindle auto gearing and 2-gear in-position detecting are valid under analog spindle control.

- Diagnosis data

0	0	5					**M44**	**M43**	**M42**	**M41**
interface pins							**XS39.8**	**XS39.14**	**XS39.1**	**XS39.5**

0	**0**	**2**	**M42I**	**M41I**						
interface pins			**XS40.19**	**XS40.20**						

- Signal connection

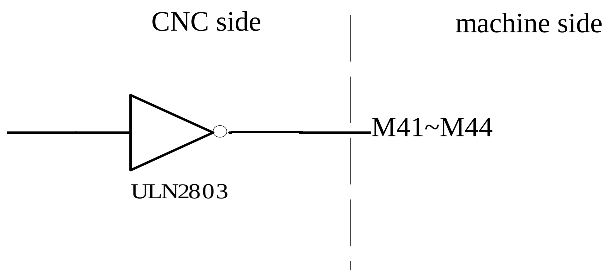


Fig. 2-49

● Control parameter

State parameter

0	0	1					analog spindle				

Bit4＝1：spindle speed controlled by analog voltage

＝0：spindle speed controlled by on-off variable

Bit4 must be 1 under spindle auto gearing control.

1	**6**	**4**	**AGER**	**AGIN**	**AGIM**					

AGER＝1：spindle auto gearing control is valid

＝0：spindle auto gearing control is invalid

AGIN＝1：detect M41I,M42I signals when gearing to 1,2 gears

＝0：not detect M41I,M42I signals when gearing to 1,2 gears

AGIM＝1：M41I,M42I signals are valid when disconnecting with＋24V

＝0：M41I,M42I signals are valid when connecting to＋24V

Data parameter

0	**3**	**7**	**GRMAX1**
0	**3**	**8**	**GRMAX2**
0	**3**	**9**	**GRMAX3**
0	**4**	**0**	**GRMAX4**

GRMAX1 ,GRMAX2, GRMAX3, GRMAX4：the highest speed of gear1,2,3,4 under analog spindle control or the spindle speed for M41, M42, M43, M44 respectively under spindle auto gearing control，the gear 1 speed is in default when powering on or invalid spindle auto gearing control.

0	**6**	**5**	**SFT1TME**

The delay time 1 for auto gearing signal outputting

0	6	6	**SFT2TME**

The delay time 2 for auto gearing signal outputting

0	**6**	**7**	**SFTREV**

The output voltage（0～10000，unit:mv）under auto gearing control

● Function description(defined by the standard PLC program)

2.8.7　Outside Cycle Start and Cycle Stop

● Relative signal（defined by the standard PLC program）

ST：outside cycle start signal，with the same function as CYCLE START key in operation panel

SP：outside cycle stop signal，with the same function as CYCLE STOP key in operation panel and the same port with defence gate detect signal（SAGT）

● Diagnosis data

0	0	1	**SP**	**ST**						
interface pins			**XS40.7**	**XS40.8**						

Book3 connection

● Signal connection

The interface circuit of ST/SP as fig．2-50

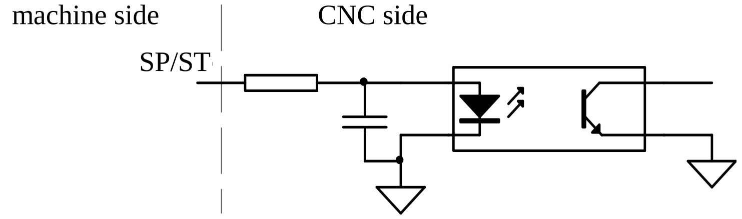


Fig.2-50

● Control parameter

State parameter

1	7	2		**MST**	**MSP**					

MST=1：outside cycle start signal（ST）is invalid，defined by the macro program（#1014）

=0：outside cycle start signal（ST）is valid

MSP＝1：outside cycle stop signal（SP）is invalid，defined by the macro program（＃1015）

＝0：outside cycle stop signal（SP）is valid

● Connection circuit

The connection circuit of ST/SP as fig.2-51

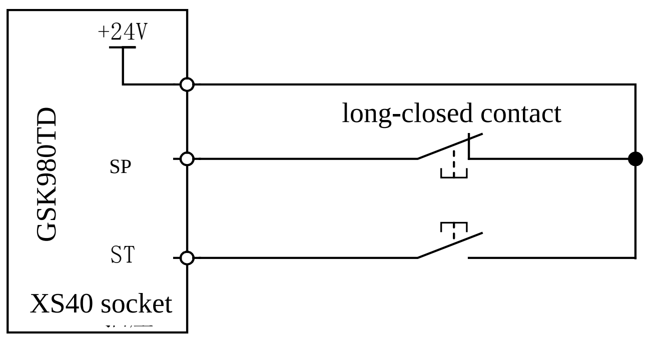


Fig.2-51

2.8.8 Coolant Control

● Relative signal（defined by the standard PLC program）

M08：coolant on

● Diagnosis data

0	0	4					**M08**			
interface pins							**XS39.15**			

● Signal connection

The inside circuit as fig.2-52

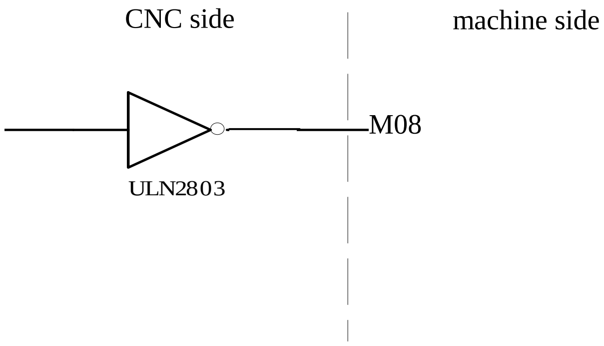


Fig.2-52 the inside circuit of M08

● Function description (defined by the standard PLC program)

When powering on or commanding M09 code，M08 output is invalid and coolant is off until commanding M08 code.

Note1：M08 output being cancelled in the emergency stop

Note2：whether M08 output being cancelled at reset is specified by bit3 of No.009 parameter

Bit3＝0：cancelling M08output at reset

Bit3＝1：not cancelling M08 output at reset

Note3：M09 output signal does not exist

Note4： 

in the operation panel can control coolant on or off，please read the BOOK 2 of manual for details

2.8.9 Lubrication Control

● Relative signal（defined by the standard PLC program）

M32：lubrication on

● Diagnosis data

0	0	4					**M32**				
interface pins							**XS39.6**				

● Signal connection

The inside circuit as fig.2-53

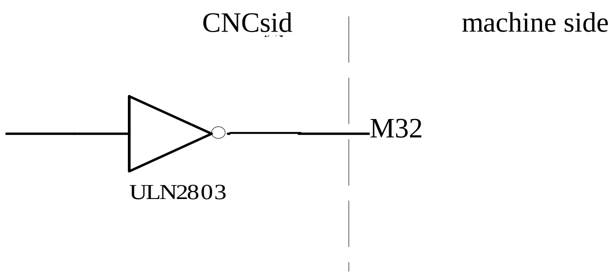
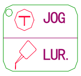


Fig. 2-53 the inside circuit of M32

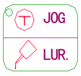
● Control parameter

State parameter

1	7	5	**SPHD**							

SPHD＝1： 

on the operation panel is spindle step function

＝0： 

is lubrication function

Data parameter

1	1	2	

Lubrication open time in non-automation mode（0～65535ms）（0：no time limit）

PLC data

D	T	0	1	**6**	

Lubrication time in automation mode（0～2147483647ms）

D	T	0	1	**7**	

Lubrication interval time in automation mode（0～2147483647ms）

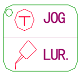
● Function description

There are two mode of automatic and non-automatic lubrication for the standard PLC program，which can be specified by the parameter.

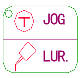
DT17＝0：non-automatic lubrication（same as the former version）

＞0：automatic lubrication

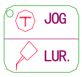
1　Non-automatic lubrication

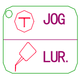
No.175.7 state parameter is 1： 

on the operation panel is spindle step function

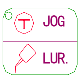
No.175.7 state parameter is 0： 

is lubrication function

When No.112 data parameter is 0，lubrication outputs when press 

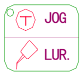
or commanding M32 code，lubrication doesn't output when press 

again or commanding M33 code.

When No.112 data parameter is more than 1，lubrication outputs for the duration time specified by No.112 parameter when press 

or commanding M32 code，lubrication output is cancelled before the time specified by No.112 parameter when commanding M33 code.

2　Automatic lubrication

With the interval time specified by D16，lubrication outputs for the duration time specified by D17 circularly，in which M32,M33 and 

are invalid

2.8.10 Chuck Control

● Relative signal（defined by the standard PLC program）

DIQP：chuck control input signal

DOQPJ：inner chuck clamping/outer chuck unclamping output signal

DOQPS：inner chuck unclamping /outer chuck clamping output signal

NQPJ：in-position input signal for inner chuck clamping/outer chuck unclamping，with the same port as T08,M42I

WQPJ：in-position input signal for inner chuck unclamping/outer chuck clamping，with the same port as T07,M41I

● Diagnosis data

0 0 0		DIQP						
interface pins		XS39.11						

0 0 2	NQPJ	WQPJ						
interface pins	XS40.19	XS40.20						

0 0 4		DOQPJ						
interface pins		XS39.4						

0 0 5			DOQPS					
interface pins			XS39.10					

● Control parameter

Sate parameter

1 6 4							SLSP	SLQP

SLSP＝1：don't detect whether the chuck is clamping when chuck function is valid

＝0：detect whether the chuck is clamping when chuck function is valid

SLQP＝1：chuck function is valid

＝0：chuck function is invalid

1 6 8						PB2		PB1

PB1＝1：outer chuck mode，in which NQPJ is outer chuck unclamping signal and WQPJ is outer chuck clamping signal

＝0：inner chuck mode，in which NQPJ is inner chuck unclamping signal and WQPJ is inner chuck clamping signal

PB2＝1：detect chuck in-position signal，in which bit7 of No.002 diagnosis parameter is NQPJ and bit6 is WQPJ，M41I and M42I are invalid

＝0：don't detect chuck in-position signal

D	T	0	1	8

DT18＞0：chuck clamping and unclamping signals are pulse，width of which is specified by DT18
＝0：chuck clamping and unclamping signals are level

● Signal connection

circuit of DOQPJ/DOQPS as fig.2-54

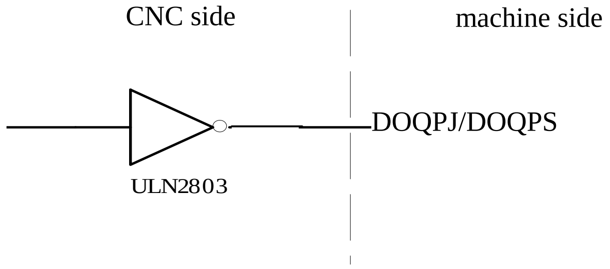


fig.2-54

● Action

① when SLQP is 1,SLSP is 0,PB1 is 0 and PB2 is 1（inner chuck mode and in-position detecting）

DOQPS：chuck unclamping   WQPJ：in-position signal for chuck unclamping

DOQPJ：chuck clamping   NQPJ：in-position signal for chuck clamping

When powering on，DOQPS and DOQPJ are high impedance until DIQP is valid at the first time，at which DOQPJ connects to 0V and the chuck is clamped.

After executing M12，DOQPS is high impedance and DOQPJ is 0V，in which the chuck is clamped and CNC waits for NQPJ.

After executing M13，DOQPJ is high impedance and DOQPS is 0V，in which the chuck is loosened and CNC waits for WQPJ.

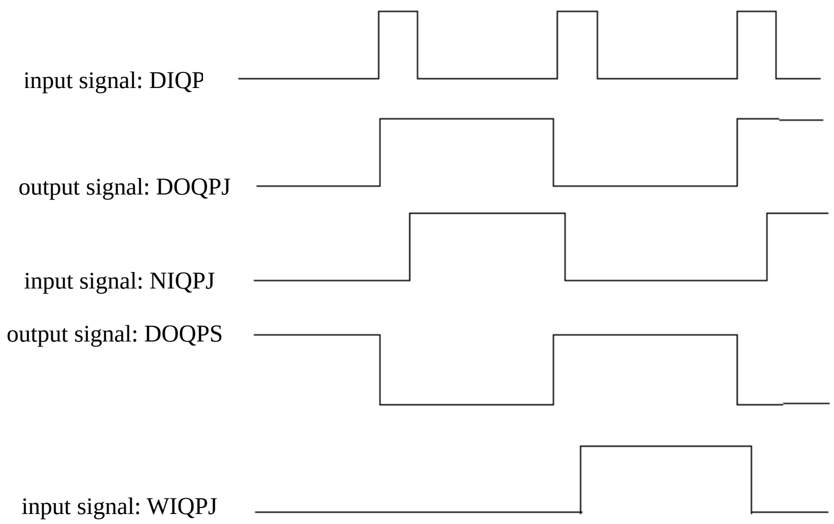


fig.2-55（DT18＝0）

② when SLQP is 1,SLSP is 0,PB1 is 1 and PB2 is 1（outer chuck mode and in-position detecting）

DOQPS：chuck clamping   WQPJ：in-position signal for chuck clamping

DOQPJ：chuck unclamping   NQPJ：in-position signal for chuck unclamping

When powering on，DOQPS and DOQPJ are high impedance until DIQP is valid at the first time，at which

Book3 connection

DOQPS connects to 0V and the chuck is clamped.

After executing M12，DOQPJ is high impedance and DOQPS is 0V，in which the chuck is clamped and CNC waits for WQPJ.

After executing M13， DOQPS is high impedance and DOQPJ is 0V，in which the chuck is loosened and CNC waits for NQPJ.

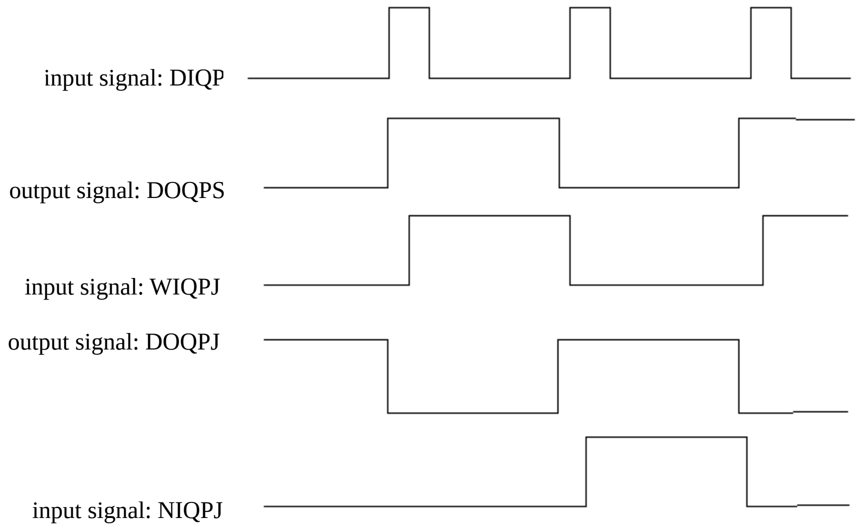


fig.2-56（DT18＝0）

DOQPS connects to 0V when DIQP is valid at the second time，that is to say，DOQPS and DOQPJ connect to 0V alternately with DIQP inputting.

③ Interlocking between chuck and spindle

When SLQP is 1，SLSP is 0，M03 or M04 is valid，alarm will be given after executing M13.

When SLQP is 1，SLSP is 0，PB2 is 0 and executing M12 in MDI or AUTO mode，CNC won't executing the next command until WQPJ or NQPJ is valid. When DIQP is valid in JOG mode，spindle clockwise rotating, spindle counterclockwise rotating are valid until WQPJ or NQPJ is valid. DIQP is invalid during spindle rotating and auto machining cycle and DOQPS,DOQPJ is kept at reset or in emergency stop.

2.8.11 Tailstock Control

- Relative signal（defined by the standard PLC program）

  DOTWJ：tailstock advancing outputting signal

  DOTWS：tailstock withdrawing outputting signal

  DITW：tailstock control input signal

- Diagnosis data

0	0	0				DITW				
interface pins						XS40.2				

0	0	5				DOTWS				
interface pins						XS39.9				

- Control parameter

  State parameter

1	6	4						SLTW		

SLTW＝1：tailstock control function is valid

＝0：tailstock control function is invalid

- Signal connection

Circuit of DOTWJ/ DOTWS as fig.2-57

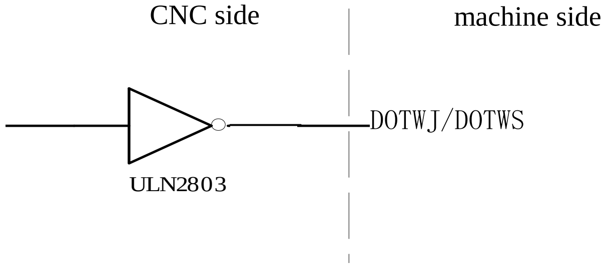


Fig. 2-57

- Action（defined by the standard PLC program）

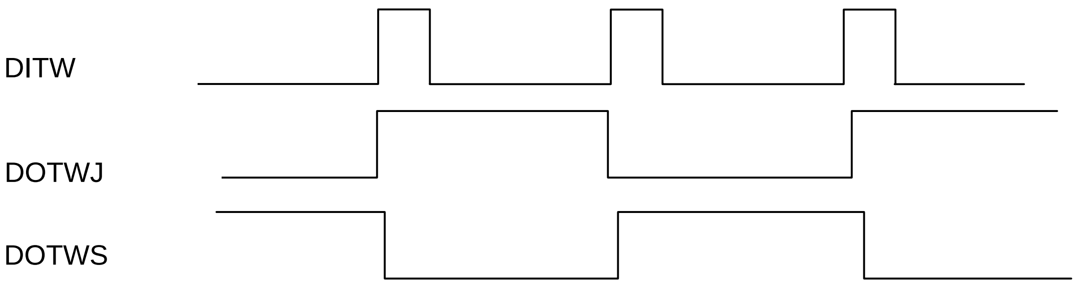


fig.2-58 tailstock control logic

DOTWJ and DOTWS are high impedance when powering on. DOTWJ connects to 0V（valid state） when DITW is valid at the first time or executing M10，and DOTWS connects to 0V（valid state） when DITW is valid at the second time or executing M11，that is to say，DOTWJ and DOTWS connect to 0 V alternately.

During the spindle rotating，alarm will be given after executing M11 and DITW is invalid，and DOTWS，DOTWJ are kept at reset or in emergency stop.

2.8.12 Low Pressure Detection

- Relative signal（defined by the standard PLC program）

PRES：low pressure detection signal，with the same port as TCP

- Diagnosis data

0	0	5	PB3							
interface pins			XS39.12							

- Control parameter

sate parameter

1	6	8			SPB3	PB3				

PB3＝0：low pressure detection function is invalid，bit7 of No.000 diagnosis data is TCP

＝1：low pressure detection function is valid，bit7 of No.000 diagnosis data is PRES

SPB3＝0：low pressure is alarmed when PRES connecting to ＋24V

＝1：low pressure is alarmed when PRES connecting to 0V

data parameter

0	6	9	0	0	0	0	1	0	0	0

Alarm waiting time for low pressure detection

- Function description

① when PB3 is 1 and SPB3 is 0，low pressure is alarmed when PRES connecting to 24V.

② when PB3 is 1 and SPB3 is 1，low pressure is alarmed when PRES connecting to 0V.

③ when PB3 is 1，No.14 alarm will be given when holding time of valid PRES signal exceeding the value specified by No.069 parameter，at which feed is paused, spindle is stopped and cycle is cancelled，until pressing “RESET” key or powering off.

④ When PB3 is 1，TCP is invalid.

2.8.13 Protection Gate Detection

- Relative signal (defined by the standard PLC program)

SAGT：protection gate detection signal，with the same port as SP

- Diagnosis data

0	0	1		PB4							
interface pins				XS40.7							

- Control parameter

state parameter

1	6	8		SPB4	PB4						

PB4＝0：protection gate detection function is invalid

＝1：protection gate detection function is valid and SP is invalid

SPB4＝0：protection gate is closed when SAGT connecting to 0V

＝1：protection gate is closed when SAGT connecting to ＋24V

- Function description

① When PB4 is 1and SPB4 is 0，protection gate is closed when SAGT connecting to 0V

② When PB4 is 1and SPB4 is 1，protection gate is closed when SAGT connecting to ＋24V

③ Alarms will be given at cycle start when protection gate being open

④ Feed being paused, spindle being stopped, coolant being closed and CNC alarm will appear when protection gate is open in auto machining cycle

⑤ Protection gate detection function is valid only in AUTO mode

⑥ SAGT and SP have the same port，that is to say，MSP（bit5 of No.172 state parameter）should be 1 when PB4 is 1 and MSP should be 0 when PB4 is 0

2.8.14 Spindle Rotation Permission

- Relative signal (defined by the standard PLC program)

SPEN：spindle rotation permission signal，with the same port as T05

- Diagnosis data

0	0	2					SPEN				
interface pins							XS40.22				

● Control parameter

state parameter

1	6	4					SPEN			

SPEN＝0：spindle rotation permission signal is invalid

＝1：spindle rotation permission signal is valid

● Control logic (defined by the standard program)

When SPEN parameter is 1 and SPEN signal isn't received，alarms will be given after executing M03 or M04

2.8.15 Program Segment Skipping

Program segment skipping function is selected when one segment is embarrassed，in which the segment with "/" at head will be skipped and not be executed when PRG SEG SWITCH is on or EDT signal is valid.

● Relative signal (defined by the standard PLC program)

BDT：program segment skipping signal，with the same port as DITW

● Diagnosis data

0	0	0				BDT				
interface pins						XS40.2				

● Control parameter

state parameter

0	6	4						SLTW		

SLTW＝0：program segment skipping function is invalid

＝1：program segment skipping function is valid and DITW is invalid

● Signal connection

circuit of BDT/DITW as fig.2-59

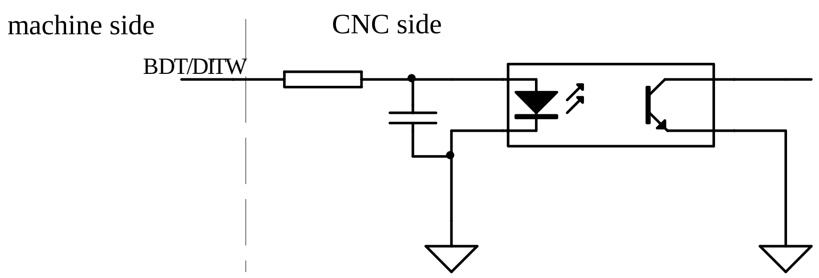


Fig.2-58

● Function description

The segment with "/" at head will be skipped and not be executed when PRG SEG SWITCH in machine panel is on or EDT signal is valid.

2.8.16 Macro Variable

● Relative signal

macro output signal：UO0～UO5 are valid when spindle speed being controlled by analog voltage and spindle auto gearing being invalid.

macro input signal：UI0～UI15 are valid when the signals with the same ports as UI0～UI15 are invalid.

● Diagnosis data

0	0	5			UO05	UO04	UO03	UO02	UO01	UO00
interfaces pins					XS39.10	XS39.9	XS39.8	XS39.14	XS39.1	XS39.5
variable number					#1105	#1104	#1103	#1102	#1101	#1100

0	0	0	UI07	UI06	UI05	UI04	UI03	UI02	UI01	UI00
interfaces pins			XS39.12	XS39.11	XS40.1	XS40.2	XS40.3	XS40.4	XS40.5	XS40.6
variable number			#1007	#1006	#1005	#1004	#1003	#1002	#1001	#1000

0	0	0	UI15	UI14	UI13	UI12	UI11	UI10	UI09	UI08
interfaces pins			XS40.7	XS40.8	XS40.9	XS40.10	XS40.19	XS40.20	XS40.21	XS40.22
variable number			#1015	#1014	#1013	#1012	#1011	#1010	#1009	#1008

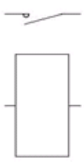
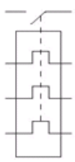
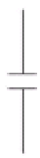
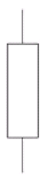
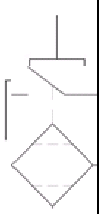
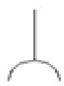
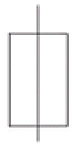
● Function description（defined by the standard PLC program）

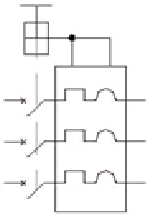
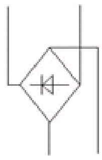
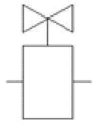
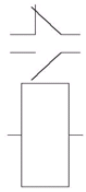
State of UO0～UO5 can be changed by evaluating ＃1100～＃1105，in which UO0～UO5 are 0V when ＃1100～＃1105 are "1" and UO0～UO5 are closed when ＃1100～＃1105 are "0".

A variety of disposal can be carried out by evaluating ＃1000～＃1015,detecting UI0～UI15 and using the transfer commands.

2.9 ELECTRIC CONNECTION for I/O SIGNAL

DC24V power of GSK980TD must be separate from that of the electromagnetism valve with large current，the component symbol as following：

graph									
symbol	KM	FR	C	R		SQ		SA	FV
name	contactor coil and auxiliary contact	fuse relay and contact	capacity	register	HALL component	journey switch	veneer jack	kick switch	Fuse

name	symbol	graph
air breaker	QF	
transformer	TC	
bridge rectifier	VC	
motor	M	
diode	VD	
Electromagnetic valve coil	YV	
relay coil and contact	KA	

The following electric connection diagrams are only for reference：

● Tailstock connection

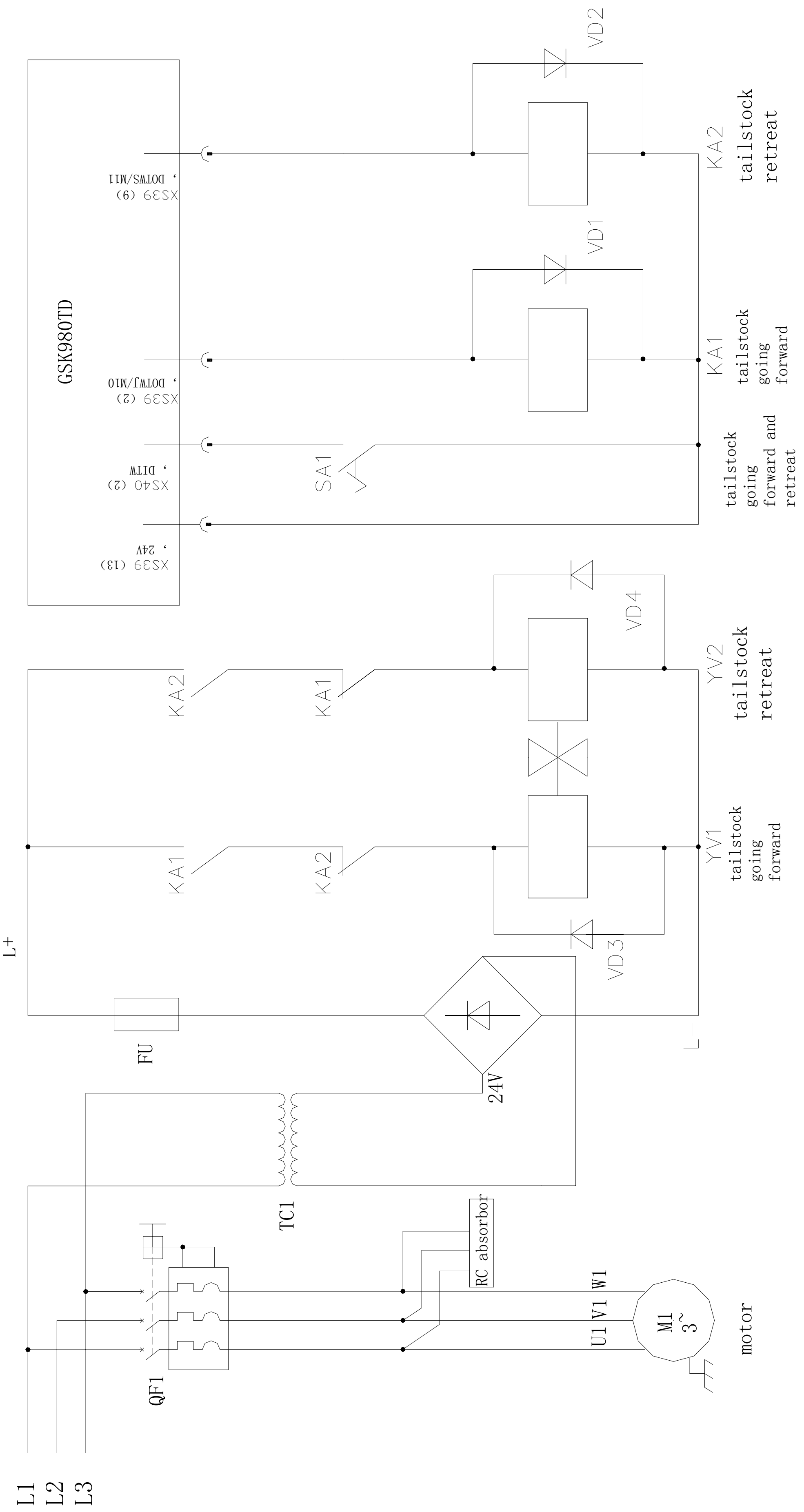


fig.2-61 tailstock connection

● Chuck connection

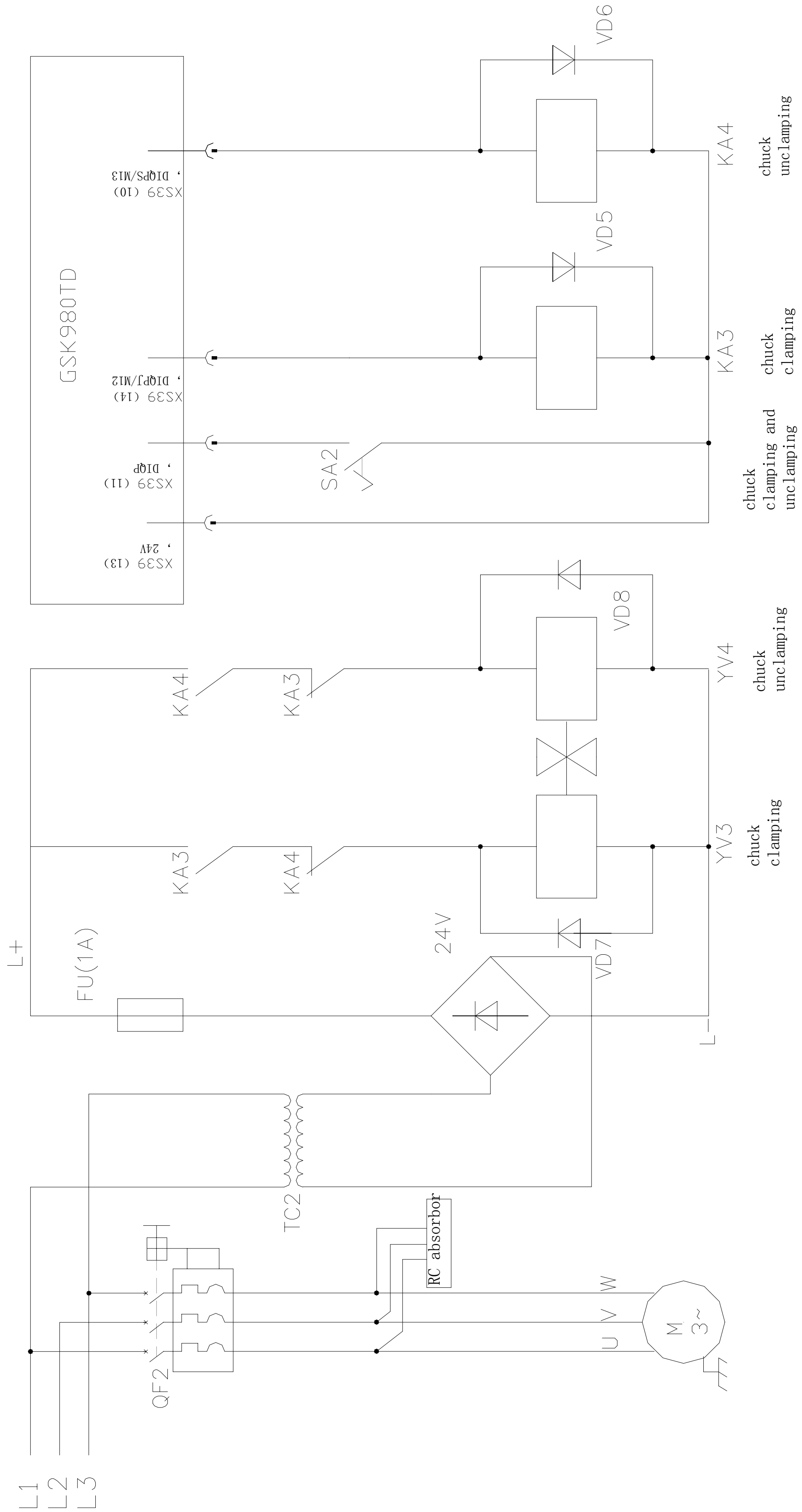


fig.2-62 chuck connection

Book3 connection

- Toolpost connection

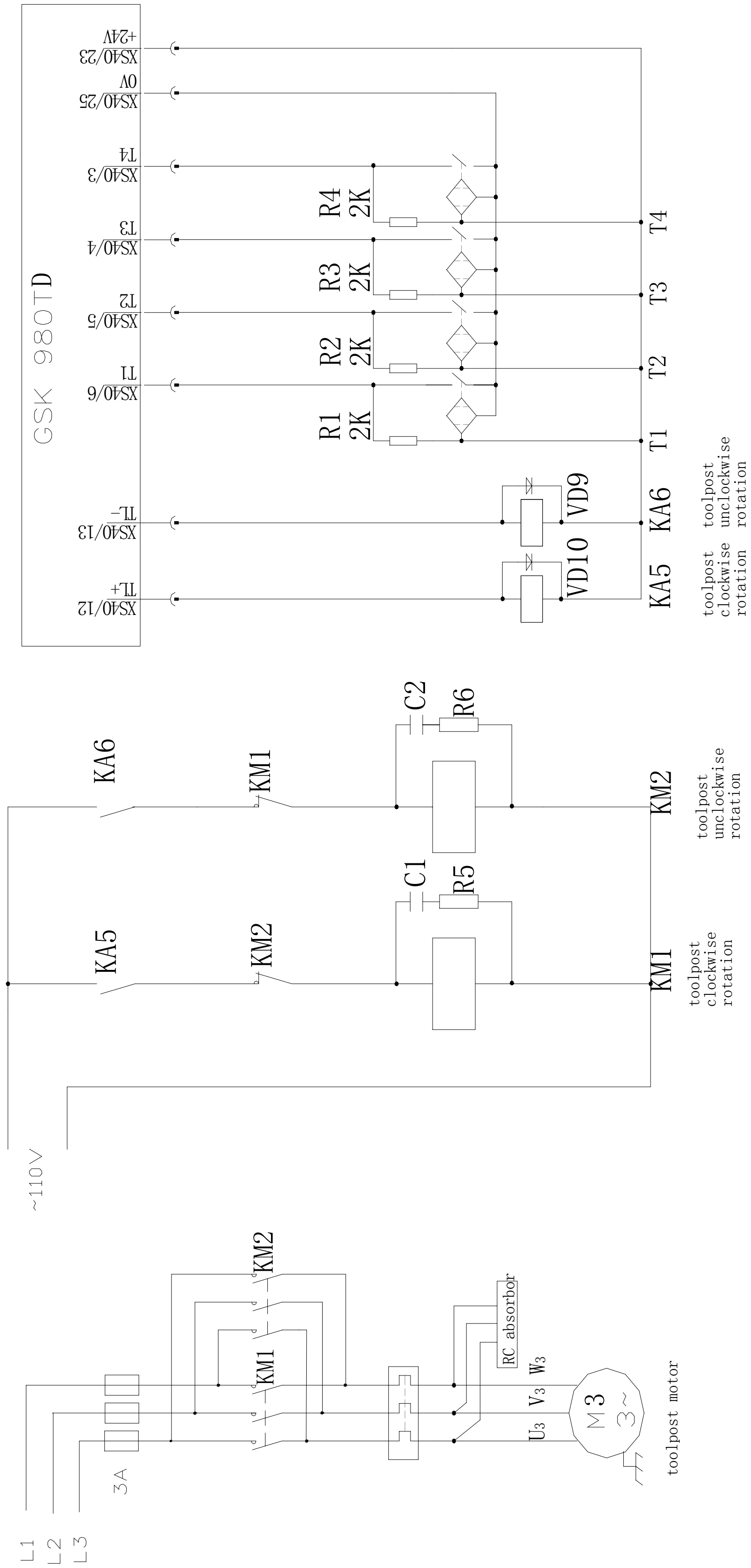


Fig.2-63 toolpost connection

● Connection of spindle auto gear shifting

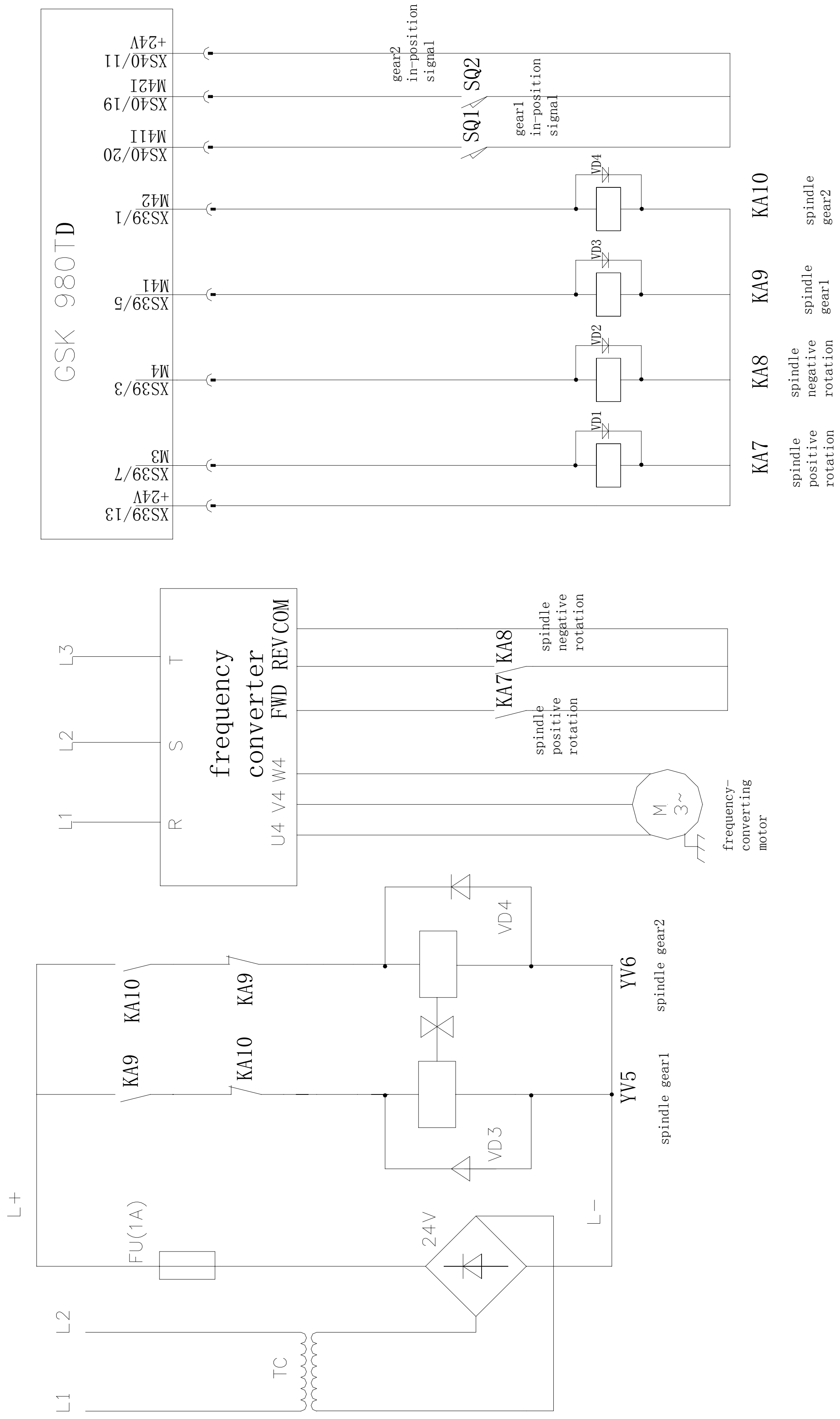
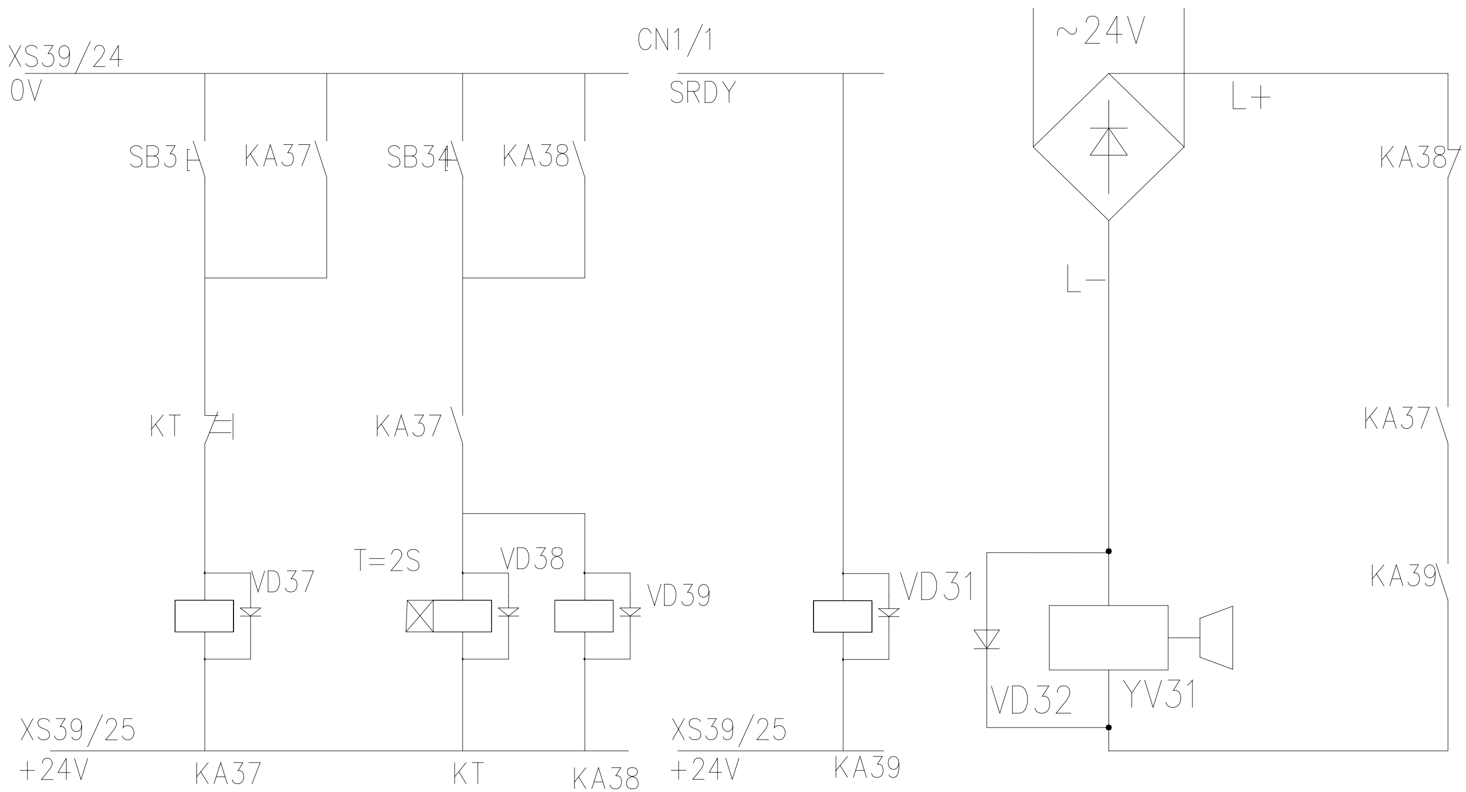


Fig.2-64 connection of spindle auto gear shifting

● Connection of motor brake



note：VD31 and VD32 are 1N4007

Fig 2-65 Connection of motor brake

CHAPTER3 PARAMETER SPECIFICATION

In this chapter, state parameters and data parameters of CNC will be introduced.

3.1 PARAMETER SPECIFICATION（with sequence）

3.1.1 State Parameter

The show mode of state parameter as following:

Number	BIT7	BIT6	BIT5	BIT4	BIT3	BIT2	BIT1	BIT0

0 0 1	***	***	***	SPIN	MPG	PROG	***	***

Bit2 is 1: programming in radius
is 0: programming in diameter
Bit3 is 1: MPG（manual pulse generator）mode

is 0: STEP mode
Bit4 is 1: analog spindle
is 0: switching control spindle

0 0 2	***	***	RS232	***	***	***	TNRC	***

Bit1 is 1: tool nose radius compensation is valid
is 0: tool nose radius compensation is invalid
Bit5 is 1: RS232 communication is valid
is 0: RS232 communication is invalid

0 0 3	***	***	PEC	TLC	***	***	***	***

Bit4 is 1: compensate tool length by coordinates
is 0: compensate tool length by tool moving
Bit5 is 1: pitch error compensation is valid
is 0: pitch error compensation is invalid

0 0 4	***	RDRN	DECI	ORC	TOC	DCS	PROD	***

Bit6 is 1: G00 in dry run at rapid traverse speed
is 0: G00 in dry run at manual feedrate
Bit5 is 1: decelerating signal in high level is valid during reference point return
is 0: decelerating signal in low level is valid during reference point return
Bit4 is 1: tool compensation is radius value
is 0: tool compensation is diameter value
Bit3 is 1: keep tool compensation at reset
is 0: clear tool compensation at reset

Book3 connection

Bit2 is 1： 

key can start program in MDI mode

is 0： 

key can’t start program in MDI mode

Bit1 is 1：relative coordinate display don’t include tool compensation

is 0：relative coordinate display includes tool compensation

0	0	5		***	***	SMAL	M30	***	***	PPD	PCMD

Bit5 is 1：manual gear shifting as executing S

is 0：automatic gear shifting as executing S

Bit4 is 1：cursor returns after M30

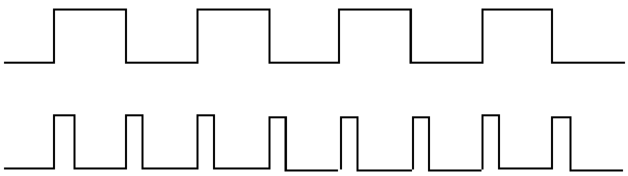
is 0：cursor doesn’t return after M30

Bit1 is 1：relative coordinate is set by G50

is 0：relative coordinate isn’t set by G50

Bit0 is 1：output CP signal with square wave

is 0：output CP signal with pulse



0	0	6		***	***	***	OVRI	***	***	ZMZ	ZMX

Bit4 is 1： feedrate override in machine control panel is reversed

is 0：feedrate override in machine control panel isn’t reversed

Bit1 is 1：reference point return mode C for Z axis

is 0：reference point return mode B for Z axis

Bit0 is 1：reference point return mode C for X axis

is 0：reference point return mode B for X axis

0	0	7		***	***	***	***	SMZ	***	ZCZ	ZCX

Bit3 is 1：execute next program segment after locating accurately according to the current segment

is 0：there is smoothness transition between two program segments

Bit1 is 1：ZDEC connects to ZPC in parallel

is 0：ZDEC connects to ZPC independently

Bit1 is 1：XDEC connects to XPC in parallel

is 0：XDEC connects to XPC independently

0	0	8		***	***	***	***	***	***	DIRZ	DIRX

Bit1 is 1：DIRZ signal is high level when moving in positive direction

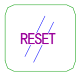
is 0：DIRZ signal is low level when moving in negative direction

Bit0 is 1：DIRX signal is high level when moving in positive direction

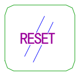
is 0：DIRX signal is low level when moving in negative direction

0	0	9

***	***	***	***	RSJG	***	ZALM	XALM

Bit3 is 1：close spindle，coolant and lubrication when pressing 

key

is 0：keep spindle，coolant and lubrication when pressing 

key

Bit1 is 1：ZALM signal at low level is valid

is 0：ZALM signal at high level is valid

Bit0 is 1：XALM signal at low level is valid

is 0：XALM signal at high level is valid

0	1	0

***	***	***	***	CPF4	CPF3	CPF2	CPF1

Bit0～bit3：Frequency setting for backlash compensation（with BCD）

Frequency ＝（setting value＋1）Kpps

CPF4	CPF3	CPF2	CPF1	Setting value（Kpps）
0	0	0	0	1
0	0	0	1	2
0	0	1	0	3
0	0	1	1	4
0	1	0	0	5
0	1	0	1	6
0	1	1	0	7
0	1	1	1	8
1	0	0	0	9
1	0	0	1	10
1	0	1	0	11
1	0	1	1	12
1	1	0	0	13
1	1	0	1	14
1	1	1	0	15
1	1	1	1	16

0	1	1

BDEC	BD8	***	***	***	ZNIK	TSGN	***

Bit7 is 1：backlash compensation mode B，with deceleration/acceleration for compensation and setting frequency is invalid

is 0：backlash compensation mode A，with fixed frequency for compensation

Bit6 is 1：backlash compensation with one setting frequency

is 0：backlash compensation with eighth setting frequency

Bit2 is 1：keep moving once pressing direction key in reference point return

is 0：stop moving once releasing direction key in reference point return

Bit1 is 1：T01~T08 signals at low level are valid

is 0：T01~T08 signals at high level are valid

0	1	2

APRS	WSFT	DOFSI	***	EAL	***	EBCL	ISOT

Bit7 is 1：set absolute coordinate-system specified by No.49 and No.50 parameter automatically after reference point return

is 0：don’t set absolute coordinate-system automatically after reference point return

Bit6 is 1：workpiece coordinate-system offset specified by No.000 tool offset is valid
is 0：workpiece coordinate-system offset is invalid
Bit5 is 1：toolsetting with trail cutting is valid
is 0：toolsetting with trail cutting is invalid
Bit3 is 1：program can be edited at alarm
is 0：program can't be edited at alarm
Bit1 is 1：end symbol EOB in part programs is with ;
is 0：end symbol EOB in part programs is with *
Bit0 is 1：rapid traverse in JOG mode is valid when powering on or before reference point return
is 1：rapid traverse in JOG mode is invalid when powering on or before reference point return

0	1	4	***	***	***	***	***	***	ZRSZ	ZRSX

Bit1 and bit0 are 1：PC signal and DEC signal are needed in reference point return
are 0：PC signal and DEC signal aren't needed in reference point return

1	6	4	AGER	AGIN	AGIM	***	SPEN	SLTW	SLSP	SLQP

Bit7 is 1：automatic spindle gear shifting is valid
is 0：automatic spindle gear shifting is invalid
Bit6 is 1：detect M41I and M42I signals at automatic spindle gear1 and gear2
is 0：don't detect M41I and M42I signals at automatic spindle gear1 and gear2
Bit5 is 1：M41I and M42I signals are valid when disconnecting with ＋24V
is 0：M41I and M42I signals are valid when connecting to ＋24V
Bit3 is 1：spindle rotation permitting input is valid
is 0：spindle rotation permitting input is invalid
Bit2 is 1：tailstock control is valid
is 0：tailstock control is invalid
Bit1 is 1：don't detect whether spindle is clamped under chuck control
is 0： detect whether spindle is clamped under chuck control，otherwise，spindle can't be started
Bit0 is 1：chuck control is valid
is 0：chuck control is invalid

1	6	8	SPB4	PB4	SPB3	PB3	***	PB2	***	PB1

Bit7 is 0：safeguard is closed when SGAT signal connecting to 0V
is 1：safeguard is closed when SGAT signal connecting to ＋24V
Bit6 is 0：safeguard is invalid
is 1：safeguard is valid and *SP signal is invalid
Bit5 is 0：low pressure alarms when PRES signal connecting to 0V
is 1：low pressure alarms when PRES signal connecting to ＋24V
Bit4 is 0：low pressure detection is invalid
is 1：low pressure detection is valid
Bit2 is 0：don't detect in-position signal of chuck
is 1：detect in-position signal of chuck
Bit0 is 0：internal chuck，NQPJ is clamping signal and WQPJ is unclamping signal for internal chuck
is 1：external chuck，NQPJ is unclamping signal and WQPJ is clamping signal for external chuck

1	7	2	***	MST	MSP	MOT	ESP	***	***	SOVI

Bit6 is 0：ST signal is valid

is 1：ST signal is invalid，which can be defined by macro program（＃1014）

Bit5 is 0：SP signal is valid

is 1：SP signal is invalid，which can be defined by macro program（＃1015）

Bit4 is 0：check software overtravel

is 1：don’t check software overtravel

Bit3 is 0：emergency stop is valid

is 1：emergency stop is invalid

Bit0 is 1：T05～T08 signals are valid

is 0：T05～T08 signals are invalid，and *0V1,*0V2,*0V4,*0V8 signals are valid

1	7	3		***	***	***	***	***	***	SINC	SOUS

Bit1 is 0：the increment of 0.001,0.01,0.1 is valid in STEP/HW mode.

is 1：the increment of 0.001,0.01 is valid in STEP/HW mode.

Note1：to avoid lost step，please specify this bit to 1 when matching step drivers

Bit0 is 0：S1,S2,S3,S4 are valid when analog spindle is invalid

is 1：S1,S2 are valid and S3,S4 are invalid when analog spindle is invalid

1	7	4		***	***	***	MDOK	KEY1	MD4	MD2	MD1

Bit4 is 0：keep operation mode when powering off

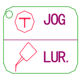
is 1：operation mode is specified by MD4,MD2,MD1 when powering on

MD4	MD2	MD1	Operation mode
0	0	0	MDI
0	0	1	AUTO
0	1	0	REF
0	1	1	EDIT
1	0	0	HW
1	0	1	JOG

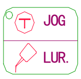
Bit3 is 0：program switch is on when powering on

is 1：program switch is off when powering on

1	7	5		SPHD	SAR	***	THDA	SPFD	***	ZVAL	XVAL

Bit7 is 1：

key serves as spindle jogging

is 0：

key serves as lubrication

Bit6 is 1：check SAR signal before machining

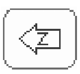
is 0：don’t check SAR signal before machining

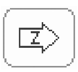
Bit4 is 1：exponential deceleration/acceleration for thread machining

is 0：linear deceleration/acceleration for thread machining

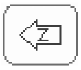
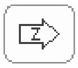
Bit3 is 1： cutting is stopped when spindle stopping during machining

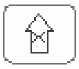
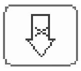
is 0： cutting isn’t stopped when spindle stopping during machining

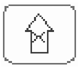
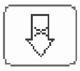
Bit1 is 1：

key is positive direction and 

key is negative direction

is 0：  key is negative direction and  key is positive direction

Bit0 is 1：  key is positive direction and  key is negative direction

is 0：  key is negative direction and  key is positive direction

1	8	0		***	***	***	***	***	***	***	SPOS

Bit0 is 1： remaining coordinates in POS and PROG windows

is 0： incremental coordinates in POS and PROG windows

1	8	2		***	***	***	***	***	PB6	***	PB5

Bit2 is 1： check tool signal at the end of tool change

is 0： don’t check tool signal at the end of tool change

Bit0 is 1： tool change mode A

is 0： tool change mode B

1	8	3		***	***	***	***	***	***	MZRZ	MARX

Bit1 is 1： return reference point by pressing the negative direction key for Z axis

is 0： return reference point by pressing the positive direction key for Z axis

Bit0 is 1： return reference point by pressing the negative direction key for X axis

is 0： return reference point by pressing the positive direction key for X axis

1	8	4		***	PTEST	***	***	***	***	***	LANG

Bit6 is 1： automatically testing ports is valid（power on over again）

is 0： automatically testing ports is invalid

Bit0 is 1： English display interface

is 0： Chinese display interface

3.1.2 Data Parameter

0	1	5		CMRX
0	1	6		CMRZ

CMRX and CMRZ are instruction multipliers，the range of which is from 1 to 255.

0	1	7		CMDX
0	1	8		CMDZ

CMDX and CMDZ are instruction denominators，the range of which is from 1 to 255.

Formula：

$$\frac{CMR}{CMD} = \frac{\delta \times 360}{\alpha \times L} \times \frac{Z_M}{Z_D}$$

α： driver rotation angle for receiving one pulse

L： screw leader

δ： the minimum input unit of CNC（0.001mm for X，0.05mm for Z of GSK980TD）

Z_M： gear number from screw side

Z_D：gear number from machine side

0	1	9	THDCH

THDCH is length of thread run-out，the range of which is from 1 to 255.
Width of thread run-out＝THDCH×0.1×screw leader

0	2	1	VCV

It's voltage compensation value when spindle speed instruction is 10V，the range of which is from –2000 to 2000（unit is mV）

0	2	2	RPDFX

RPDFX is the max. rapid traverse speed for X axis（radius value），the range of which is from 10 to 3825000（unit is mV/min）

0	2	3	RPDFZ

RPDFZ is the max. rapid traverse speed for Z axis，the range of which is from 10 to 7650000（unit is mV/min）

0	2	4	LINTX
0	2	5	LINTZ

LINTX and LINTZ are linear acceleration/deceleration time constants for X and Z axes in rapid traverse，the range of which is from 0 to 4000（unit is mS）

0	2	6	THRDT

THRDT is acceleration/deceleration time constant for the short axis in thread run-out，the range of which is from 0 to 4000（unit is mS）

0	2	7	FEDMX

It's upper limit feedrate for X and Z axes，the range of which is from 10 to 8000（unit is mm/min）

0	2	8	THDFL

It's initial speed for X and Z axis in thread cutting，the range of which is from 6 to 8000（unit is mm/min）

0	2	9	FEEDT

It's exponential acceleration/deceleration time constant for X and Z axes in cutting and manual feed，the range of which is from 0 to 4000（unit is mS）

0	3	0	FEDFL

It's start speed in acceleration and end speed in deceleration in cutting，the range of which is from 0 to 8000（unit is mm/min）

0	3	2	RPDFL

It's rapid traverse speed when rapid traverse override is F0，the range of which is from 6 to 4000（unit is mm/min）

0	3	3	ZRNFL

It's low speed for X and Z axes in reference point return，the range of which is from 6 to 4000（unit is mm/min）

0	3	4	BKLX

It's backlash compensation value of X axis，the range of which is from 0 to 2000（unit is 0.001mm）

0	3	5	BKLZ

It's backlash compensation value of Z axis，the range of which is from 0 to 2000（unit is 0.001mm）

0	3	6	SPDLC

SPDLC is voltage compensation value when SVC is 10V，the range of which is from –1000 to 1000（unit is mV）

0	3	7	GRMAX1
0	3	8	GRMAX2
0	3	9	GRMAX3
0	4	0	GRMAX4

GRMAX1 ,GRMAX2, GRMAX3, GRMAX4 are the max. spindle speeds in gear 1 to 4 when SVC is 10V or for instruction M41 to M44 when automatic gear shifting is valid. Speed of gear1 is default value when powering on or automatic gear shifting is invalid.

The range is from 10 to 9999（unit is rpm）.

0	4	1	JOGFL

JOGFL is start speed of exponential acceleration and end speed of deceleration in JOG，the range of which is from 0 to 8000（unit is mm/min）.

0	4	2	SEQINC

SEQINC is increment value of block sequence number，the range of which is from 1 to 100.

0	4	3	LOWSP

LOWSP is the min. spindle speed in G96，the range of which is from 0 to 9999(unit is rpm).

0	4	4	BRATE0

BRATE0 is serial communication rate，which is valid when bit5 of No.2 parameter is 1.The range is 1200,2400,4800,9600,19200,38400 57600 115200 (unit is bit/s)

0	4	5	LT1X1
0	4	6	LT1Z1
0	4	7	LT1X2
0	4	8	LT1Z2

LT1X1 and LT1Z1 are software overtravel in positive direction for X and Z axes separately，LT1X2 and LT1Z2 are software overtravel in negative direction for X and Z axes separately.The range is from 0 to ±9999999（unit is mm）.

Note：Value type of LT1X1 and LT1X2 is specified by bit2 of No.1 parameter.

0	5	1	MRCCD

Book3 connection

MRCCD is each infeed value in roughing(G71,G72)，which can also be specified in program instruction. The range is from 1 to 99999（unit is 0.001mm）.

0	5	2	MRCDT

MRCDT is each retraction value in roughing(G71,G72)，which can also be specified in program instruction. The range is from 1 to 99999（unit is 0.001mm）.

0	5	3	PECSCX

PECSCX is retraction value in roughing of X axis in G73，which can also be specified in program instruction. The range is from -99999 to 99999（unit is 0.001mm）.

0	5	4	PECSCZ

PECSCZ is retraction value in roughing of Z axis in G73，which can also be specified in program instruction. The range is from -99999 to 99999（unit is 0.001mm）.

0	5	5	PATIM

PATIM is cutting times of G73，which can also be specified in program instruction. The range is from 1 to 99999（unit is times）.

0	5	6	GROVE

GROVE is retraction value of Z axis in G74 or X axis in G75，which can also be specified in program instruction. The range is from 0 to 99999（unit is 0.001mm）.

0	5	7	THRPT

THRPT is finishing times in G76，which can also be specified in program instruction. The range is from 1 to 99（unit is times）.

0	5	8	THANG

TFANG is tool angle in G76，which can also be specified in program instruction. The range is from 0 to 99（unit is degree）.

0	5	9	THCLM

THCLM is least cutting depth in G76，which can also be specified in program instruction. The range is from 0 to 99999（unit is 0.001mm）.

0	6	0	THDFN

THDFN is remaining cutting depth of finishing in G76，which can also be specified in program instruction. The range is from 0 to 99999（unit is 0.001mm）.

0	6	5	SFT1TME

SFT1IME is gear shifting time 1 of spindle，the range is from 0 to 60000（unit is ms）.

0	6	6	SFT2TME

SFT2IME is gear shifting time 2 of spindle，the range is from 0 to 60000（unit is ms）.

0	6	7	SFTREV

SFTREV is spindle voltage as gear shifting，the range is from 0 to 10000（unit is mV）.

0	6	9	PEALMTIM

PEALTIM is low pressure alarm time，the range is from 0 to 60000（unit is ms）.

0	7	0	ENCODER_CNT

ENCODER_CNT is spindle encoder line number，the range is 100 to 5000（unit is lpr）.

0	7	1	RESET_TIME

RESET_TIME is output time for reset signal，the range is from 16 to 4080（unit is ms）.

0	7	2	SAR_DELEY

SAR_DELEY is delay time of detecting SAR signal，the range is form 0 to 4080（unit is ms）.

0	7	6	T1MAXT

T1MAXT is upper limit time for changing one tool，the range is form 100 to 5000（unit is ms）.

0	7	8	TLMAXT

TLMAXT is upper limit time for changing total tools，the range is form 1000 to 60000（unit is ms）.

0	8	0	MTIME

MTIME is duration for M code being executed，the range is form 100 to 5000（unit is ms）.

0	8	1	STIME

STIME is duration for S code being executed，the range is form 100 to 5000（unit is ms）.

0	8	2	T1TIME

T1TIME is delay time from TL+ ceased to TL- output，the range is from 0 to 4000（unit is ms）.

0	8	3	TCPWRN

TCPWRN is alarm time for not receiving *TCP signal，the range is from 0 to 4000（unit is ms）.

0	8	4	TMAX

TMAX is total tool number selection，the range is from 1 to 32.

0	8	5	TCPTIME

TCPTIME is time .from TL- output to receive TCP，the range is from 0 to 4000（unit is ms）.

0	8	7	SPDDLT

SPDDLT is delay time for M05 and SPZD outputting，the range is from 0 to 10000（unit is ms）.

0	8	9	SPZDTIME

SPZDTIME is SPZD output time，the range is from 0 to 60000（unit is ms）.

0	9	8	

It's position number of pitch error compensation for X machine reference point, the range is from 0 to 255.

0	9	9	

It's position number of pitch error compensation for Z machine reference point, the range is from 0 to 255.

1	0	2	

It's distance between each pitch error compensation of X axis, the range is from 10000 to 999999 (unit is ms).

1	0	3	

It's distance between each pitch error compensation of Z axis, the range is from 10000 to 999999 (unit is ms).

1	0	6	THD_SPD_VAR

THD_SPD_VAR is spindle speed fluctuating limit in thread cutting, the range is from 0 to 100 (unit is%).

1	0	7	THD_TAIL_SPD

THD_TAIL_SPD is speed of short axis in thread tailing, the range is from 0 to 8000 (unit mm/min).

1	0	8	SPL_REV_TIME

SPL_REV_TIME is spindle jogging time, the range is from 0 to 60000 (unit is ms).

1	0	9	SPL_REV_SPD

SPL_REV_SPD is spindle jogging speed, the range is from 1 to 8000 (unit is rpm).

1	1	0	MGR

MGR is spindle gear teeth number in driving ratio, the range is from 1 to 255.

1	1	1	SGR

SGR is encoder gear teeth number in driving ratio, the range is from 1 to 255.

1	1	2	LUBRICATE_TIME

LUBRICATE_TIME is lubricating time, the range is from 0 to 60000 (unit is ms).

1	1	3	REF_SPD

REF_SPD is rapid traverse speed of each axis in reference point return, the range is from 10 to 7650000 (unit is mm/min).

1	1	4	

It's offset in X machine reference point, the range is from –99999 to 99999 (unit is 0.001mm).

1	1	5	

It's offset in Z machine reference point, the range is from –99999 to 99999 (unit is 0.001mm).

Book3 connection

1	1	9	

It's valid key number, which can be pressed simultaneously. The range is from 2 to 5.

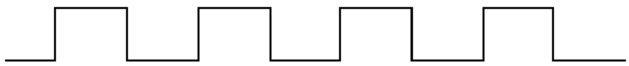
3.2 PARAMETER SPECIFICATION（with function）

3.2.1 Control Logic for X and Z Axes

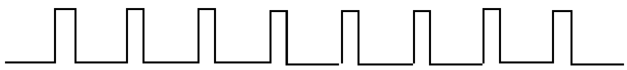
0	0	5									PCMD

PCMD is 1: output CP signal with square wave

is 0: output CP signal with pulse



square wave, the max. frequency is 266KHz



pulse, the max. frequency is 266KHz and pulse width is 1μs.

0	0	8								DIRZ	DIRX

DIRZ is 1: DIRZ signal is high level when moving in positive direction

is 0: DIRZ signal is low level when moving in negative direction

DIRX is 1: DIRX signal is high level when moving in positive direction

is 0: DIRX signal is low level when moving in negative direction

0	0	9								ZALM	XALM

ZALM is 1: ZALM signal at low level is valid

is 0: ZALM signal at high level is valid

XALM is 1: XALM signal at low level is valid

is 0: XALM signal at high level is valid

1	7	5					SPFD				

SPFD is 1: cutting is stopped when spindle stopping during machining

is 0: cutting isn't stopped when spindle stopping during machining

3.2.2 Acceleration and Deceleration Control

0	2	2	RPDFX
0	2	3	RPDFZ

RPDFX and RPDFZ are the max. rapid traverse speed（radius value）for X and Z axes, the range is from 10 to 3825000 for X axis and is from 10 to 7650000 for Z axis（unit is mm/min）.

Generally, RPDFX is recommended to be 3800 for servo motor and be 3000 for step motor, then RPDFZ is recommended to be 7600 for servo motor and be 6000 for step motor.

0	2	4	LINTX
0	2	5	LINTZ

LINTX and LINTZ are linear acceleration/deceleration time constants for X and Z axes in rapid traverse，the range of which is from 0 to 4000（unit is mS）.

Generally，LINTX and LINTZ are recommended to be 100 for servo motor and be 350 for step motor.

0	2	7		FEDMX

FEDMX is upper limit feedrate for X and Z axes，the range of which is from 10 to 8000（unit is mm/min）.

0	2	9		FEEDT

FEEDT is exponential acceleration/deceleration time constant for X and Z axes in cutting and manual feed，the range of which is from 0 to 4000（unit is mS）.

0	3	0		FEDFL

FEDFL is start speed in acceleration and end speed in deceleration during cutting，the range of which is from 0 to 8000（unit is mm/min）.

0	3	2		RPDFL

RPDFL is rapid traverse speed when rapid traverse override is F0，the range of which is from 6 to 4000（unit is mm/min）.

3.2.3 Safeguard for Machine

1	6	4						SPEN			

SPEN is 1：spindle rotation permitting input is valid
is 0：spindle rotation permitting input is invalid

1	6	8		SPB4	PB4	SPB3	PB3	***	***	***	***

SPB4 is 0：safeguard is closed when SGAT signal connecting to 0V
is 1：safeguard is closed when SGAT signal connecting to ＋24V
PB4 is 0：safeguard is invalid
is 1：safeguard is valid and *SP signal is invalid
SPB3 is 0：low pressure alarms when PRES signal connecting to 0V
is 1：low pressure alarms when PRES signal connecting to ＋24V
PB3 is 0：low pressure detection is invalid
is 1：low pressure detection is valid

1	7	2			MST	MSP	MOT	ESP			

MST is 0：ST signal is valid
is 1：ST signal is invalid，which can be defined by macro program（＃1014）
MSP is 0：SP signal is valid
is 1：SP signal is invalid，which can be defined by macro program（＃1015）
MOT is 0：check software overtravel
is 1：don't check software overtravel
ESP is 0：emergency stop is valid
is 1：emergency stop is invalid

0	6	9	PEALMTIM

PEALTIM is low pressure alarm time，the range is from 0 to 60000（unit is ms）.

PLC data DT021

0	2	1	

It's delay time for responding to DIQP signal after outputting M05 signal，the range is from 0 to1000（unit is ms）.

3.2.4 Reference Point Return

0	0	4		RDRN	DECI					

RDRN is 1：G00 in dry run at rapid traverse speed

is 0：G00 in dry run at feed speed

DECI is 1：decelerating signal at high level is valid in reference point return

is 0：decelerating signal at low level is valid in reference point return

0	0	5							PPD	

PPD is 1：relative coordinate is set by G50

is 0：relative coordinate isn't set by G50

0	0	6							ZMZ	ZMX

ZMZ is 1：reference point return mode C for Z axis

is 0：reference point return mode B for Z axis

ZMX is 1：reference point return mode C for X axis

is 0：reference point return mode B for X axis

0	0	7							ZCZ	ZCX

ZCZ is 1：ZDEC connects to ZPC in parallel

is 0：ZDEC connects to ZPC independently

ZCX is 1：XDEC connects to XPC in parallel

is 0：XDEC connects to XPC independently

0	1	1						ZNLK		

ZNLK is 1：keep moving once pressing direction key in reference point return

is 0：stop moving once releasing direction key in reference point return

0	1	2	APRS							ISOT

APRS is 1：set absolute coordinate-system specified by No.49 and No.50 parameter automatically after reference point return

is 0：don't set absolute coordinate-system automatically after reference point return

ISOT is 1：rapid traverse in JOG mode is valid when powering on or before reference point return

is 1：rapid traverse in JOG mode is invalid when powering on or before reference point return

0	1	4								ZRSZ	ZRSX

ZRSZ and ZRSX are 1：PC signal and DEC signal are needed in reference point return
are 0：PC signal and DEC signal aren't needed in reference point return

1	8	3	***	***	***	***	***	***	MZRZ	MARX

MZRZ is 1：return reference point by pressing the negative direction key for Z axis
is 0：return reference point by pressing the positive direction key for Z axis
MZRX is 1：return reference point by pressing the negative direction key for X axis
is 0：return reference point by pressing the positive direction key for X axis

0	3	3	ZRNFL

ZRNFL is low speed for X and Z axes in reference point return，the range of which is from 6 to 4000（unit is mm/min）

0	4	9	PRSX
0	5	0	PRSZ

PRSX and PRSZ are absolute coordinates value for X and Z axes after reference point return，the range is from 0 to 9999999.

1	1	3	REF_SPD

REF_SPD is rapid traverse speed of each axis in reference point return，the range is from 10 to 7650000（unit is mm/min）.

1	1	4	

It's offset in X machine reference point，the range is from –99999 to 99999（unit is 0.001mm）.

1	1	5	

It's offset in Z machine reference point，the range is from –99999 to 99999（unit is 0.001mm）.

3.2.5 Thread Function

1	7	5				THDA				

THDA is 1：exponential deceleration/acceleration in thread cutting
is 0：linear deceleration/acceleration in thread cutting

0	1	9	THDCH

THDCH is length of thread run-out in thread cutting，the range of which is from 1 to 255.
Width of thread run-out＝THDCH×0.1×screw leader

0	2	6	THRDT

THRDT is acceleration/deceleration time constant for the short axis in thread cutting，the range of which is from 0 to 4000（unit is mS）

0	2	8	THDFL

THDFL is initial speed for X and Z axis in thread cutting，the range of which is from 6 to 8000（unit is mm/min）

0	7	0

ENCODER_CNT

ENCODER_CNT is spindle encoder line number, the range is 100 to 5000 (unit is lpr).

1	1	0

MGR

MGR is spindle gear teeth number in driving ratio, the range is from 1 to 255.

1	1	1

SGR

SGR is encoder gear teeth number in driving ratio, the range is from 1 to 255.

3.2.6 Spindle Control

0	0	1

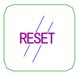
			SPIN				

Bit4 is 1: analog spindle

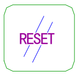
is 0: switching control spindle

0	0	9

				RSJG			

RSJG is 1: close spindle, coolant and lubrication when pressing 

key

is 0: keep spindle, coolant and lubrication when pressing 

key

1	6	4

AGER	AGIN	AGIM					

AGER is 1: automatic spindle gear shifting is valid

is 0: automatic spindle gear shifting is invalid

AGIN is 1: detect M41I and M42I signals in automatic spindle gear1 and gear2

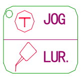
is 0: don't detect M41I and M42I signals in automatic spindle gear1 and gear2

AGIM is 1: M41I and M42I signals are valid when disconnecting with +24V

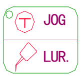
is 0: M41I and M42I signals are valid when connecting to +24V

1	7	5

SPHD							

SPHD is 1: 

key serves as spindle jogging

is 0: 

key serves as lubrication

0	2	1

VCV

It's voltage compensation value when spindle speed instruction is 10V, the range of which is from –2000 to 2000 (unit is mV)

0	3	6

SPDLC

SPDLC is voltage compensation value when SVC is 10V, the range of which is from –1000 to 1000 (unit is mV)

0	3	7	GRMAX1
0	3	8	GRMAX2
0	3	9	GRMAX3
0	4	0	GRMAX4

GRMAX1 ,GRMAX2, GRMAX3, GRMAX4 are the max. spindle speeds in gear 1 to 4 when SVC is 10V or for instruction M41 to M44 when automatic gear shifting is valid. Speed of gear1 is default value when powering on or automatic gear shifting is invalid.

The range is from 10 to 9999（unit is rpm）.

0	6	5	SFT1TME

SFT1IME is gear shifting time 1 of spindle，the range is from 0 to 60000（unit is ms）.

0	6	6	SFT2TME

SFT2IME is gear shifting time 2 of spindle，the range is from 0 to 60000（unit is ms）.

0	6	7	SFTREV

SFTREV is spindle voltage as gear shifting，the range is from 0 to 10000（unit is mV）.

0	8	0	MTIME

MTIME is duration for M code being executed，the range is form 100 to 5000（unit is ms）.

0	8	1	STIME

STIME is duration for S code being executed，the range is form 100 to 5000（unit is ms）.

0	8	7	SPDDLT

SPDDLT is delay time for M05 and SPZD outputting，the range is from 0 to 10000（unit is ms）.

0	8	9	SPZDTIME

SPZDTIME is SPZD output time，the range is from 0 to 60000（unit is ms）.

1	0	8	SPL_REV_TIME

SPL_REV_TIME is spindle jogging time，the range is from 0 to 60000（unit is ms）.

1	0	9	SPL_REV_SPD

SPL_REV_SPD is spindle jogging speed，the range is from 1 to 8000（unit is rpm）.

3.2.7 Chuck Control

1	6	4							SLSP	SLQP

SLSP is 1：don't detect whether spindle is clamped under chuck control

is 0： detect whether spindle is clamped under chuck control，otherwise，spindle can't be started

Book3 connection

SLQP is 1：chuck control is valid

is 0：chuck control is invalid

1	6	8				SPB3	PB3		PB2		PB1

SPB3 is 0：low pressure alarms when PRES signal connecting to 0V

is 1：low pressure alarms when PRES signal connecting to ＋24V

PB3 is 0：low pressure detection is invalid

is 1：low pressure detection is valid

PB2 is 0：don't detect to-position signal of chuck

is 1：detect to-position signal of chuck

PB1 is 0：internal chuck，NQPJ is clamping signal and WQPJ is unclamping signal for internal chuck

is 1：external chuck，NQPJ is unclamping signal and WQPJ is clamping signal for external chuck

0	6	9		PEALMTIM

PEALTIM is low pressure alarm time，the range is from 0 to 60000（unit is ms）.

PLC data DT021

0	2	1		

It's delay time for responding to DIQP signal after outputting M05 signal，the range is from 0 to1000（unit is ms）.

3.2.8 Tailstock Control

1	6	4							SLTW		

SLTW is 1：tailstock control is valid

is 0：tailstock control is invalid

3.2.9 Tool Nose Radius Compensation

0	0	2							刀补C		

Bit1 is 1：tool nose radius compensation is valid

is 0：tool nose radius compensation is invalid

0	0	3					TLC				

Bit4 is 1：compensate tool length with coordinates

is 0：compensate tool length with tool moving

0	0	4					ORC	TOC		PROD	

ORC is 1：radius value for tool compensation

is 0：diameter value for tool compensation

TOC is 1：keep tool compensation at reset

is 0：clear tool compensation at reset

PROD is 1：relative coordinate display don't include tool compensation

is 0：relative coordinate display includes tool compensation

0	1	2			DOFSI					

DOFSL is 1：toolsetting with trail cutting is valid

is 0：toolsetting with trail cutting is invalid

3.2.10 Toolpost Control

0	1	1							TSGN	TCPS

TSGN is 1：T01~T08 signals are valid when disconnecting with ＋24V

is 0：T01~T08 signals are valid when connecting to ＋24V

TCPS is 1：

is 0：

1	8	2						PB6		PB5

PB6 is 1：check tool signal at the end of tool change

is 0：don't check tool signal at the end of tool change

PB5 is 1：tool change mode A

is 0：tool change mode B

0	7	6	T1MAXT

T1MAXT is upper limit time for changing one tool，the range is form 100 to 5000（unit is ms）.

0	7	8	TLMAXT

TLMAXT is upper limit time for changing total tools，the range is form 1000 to 60000（unit is ms）.

0	8	2	T1TIME

T1TIME is delay time from TL+ ceased to TL- output，the range is from 0 to 4000（unit is ms）.

0	8	3	TCPWRN

TCPWRN is alarm time for not receiving *TCP signal，the range is from 0 to 4000（unit is ms）.

0	8	4	TMAX

TMAX is most tool number selection，the range is from 1 to 32.

0	8	5	TCPTIME

TCPTIME is time .from TL- output to receive TCP，the range is from 0 to 4000（unit is ms）

3.2.11 Edit and Display

0	0	5				M30				

M30 is 1：cursor returns after M30

is 0：cursor doesn't return after M30

0	1	2					EAL		EBCL	

EAL is 1：program can be edited at alarm

is 0：program can't be edited at alarm

EBCL is 1：end symbol EOB in part program is with “；”

is 0：end symbol EOB in part programs is with “*”

1	8	0									SPOS

SPOS is 1：remaining coordinates in POS and PROG windows

is 0：incremental coordinates in POS and PROG windows

1	8	4									LANG

LANG is 1：English display interface

is 0：Chinese display interface

3.2.12 Accuracy Compensation

0	0	3				PEC					

Bit5 is 1：pitch error compensation is valid

is 0：pitch error compensation is invalid

0	1	0		***	***	NOFC	***	CPF4	CPF3	CPF2	CPF1

CPF4~CPF1：frequency setting for backlash compensation（with BCD）

Frequency ＝（setting value＋1）Kpps

CPF4	CPF3	CPF2	CPF1	Setting value（Kpps）
0	0	0	0	1
0	0	0	1	2
0	0	1	0	3
0	0	1	1	4
0	1	0	0	5
0	1	0	1	6
0	1	1	0	7
0	1	1	1	8
1	0	0	0	9
1	0	0	1	10
1	0	1	0	11
1	0	1	1	12
1	1	0	0	13
1	1	0	1	14
1	1	1	0	15
1	1	1	1	16

0	1	1		BDEC	BD8						

BDEC is 1：backlash compensation mode B，with deceleration/acceleration for compensation and setting frequency is invalid

is 0：backlash compensation mode A，with fixed frequency for compensation

BD8 is 1：compensate backlash with one setting frequency

is 0：compensate backlash with eighth setting frequency

0	3	4	BKLX

BKLX is backlash compensation value of X axis，the range of which is from 0 to 2000（unit is 0.001mm）

0	3	5	BKLZ

BKLZ is backlash compensation value of Z axis，the range of which is from 0 to 2000（unit is 0.001mm）

0	9	8	

It's position number of pitch error compensation for X machine reference point，the range is from 0 to 255.

0	9	9	

It's position number of pitch error compensation for Z machine reference point，the range is from 0 to 255.

1	0	2	

It's distance between each pitch error compensation of X axis，the range is from 10000 to 999999（unit is ms）.

1	0	3	

It's distance between each pitch error compensation of Z axis，the range is from 10000 to 999999（unit is ms）.

Pitch error compensation table：

Serial number	X	Z
000	0	0
⋮		⋮
255	0	0

3.2.13 Communication Setting

0	0	2			RS232					

RS232 is 1：RS232 communication is valid

is 0：RS232 communication is invalid

0	4	4	BRATE0

BRATE0 is serial communication rate，which is valid when bit5 of No.2 parameter is 1.The range is 1200, 2400, 4800, 9600, 19200, 38400 57600 115200 (unit is bit/s).

Chapter 4　MACHINE DEBUGGING

Only debugging the machine when first powering on as following method，which will be introduced in detail in this chapter，can users operate the machine.

4.1　EMERGENCY STOP and OVERTRAVEL

For safety，hardware overtravel measure is recommended for GSK980TD，in which the travel-limit switch is fixed in positive and negative direction for each axis.The connection diagram as following：

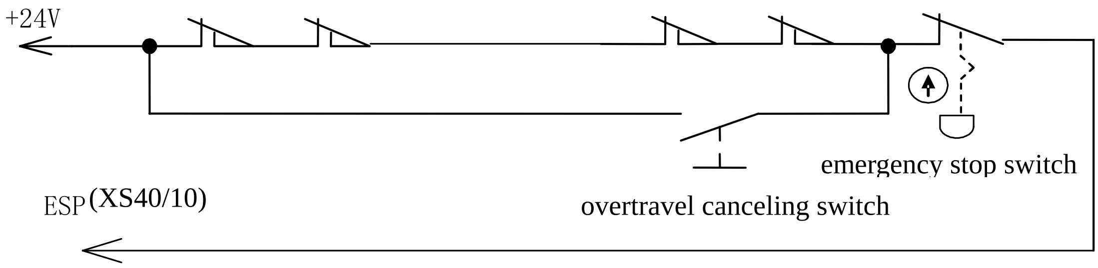


Fig.4-1

In this connection，bit2 of No.172 state parameter should be specified to 0.

Slowly moving for axis in MANUAL or HANDWHEEL mode can verify the validity of overtravel canceling switch、travel-limit switch and alarm display.When overtravel in presence or pushing the emergency stop switch，ESP alarm will be given，which can be cancelled by pushing the overtravel canceling switch and moving in the negative direction.

4.2　DRIVER SETTING

Specify bit1、bit0 of No.9 state parameter according to the driver alarm voltage，which should be specified to 1 to fit our own driver.

Bit1、bit0 of No.8 state parameter can be modified when the machine moving direction isn't consistent with the dictate desired direction.

4.3　GEAR RATIO ADJUSTING

When the machine moving distance isn't consistent with the distance display，adjust the gear ratio by specifying No.015～No.018 parameter to fit the different machine transmission ratio.

Formula：

$$\frac{CMR}{CMD} = \frac{\delta \times 360}{\alpha \times L} \times \frac{Z_M}{Z_D}$$

CMR：dictate multiple coefficient（No.15、No.16 data parameter）

CMD：dictate divisor coefficient（No.17、No.18 data parameter）

α：driver rotation angle for receiving one pulse

L：screw leader

Book3 connection

δ：the minimum input unit of CNC（0.001mm for X，0.05mm for Z of GSK980TD）

Z_M：gear number from screw side

Z_D：gear number from machine side

For example：Z_M＝50、Z_D＝30、α =0.075°、L＝4mm

Gear ratio of X ：

$$\frac{CMR}{CMD}=\frac{\delta\times 360}{\alpha\times L}\times\frac{Z_M}{Z_D}=\frac{0.0005\times 360}{0.075\times 4}\times\frac{50}{30}=\frac{1}{1}$$

Gear ratio of Z：

$$\frac{CMR}{CMD}=\frac{\delta\times 360}{\alpha\times L}\times\frac{Z_M}{Z_D}=\frac{0.001\times 360}{0.075\times 4}\times\frac{50}{30}=\frac{2}{1}$$

From above，No.15、16、18 parameters are 1 and No.17 parameter is 2.

The permitted max. Speed will descend when CMR is less than CMD.For example，the permitted max. speed of Z is 8000mm/min when No.16 parameter is 1 and No.18 parameter is 2.

The position accuracy will decline when CMR isn't equivalent to CMD.For example，CNC only outputs one pulse with the input increment being 0.005mm when No.16 parameter is 1 and No.18 parameter is 5.

To insure the position accuracy and speed，recommend to set the gear ratio of CNC to 1：1 and set that of servo to the calculated value.

As possible as one can，choose the step driver with subdivision function and select the reasonable machine transmission ratio to keep the gear ratio of CNC with 1：1.

4.4 ACCELERATING and DECELERATING CHARACTERISTIC

Adjust the relative parameters according to the factors of driver、motor characteristic and machine load etc：

No.22、No.23 data parameter：rapid traverse speed of X、Z.

No.24、No.25 data parameter：linear acc./dec. time constant of X、Z in rapid traverse.

No.26 data parameter：exponential acc./dec. time constant of X in thread cutting.

No.28 data parameter：the start/end speed during exponential acc./dec. in thread cutting.

No.29 data parameter：exponential acc./dec. time constant in cutting feed or manual feed.

No.30 data parameter：the start/end speed during exponential acc./dec. in cutting feed.

Bit3 of No.27 state parameter（SMZ）：whether there is smooth transition between the adjacent cutting segments.

Larger the acc./dec. time constant is，the acc./dec. process is more slow，the machine strike is smaller and the cutting efficiency is lower. v.v.

When the acc./dec. time constant is same，higher the start/end speed is，the acc./dec. process is more rapid，the machine strike is greater and the cutting efficiency is higher. v.v.

Acc./dec. characteristic regulating is on the principle of reducing the time constant and increasing the start/end speed properly，and insuring no driver alarm、no lost step、no obvious machine strike.

When bit3 of No.27 state parameter is 1，the cutting point of intersection is commanded position and the efficiency is be low，otherwise，arc transition occurs in this point which will cause smooth workpiece surface，and the efficiency is high.To avoid lost step，this bit parameter should be 1 when adopting step drivers.

Recommend to specify the parameters as following when adopting step drivers（gear ratio is 1：1）：

No.022≤2500	No.023≤5000
No.024≥350	No.025≥350
No.029≥150	No.028≤100
No.026≥200	No.030≤50

Book3 connection

Recommend to specify the parameters as following when adopting servo drivers（gear ratio is 1：1）:

No.022＝5000　　No.023＝10000

No.024≤60　　No.025≤60

No.029≤50　　No.028≤500

No.026≤50　　No.030≤400

Factually parameter setting must keep the motor，driver characteristic and machine load for reference.

4.5 REFERENCE POINT ADJUSTING

The parameters about reference point return as following：

Bit5（DECI）of No.4 parameter：the valid voltage level for decelerating signal in reference point return.

Bit0/1（ZMX/ZMZ）of No.6 parameter：the return direction in deceleration and backlash direction for X、Z axes in reference point return.

Bit0/1（ZCX/ZCZ）of No.7 parameter：whether take one approaching switch as decelerating signal and zero signal.

Bit2（ZNLK）of No.11 parameter：whether direction key is self-lock in reference point return.

Bit0/1（ZRSCX、ZRSCZ）of No.14 parameter：whether detect decelerating signal and zero signal for X、Z axes in reference point return.

No.33 parameter：the low speed for X、Z axes in reference point return.

Bit0/1（MZRX、MZRZ）of No.183 parameter：the reference point return direction is whether positive or negative for X、Z axes.

Reference point return can be realized after the travel-limit switch is valid.

Generally，the reference point is fixed at the farthest travel position and the bumper length is more than 25mm，which is long enough to insure the adequate decelerating distance，and confirm decelerating and accurate return.More rapid the speed is and longer the bumper is，the accuracy of reference point return is higher.

Generally，there are two connection methods for reference point return：

1、 matching the servo driver，the diagram of using a travel-limit switch and motor phase C signal as following：

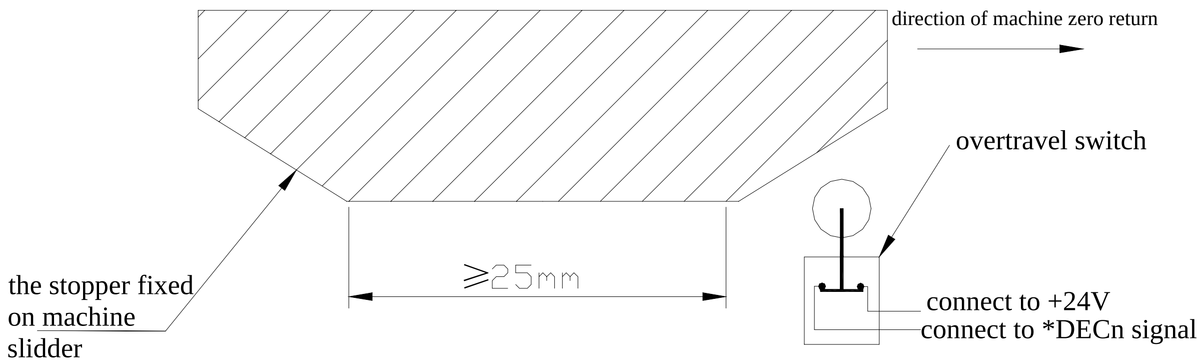


Fig.4-2

To improve the accuracy of reference point return，the encoder phase C signal should arrive when the motor rotates at half circle，at which the machine moving distance is gear number by the side of motor divided by that gear number by the side of thread multiplied by 2.

Parameter specification as following：

Bit5 of No.4 parameter is 0

Bit0、1 of No.6 parameter are 0

Bit0、1 of No.7 parameter are 0

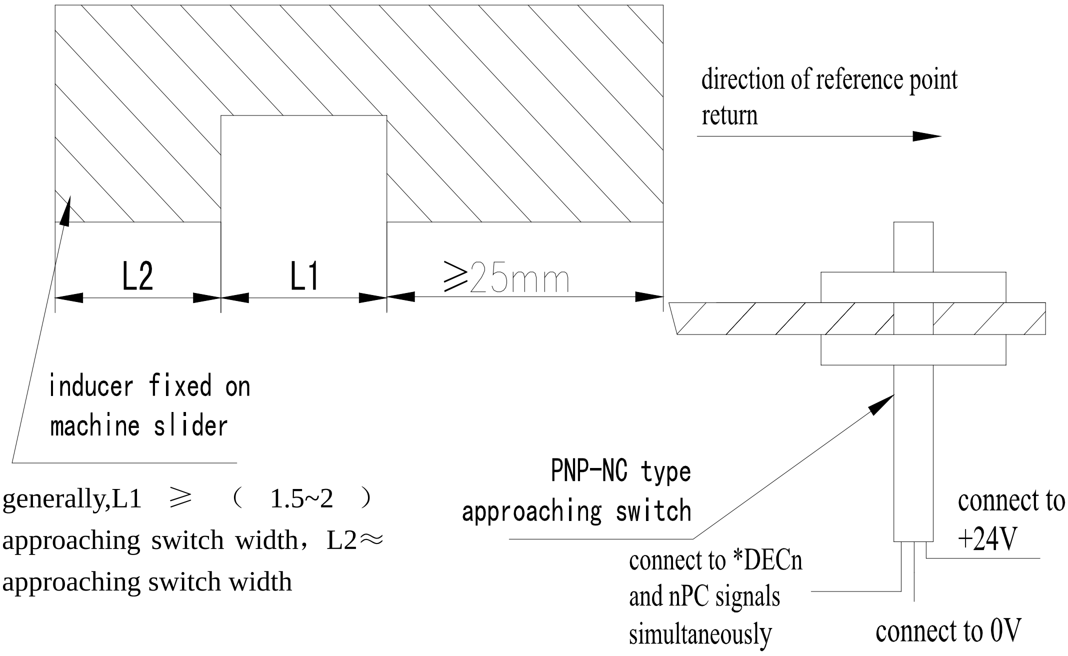
Bit2 of No.11 parameter are 1

Bit0、1 of No.14 parameter are 1

No.33 parameter is 200

Bit0、1 of No.183 parameter are 0

2、 matching the step driver，the diagram of using one approaching switch as following：



generally,L1 ≥ （ 1.5~2 ） approaching switch width，L2≈ approaching switch width

To avoid lost step in high speed，rapid override had better be specified to 50％.The parameter specification as following：

Bit5 of No.4 parameter is 0

Bit0、1 of No.6 parameter are 0

Bit0、1 of No.7 parameter are 1

Bit2 of No.11 parameter are 0

Bit0、1 of No.14 parameter are 1

No.33 parameter is 200

Bit0、1 of No.183 parameter are 0

Bit5 of DGN.000 diagnosis indicates decelerating signal，bit5 of DGN.001 indicates zero signal and bit0、1 of DGN.008 indicate PC signal.

4.6 SPINDLE FUNCTION ADJUSTING

4.6.1 Spindle Encoder

Encoder is in need for thread machining，which line number is from 100 to 5000 specified by No.70 parameter.The transmission ratio between encoder and spindle is from 1/255 to 255，in which the spindle gear number is specified by No.110 parameter and the encoder gear number is specified by No.111 parameter. Only synchronization strap transmission mode is in need（no glide transmission）.

Bit2 of DGN.008 diagnosis indicates whether the encoder phase C signal is valid.

DGN.11 and DGN.12 diagnosis indicate whether the encoder phase A/B signal is valid.

4.6.2 Spindle Brake

For improving efficiency，proper spindle brake time must be specified to stop spindle rapidly，and too long brake time will cause spindle burnout.

No.87 parameter：time for from commanding M05 to spindle braking

No.89 parameter：spindle brake time

4.6.3 Spindle Speed Controlled by On-off Variable

Speed dictate is S01～S04 when machine controlled by multi-speed motor，the relative parameters as following：

Bit4 of No.1 parameter is 0：spindle controlled by on-off variable

Bit0 of No.3 parameter is 0：four-gear spindle

4.6.4 Spindle Speed Controlled by Analog Voltage

Spindle speed controlled by analog voltage can be realized by specifying the parameters，in which 0～10V voltage will be outputted to control frequency convertor.The relative parameters as following：

Bit4 of No.1 parameter is 1：spindle speed controlled by analog voltage

No.21 parameter：the compensation voltage when spindle speed dictate is 10V

No.36 parameter：the compensation voltage when spindle speed dictate is 0V

No.37～No.40 parameters：the highest spindle speed for gear1～gear4，gear1 is default when powering on

Some parameters need to specify for the frequency convertor：

Positive or negative rotation mode：decided by VF port

Frequency setting mode：decided by FR mode

No.37～No.40 parameters can be specified to let the commanded speed be consistent with the real speed detected by encoder.The method is to select gear1 and command S9999 in MDI mode，then specified No.37 parameter with the real spindle speed value，which is displayed in the screen. The same methods as other gears.

The voltage should be 0V at S0 when the voltage is 10V at S9999，and the bias voltage can be cancelled by specifying No.21 and No.36 parameters（which have been adjusted correctly before leaving factory）.

If the voltage is more than 10V at S9999，No.21 parameter should be diminished.If the voltage is more than 0V at S0，No.36 parameter should be diminished.

That command S9999 in MDI mode and specified No.37 parameter with the speed value，which be displayed in the speed induction instrument，is feasible without encoders.

4.7 BACKLASH COMPENSATION

Backlash compensation value，which can be measured by centi-meter、thousandth meter、laser detector，is always diameter input and has nothing to do with program mode，the input unit is 0.001mm.Recommend to measure backlash compensation value as following：

- edit program

```
O0001；
N10 G01 W10 F800 ；
N20 W15 ；
```

N30 W1 ；

N40 W-1 ；

N50 M30 .

- set the backlash compensation value to 0 before measuring
- run the program in single block，and confirm the A point after positioning two times and record the current data，then move the distance of 1mm and move the same distance in reverse to B point，at which record the current data.

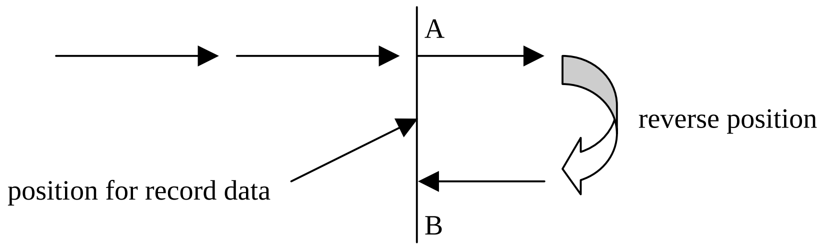


Fig.4-4 sketch map for measuring backlash compensation

- backlash compensation value is the absolute value of data at A minus data at B，which multiplied by 2 can be inputted to No.34 or No.35 parameter.

Data A is the centi-table data at A point

Data B is the centi-table data at B point

Note 1：backlash compensation mode and frequency can be specified by bit7 and bit6 of No.11 parameter.

Note 2：measure the backlash compensation value every three months for machine.

4.8 TOOLPOST ADJUSTING

All kinds of toolposts suit for GSK980TD，please read machine manuals for details.The relative parameters as followings：

Bit1 of No.11 parameter：valid voltage level of tool position signal.A pull-up resistor is needed when the voltage level is low.

No.76 parameter：time for moving one tool.

No.78 parameter：upper limit time for changing one tool.

No.82 parameter：total tools number.

No.85 parameter： lock-up time in negative rotation direction for toolpost

Incorrect phase sequence connection for three-phase power will lead to striking toolpost，when need to press RESET key and check connection.

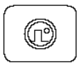
To avoid damaging motor or not locking toolpost up，No.85 parameter should be specified properly.The method for inspect whether the toolpost is locked-up is that take centi-meter close to the toolpost and turn the toolpost artificially，and the index float should be less than 0.01mm.

Bit7 and bit6 of DGN.005 diagnosis indicate whether TL+ and TL- signals are valid.

Bit0～3 of DGN.000 diagnosis indicate whether T01~T04 signals are valid.

To insure correctness of tool changing and proper time characteristic，each tool and max. tools must be changed in adjusting.

4.9 STEP/MPG ADJUSTING

whether  key is STEP or MPG mode can be specified by bit3 of No.11 parameter.

Bit3 is 1：STEP mode

is 0：MPG mode

To avoid too rapid rotation for manual pulse generator，the increment of 0.1mm should be screened when matching step drivers，which can be specified by bit1 of No.173 parameter.

Parameter number

1	7	3								SINC	

SINC is 0：the increment of 0.001、0.01、0.1 is valid in STEP/MPG mode.

is 1：the increment of 0.001、0.01 is valid in STEP/MPG mode.

4.10 OTHER ADJUSTING

1	6	4		AGER	AGIN	AGIM		SPEN	SLTW	SLSP	SLQP

AGER is 1：spindle gears automatically

is 0：spindle doesn’t gear automatically

AGIN is 1：detect M41I、M42I when spindle automatically gears to 1、2 gear

is 0：don’t detect M41I、M42I when spindle automatically gears to 1、2 gear

AGIM is 1：M41I and M42I are valid when connecting to 0V

is 0：M41I and M42I are valid when connecting to ＋24V

SPEN is 1：the function of spindle rotation permission is valid

is 0：the function of spindle rotation permission is invalid

SLTW is 1：tailstock control is valid

is 0：tailstock control is invalid

SLSP is 1：don’t detect whether the chuck is clamping

is 0：detect whether the chuck is clamping

SLQP is 1：chuck control is valid

is 0：chuck control is invalid

1	6	8		SPB4	PB4	SPB3	PB3		PB2		PB1

SPB4 is 0：defence gate is closed when SGAT signal connecting to 0V

is 1：defence gate is closed when SGAT signal connecting to ＋24V

PB4 is 0：defence gate is invalid

is 1：defence gate is valid and *SP signal is invalid

SPB3 is 0：low pressure alarms when PRES signal connecting to 0V

is 1：low pressure alarms when PRES signal connecting to ＋24V

PB3 is 0：low pressure detection is invalid

is 1：low pressure detection is valid

PB2 is 0：don’t detect to-position signal of chuck

is 1：detect to-position signal of chuck

PB1 is 0：inner chuck，NQPJ is clamping signal and WQPJ is loosing signal for inner chuck

is 1：outer chuck，NQPJ is loosing signal and WQPJ is clamping signal for outer chuck

1	7	2			MST	MSP	MOT				SOVI

MST is 0：ST signal is valid

is 1：ST signal is invalid，which can be defined by macro program（＃1014）

MSP is 0：SP signal is valid

is 1：SP signal is invalid，which can be defined by macro program（＃1015）

MOT is 0：check soft travel-limit

is 1：don’t check soft travel-limit

SOVI is 1：T01～T08 signals are valid

is 0：T01～T08 signals are invalid，and *0V1、*0V2、*0V4、*0V8 signals are valid

1	7	3								SINC	SOUS

SINC is 0：the increment of 0.001、0.01、0.1 is valid in STEP/MPG mode.

is 1：the increment of 0.001、0.01 is valid in STEP/MPG mode.

SOUS is 0：S1～S4 signals are valid when spindle controlled by on-off variable

is 1：S1、S2 signals are valid when spindle controlled by on-off variable，S3 and S4 are defined by macro program（＃1102、＃1103）

CHAPTER5 DIAGNOSIS INFORMATION

5.1 DIAGNOSIS of CNC

In this chapter，diagnosis of interface signals and inner states will be introduced.

5.1.1 Diagnosis Information from Machine

0 0 0	TCP	DIQP	XDEC	BDT	T04	T03	T02	T01
pins	XS39.12	XS39.11	XS40.1	XS40.2	XS40.3	XS40.4	XS40.5	XS40.6

TCP：*TCP signal /low pressure detecting signal（machine to PLC）

DIQP：chuck control signal（machine to PLC）

XDEC：deceleration signal for reference point return in X direction（machine to PLC）

BDT：optional block jumping signal（machine to PLC）

T04：tool selection signal T4（machine to PLC）

T03：tool selection signal T3（machine to PLC）

T02：tool selection signal T2（machine to PLC）

T01：tool selection signal T1（machine to PLC）

0 0 1	SP	ST	ZDEC	ESP	***	***	***	***
pins	XS40.7	XS40.8	XS40.9	XS40.10				

SP：pause signal SP（machine to PLC）

ST：cycle start signal ST（machine to PLC）

ZDEC：deceleration signal for reference point return in Z direction（machine to PLC）

ESP：ESP signal（machine to PLC）

0 0 2	T08/M42I	T07/M41I	T06	T05				
pins	XS40.19	XS40.20	XS40.21	XS40.22				

T08/M42I：T8/gear shifting in-position signal of spindle（machine to PLC）

T07/M41I：T7/gear shifting in-position signal of spindle（machine to PLC）

T06：tool signal T6（machine to PLC）

T05：tool signal T5（machine to PLC）

5.1.2 Diagnosis Information from CNC

0 0 4	SPZD	DOQPJ	M05	M32	M08	DOTWJ	M04	M03
pins	XS39.17	XS39.4	XS39.16	XS39.6	XS39.15	XS39.2	XS39.3	XS39.7

SPZD：spindle braked signal（PLC to machine）

DOQPJ：chuck clamping signal（PLC to machine）

M05：spindle stopping signal（PLC to machine）

M32：lubrication ON signal（PLC to machine）

M08：coolant ON signal（PLC to machine）

DOTWJ：tailstock going forward signal（PLC to machine）

Book3 connection

M04：spindle CCW rotation signal（PLC to machine）

M03：spindle CW rotation signal（PLC to machine）

0	0	5	TL-	TL+						
pins			XS40.13	XS40.12						

TL-：toolpost CCW rotation signal（PLC to machine）

TL+：toolpost CW rotation signal（PLC to machine）

5.1.3 Diagnosis Information of Axis State

0	0	8			RFZ	RFX		MPCS	ZPC	XPC

ZPC：reference point signal in Z direction（machine to CNC）

XPC：reference point signal in X direction（machine to CNC）

0	0	9							ZALM	XALM
pins									XS31.5	XS30.5

ZALM：alarm signal of Z axis（machine to CNC）

XALM：alarm signal of X axis（machine to CNC）

0	1	0								

Handwheel speed data：relative bit will be changed under valid input

0	1	1								
0	1	2								

Spindle feedback data：relative bit will be changed under valid spindle encoder input

0	1	3								
0	1	4								

Spindle analog voltage output：relative bit will be changed under spindle analog voltage output

5.1.4 Diagnosis of Keys

DGN.016～DGN.022 is for keys on edit panel and DGN.024～DGN.029 is for keys on machine control panel，the relative bit of which is 1 for being pressed and is 0 for being released，otherwise，the panel circuit is in fault.

0	1	6	RST	O	N	G	P/Q	7	8	9
keys			RESET	O	N	G	P Q	7	8	9

0	1	7	PGU	X	Z	U	W	4	5	6
keys				X	Z	U	W	4	5	6

0 1 8	PGD	H/Y	F/E	R/V	D/L	1	2	3
keys		H Y	F E	R V	L D	1	2	3

0 1 9	CRU	RIGHT	I/A	J/B	K/C	-	0	.
keys	⇧	⇨	I A	J B	K C	− ␣	0	·

0 2 0	CRD	LEFT	M	S	T	EOB	INS/ALT	DEL
keys	⇩	⇦	M	S	T	EOB	INSERT ALTER	DELETE

0 2 1	***	POS	RPG	OFT	ALM	SET	PAR	DGN
keys		POSITION	PROGRAM	OFFSET	ALARM	SETTING	PARAMETER	DIAGNOSIS

0 2 2	IN	OUT	CHG	/、#	CAN	***	***	***
keys	INPUT	OUTPUT	CHANGE	/ #	CANCEL			

0 2 4	EDT	AUT	MDI	HOME	HNDL	JOG	SBK	BDT
keys	EDIT	AUTO	MDI	MACHINE ZERO	MPG	JOG	SINGLE	SKIP

0 2 5	MLK	AFL	DRN	PHOME	0.001	0.01	0.1	HX
keys	MACHINE LOCK	MST M. S. T. LOCK	DRY	PROGRAM ZERO	0.001	0.01	0.1	X

0 2 6	***	HZ	***	X↑	***	Z←	RT	Z→
keys		Z						

0 2 7	***	X↓	***	SPP	COOL	SPS	RHST	SPM
keys				CW	COOLANT	STOP	JOG LUR.	CCW

0 2 8	JTOL	SP0+	TRV+	0V+	SP0-	TRV-	0V-	SP

Book3 connection

keys	TOOL	SPINDE OVERRIDE	RAPID OVERRIDE	FEEDRATE OVERRIDE	SPINDE OVERRIDE	RAPID OVERRIDE	FEEDRATE OVERRIDE	STOP

0	2	9								
keys			ST	***	***	***	***	***	***	***
			RUN							

5.1.5 Diagnosis between PLC and CNC

These signals between PLC and CNC are for user to check inner work state of CNC.

0	3	2	HX/RV1	***	XDEC	***	-X	+X	***	***

HX/RV1：X handwheel/rapid override signal（PLC to CNC）

XDEC：deceleration signal of X axis

-X：negative movement key for X axis（CNC to PLC）

+X：positive movement key for X axis（CNC to PLC）

0	3	3	HZ/RV2	***	ZDEC	***	-Z	+Z	***	***

HZ/RV1：Z handwheel/rapid override signal（PLC to CNC）

XDEC：deceleration signal of Z axis

-Z：negative movement key for Z axis（CNC to PLC）

+Z：positive movement key for Z axis（CNC to PLC）

0	3	4	DRN	***	***	***	GR2	GR1	***	***

DRN：dry run signal（PLC to CNC）

GR2：gear selecting input signal 2（PLC to CNC）

GR1：gear selecting input signal 1（PLC to CNC）

0	3	5	MLK	MP2	MP1	***	SBK	BDT	***	***

MLK：machine locked signal（PLC to CNC）

MP2：step/handwheel override signal（PLC to CNC）

MP1：step/handwheel override signal（PLC to CNC）

SBK：single block signal（PLC to CNC）

BDT：optional block jumping signal（PLC to CNC）

0	3	6	ZNR	SSTP	SOR	SAR	FIN	ST	STLK	***

ZNR：reference point return signal（PLC to CNC）

SSTP：spindle stop signal（PLC to CNC）

SOR：spindle orientation signal（PLC to CNC）

SAR：spindle speed reaching signal（PLC to CNC）

FIN：auxiliary function end signal（PLC to CNC）

ST: cycle start signal (PLC to CNC)

STLK: startup interlocking signal (PLC to CNC)

0	3	7		ERS	RT	SP	ESP	FV03	FV02	FV01	FV00

ERS: external reset signal (PLC to CNC)

RT: signal of rapid traverse in JOG mode (PLC to CNC)

SP: feed holding signal (PLC to CNC)

ESP: emergency stop signal (PLC to CNC)

FV03: feedrate override signal *OV8 (PLC to CNC)

FV02: feedrate override signal *OV4 (PLC to CNC)

FV01: feedrate override signal *OV2 (PLC to CNC)

FV00: feedrate override signal *OV1 (PLC to CNC)

0	3	8		PN8	PN4	PN2	PN1	KEY1	MD4	MD2	MD1

PN8: external program number selecting signal PN8 (PLC to CNC)

PN4: external program number selecting signal PN4 (PLC to CNC)

PN2: external program number selecting signal PN2 (PLC to CNC)

PN1: external program number selecting signal PN1 (PLC to CNC)

KEY1: program switch signal (PLC to CNC)

MD4: mode selecting (PLC to CNC)

MD2: mode selecting (PLC to CNC)

MD1: mode selecting (PLC to CNC)

0	4	0		R08I	R07I	R06I	R05I	R04I	R03I	R02I	R01I

R08I: spindle speed instruction signal (PLC to CNC)

R07I: spindle speed instruction signal (PLC to CNC)

R06I: spindle speed instruction signal (PLC to CNC)

R05I: spindle speed instruction signal (PLC to CNC)

R04I: spindle speed instruction signal (PLC to CNC)

R03I: spindle speed instruction signal (PLC to CNC)

R02I: spindle speed instruction signal (PLC to CNC)

R01I: spindle speed instruction signal (PLC to CNC)

0	4	1		SIND	SGN	***	***	R12I	R11I	R10I	R09I

SIND: spindle speed instruction selecting signal (PLC to CNC)

SGN: spindle instruction polarity selecting signal (PLC to CNC)

R12I: spindle speed instruction signal (PLC to CNC)

R11I: spindle speed instruction signal (PLC to CNC)

R10I: spindle speed instruction signal (PLC to CNC)

R09I: spindle speed instruction signal (PLC to CNC)

0	4	2		CDZ	SMZ	AFL	OVC	***	SOV2	SOV1	SOV0

CDZ: chamfering signal (PLC to CNC)

SMZ: error detecting signal (PLC to CNC)

AFL: auxiliary function locking signal (PLC to CNC)

OVC: feedrate override canceling signal (PLC to CNC)

SOV2: spindle override signal (PLC to CNC)

SOV1：spindle override signal（PLC to CNC）

SOV0：spindle override signal（PLC to CNC）

0	4	3	SKIP	***	***	***	***	***	***	***

SKIP：jumping signal

0	4	6	UI07	UI06	UI05	UI04	UI03	UI02	UI01	UI00

UI07：macro input signal UI7（PLC to CNC）

UI06：macro input signal UI6（PLC to CNC）

UI05：macro input signal UI5（PLC to CNC）

UI04：macro input signal UI4（PLC to CNC）

UI03：macro input signal UI3（PLC to CNC）

UI02：macro input signal UI2（PLC to CNC）

UI01：macro input signal UI1（PLC to CNC）

UI00：macro input signal UI0（PLC to CNC）

0	4	7	UI15	UI14	UI13	UI12	UI11	UI10	UI09	UI08

UI15：macro input signal UI15（PLC to CNC）

UI14：macro input signal UI14（PLC to CNC）

UI13：macro input signal UI13（PLC to CNC）

UI12：macro input signal UI12（PLC to CNC）

UI11：macro input signal UI11（PLC to CNC）

UI10：macro input signal UI10（PLC to CNC）

UI09：macro input signal UI9（PLC to CNC）

UI08：macro input signal UI8（PLC to CNC）

0	4	8	OP	SA	STL	SPL	ENB	***	ZP2	ZP1

OP：run signal（CNC to PLC）

SA：servo ready signal（CNC to PLC）

STL：cycle start signal（CNC to PLC）

SPL：feed holding signal（CNC to PLC）

ENB：spindle enabling signal（CNC to PLC）

ZP2：signal for reference point return ending（CNC to PLC）

ZP2：signal for reference point return ending（CNC to PLC）

0	4	9	MA	***	***	***	DEN	***	RST	AL

MA：CNC ready signal（CNC to PLC）

DEN：movement ending signal（CNC to PLC）

RST：reset signal（CNC to PLC）

AL：alarm signal（CNC to PLC）

0	5	0			DST		TF	SF		MF

DST：start in MDI mode（CNC to PLC）

TF：tool function selecting signal（CNC to PLC）

SF：spindle speed selecting signal（CNC to PLC）

MF：auxiliary function selecting signal（CNC to PLC）

0	5	1		M28	M24	M22	M21	M18	M14	M12	M11

M11～M28：2-bit BCD M code output

0	5	2		S28	S24	S22	S21	S18	S14	S12	S11

S11～S28：2-bit BCD S code output

0	5	3		T28	T24	T22	T21	T18	T14	T12	S11

T11～T28：2-bit BCD T code output

0	5	6		EX16	EX15	EX14	EX13	EX12	EX11	EX10	EX09

EX16~EX09：external information bit

0	5	7		EX08	EX07	EX06	EX05	EX04	EX03	EX02	EX01

EX08~EX01：external information bit

0	6	0		R08O	R07O	R06O	R05O	R04O	R03O	R02O	R01O

R08O～R01O：12-bit S code signal（CNC to PLC）

0	6	1			SFW		SRV	R12O	R11O	R10O	R09O

SFW：SFW=1 SRV=0

SRV：SFW=1 SRV=0

R12O～R09O：12-bit S code signal（CNC to PLC）

0	6	2		UO07	UO06	UO05	UO04	UO03	UO02	UO01	UO00

UO07~UO00：macro output signal（CNC to PLC）

0	6	3		UO15	UO14	UO13	UO12	UO11	UO10	UO09	UO08

UO15~UO08：macro output signal（CNC to PLC）

0	6	4		AGER	AGIN	AGIM	***	SPEN	SLTW	SLSP	SLQP

AGER：spindle automatic gear shifting is valid（CNC to PLC）

AGIN：detect gear shifting in-position signal（CNC to PLC）

AGIM：valid level for M41,M42（CNC to PLC）

SPEN：spindle rotation permitting input signal（CNC to PLC）

SLTW：Tailstock control is valid/invalid（CNC to PLC）

SLSP：check whether chuck is clamped or not（CNC to PLC）

SLQP：chuck control is valid/invalid（CNC to PLC）

0	6	5									

Bit 0～bit7 of gear shifting time 1

0	6	6									

Bit 0～bit7 of gear shifting time 2

0	6	7									

Bit0～bit7 of spindle gear shifting speed

0	6	8		SPB4	PB4	SPB3	PB3		PB2		PB1

SPB4：level for closing protection gate（CNC to PLC）

PB4：protection gate function is valid（CNC to PLC）

SPB3：alarm level for low pressure（CNC to PLC）

PB3：low pressure detection is valid（CNC to PLC）

PB2：chuck in-position signal detecting（CNC to PLC）

PB1：chuck mode selecting（CNC to PLC）

0	7	1								MZRZ	MZRX

MZRZ：direction selecting of reference point return in Z direction

MZRX：direction selecting of reference point return in X direction

0	7	2			MST	MSP	MOT	MESP			SOVI

MST：shield external cycle start signal

MSP：shield external pause signal

MOT：no detecting software overtravel

MESP：shield external emergency stop signal

SOVI：T05~T08 is external override switch

0	7	3		SOT	MPOF		ESCD			SINC	SUOS

SOT：valid overtravel after reference point return

MPOF：no detecting low voltage alarm

ESCD：losing S code at emergency stop

SINC：handwheel increment ×100 is invalid

SUOS：S code/macro output selection

0	7	4						KY1	MD4	MD2	MD1

KY1：program switch ON as power on

MD4：operating mode selecting as power on

MD2：operating mode selecting as power on

MD1：operating mode selecting as power on

0	7	5		T07	T06	T05	T04	T03	T02	T01	T00

T07～T00：tool signal T8～T1

0	7	6									

Bit0～bit7 of the max .time of changing one tool（×16ms）

0	7	7									

Bit8～bit15 of the max .time of changing one tool（×16ms）

0	7	8									

Bit0～bit7 of the max .time of changing total tools（×16ms）

0	7	9									

Bit8～bit15 of the max .time of changing total tools（×16ms）

080								

Bit0～bit7 of M code waiting time/pulse width（×128ms）

081								

Bit0～bit7 of S code waiting time/pulse width（×128ms）

082								

Bit0～bit7 of time from TL+ cease to TL- output（×16ms）

083								

Bit0～bit7 of alarm time for not receiving *TCP（×64ms）

084								

Bit0～bit7 of total tool numbers

085								

Bit0～bit7 of time from TL- output to receiving *TCP（×16ms）

087								

Bit0～bit7 of time from M05 to SPZD output（×16ms）

088								

Bit0～bit7 of time from M05 to SPZD output（×16ms）

089								

Bit0～bit7 of SPZD output time （×16ms）

090								

Bit7～bit15 of SPZD output time （×16ms）

5.1.6 Inner State of CNC

DGN.096 and DGN.097 are for user to check current work state of CNC under the condition of no alarm and no movement in AUTO mode .

096		CSCT	CITL	COVL	CINP	CDWL	CMTN	CFIN

CSCT：waiting for spindle gear shifting signal

CITL：interlock signal is valid

COVL：override is 0%

CINP：detecting bit

CDWL：G04 being executed

CMTN：move instruction being executed

CFIN：M.S.T instruction being executed

097			CRST				CTRD	CTPU

CRST：emergency stop/external reset

CTRD：RS232 interface is receiving

CTRD：RS232 interface is transmitting

100	STP	REST	EMS		RSTB			CSU

STP：stop distributing interpolation

REST：external reset button

EMS：external emergency stop is valid

RSTB：RESET key in MDI panel is valid

CSU：emergency stop ON/servo alarm set

101								

Bit0～bit7 of X track error

102								

Bit0～bit7 of Z track error

103								

PLC execution time（ms）

104								

Error times of hardware test

105								

Spindle encoder counter value

106								

Handwheel counter value

5.2 STATE of PLC

This section is to detect the signal state between CNC and machine or between CNC and PLC or of alarm address A，which can't be modified.

5.2.1 Address X（machine to PLC，defined by standard PLC）

X0000	TCP	DIQP	ESP	T5	XDEC	BDT	T4	T3

TCP：Toolpost locking signal

DIQP：Chuck inputting signal(DIQP)

ESP：Emergency stop signal

T5：Tool signal T05

XDEC：Deceleration signal in X direction (XDEC)

BDT：Optional block jumping signal(BDT)

T4：Tool signal T04

T3：Tool signal T03

X0001	T2	T1	T8	SP	ZDEC	T6	ST	T7

T2：Tool signal T02
T1：Tool signal T01
T8：Tool signal T08
SP：External pause signal(SP)/safeguard signal
ZDEC：Deceleration signal in Z direction
T6：Tool signal T06
ST：External signal for cycle starting(ST)
T7：Tool signal T07

X0008				ESP				

ESP：Emergency stop signal

X0009							ZDEC	XDEC

ZDEC：Deceleration signal in Z direction
XDEC：Deceleration signal in X direction

X0010					XPC	XALM		

XPC：Zero signal in X direction
XALM：Driver alarm signal in X direction

X0015					ZPC	ZALM		

ZPC：Zero signal in Z direction
ZALM：Driver alarm signal in Z direction

X0020	OBJ	SBK	JOG	HDW	MRT	MDI	AUTO	EDIT

OBJ：Optional block jumping mode key
SBK：Single block mode key
JOG：JOG mode key
HDW：Handwheel mode key
MRP：Reference point return mode key
MDI：MDI mode key
AUTO：AUTO mode key
EDIT：EDIT mode key

X0021	XHW	0.1	0.01	0.001	PRT	DYR	M.S.T	MLK

XHW：X handwheel key
0.1：0.1 increment key
0.01：0.01 increment key
0.001：0.001 increment key
PRT：Program reference point return mode key
DYR：Dry run mode key
M.S.T：Auxiliary function locked key
MLK：Machine locked key

X0022	ZRG	RTR	ZLF		XUP		ZHW	YHW

ZRG："Z (RIGHT)" key in JOG mode

RTR：Rapid traverse key in JOG mode

ZLF："Z (LEFT)" key in JOG mode

XUP："X (UP)"key in JOG mode

ZHW：Z handwheel key

YHW：Y handwheel key

X0023	M4	M32	M5	M8	M3		XDW	

M4：M4 key

M32：M32 key

M5：M5 key

M8：M8 key

M3：M3 key

XDW："X (DOWN)"key in JOG mode

X0024	FDH	FO－	RO－	SO－	FO+	RO＋	SO＋	TCH

FDH：Feed hold key

FO-：Feedrate override - key

RO-：Rapid traverse override- key

SO-：Spindle override- key

FO+：Feedrate override + key

RO+：Rapid traverse override+ key

SO+：Spindle override+ key

TCH：Manual tool change key

X0025								ST

ST：Cycle start key

X0026								RST

RST：Reset key

5.2.2 Address Y （machine to PLC，defined by standard PLC）

Y0000	SPZD	DOQPJ	M5	M4	M8	M10	S3	S2

SPZD：Spindle braked signal/Y17

DOQPJ：Chuck clamping signal/Y16

M5：M05

M4：M04

M8：M08

M10：M10

S3：S3/M43/UO2

S2：S2/M42/UO1

Y0001	TL−	**TL+**	DOQPS	U4	S4	M3	M32	S1

TL-：Signal of toolpost rotating CCW

TL+：Signal of toolpost rotating CW

DOQPS：Chuck unclamping signal

U4：UO4 signal

S4：S04 signal

M3：M03 signal

S1：S01 signal

Y0002	***	***	***	***	***	***	***	***

Signals from PLC to machine

Y0003	***	***	***	***	***	***	***	***

Signals from PLC to machine

Y0004								

Lamp in machine control panel

Bit7：Lamp for machine reference point return or program reference point return ended in X direction

Bit5：Lamp for machine reference point return or program reference point return ended in Z direction

Bit4：Rapid traverse lamp

Bit3：Single block running lamp

Bit2：Optional block jumping lamp

Bit1：Lamp for machine locked

Bit0：Lamp for auxiliary function locked

Y0005								

Lamp in machine control panel

Bit7：EDIT mode lamp

Bit6：AUTO mode lamp

Bit5：MDI mode lamp

Bit4：Reference point return mode lamp

Bit3：Handwheel/increment mode lamp

Bit2：JOG mode lamp

Bit1：Spindle CW rotation lamp

Bit0：Coolant lamp

Y0006								

Lamp in machine control panel

Bit7：Single block mode lamp

Bit6：Optional block jumping mode lamp

Bit5：Lamp for machine locked

Bit4：Lamp for auxiliary function locked

Bit3：Dry run lamp

Bit2：Program reference point return mode lamp

Bit1：Rapid feed lamp

Bit0：Spindle stopping lamp

Y0007								

Lamp in machine control panel

Bit7：Single block mode lamp

Bit6：Optional block jumping mode lamp

Bit5：Lamp for machine locked

Bit4：Lamp for auxiliary function locked

Bit3：Dry run lamp

Bit2：Program reference point return mode lamp

Bit1：Rapid feed lamp

Bit0：Spindle stopping lamp

Y0008								

Lamp in machine control panel

Bit7：Spindle jogging lamp

Bit6：Max. spindle override lamp

Bit5：Min. rapid traverse override lamp

Bit4：Max. feedrate override lamp

Bit3：Min. spindle override lamp

Bit2：Max. rapid traverse override lamp

Bit1：Min. feedrate override lamp

Bit0：Cycle start lamp

Y0009	***	***	***	***	***	***		

Bit1：Dry run lamp in MDI panel

Bit0：Feed pausing lamp in machine control panel

Y0010	***	***	***	***	***	***	***	XEN1

XEN1：Driver enabling signal 1 in X direction

Y0011	***	***	***	***	***	***	XSET	XEN2

XSET：Driver move signal in X direction

XEN2：Driver enabling signal 2 in X direction

Y0012	***	***	***	***	***	***	***	***

Signals from PLC to machine

Y0013	***	***	***	***	***	***	***	***

Signals from PLC to machine

Y0014	***	***	***	***	***	***	***	ZEN1

ZEN1：Driver enabling signal 1 in Z direction

Y0015	***	***	***	***	***	***	ZSET	ZEN2

ZSET：Driver move signal in Z direction

ZEN2：Driver enabling signal 2 in Z direction

Y0016	***	***	***	***	***	***	***	***

Signals from PLC to machine

Y0017	***	***	***	***	***	***	***	***

Signals from PLC to machine

Y0018	***	***	***	***	***	***	***	***

Signals from PLC to machine

Y0019	***	***	***	***	***	***	***	***

Reserved

5.2.3 Address F（CNC to PLC）

F0000	OP	SA	STL	SPL	***	***	***	***

OP：Running signal in AUTO mode

SA：Servo ready signal

STL：Signal of cycle starting lamp

SPL：Signal of feedrate pausing lamp

F0001	MA	***	TAP	ENB	DEN	***	RST	AL

MA： CNC ready signal

TAP：Tapping signal

ENB：Spindle enabling signal

DEN：Distribution ending signal

RST：Reset signal

AL：Alarm signal

F0002	MDRN	CUT	***	SRNMV	THRD	CSS	RPDO	***

MDRN：Signal of Dry run mode detecting

CUT：Cutting feed signal

SRNMV：Program starting signal

THRD：Thread cutting signal

CSS：Constant surface speed signal

RPDO：Rapid traverse signal

F0003	***	MEDT	MMEM	***	MMDI	MJ	MH	MINC

MEDT：Detection signal of EDIT mode selecting

MMEM：Detection signal of AUTO mode selecting

MMDI：Detection signal of MDI mode selection

MJ：Detection signal of JOG mode selection

MH：Detection signal of Handwheel mode selection

MINC：Detection signal of Increment mode selection

F0004	***	MPST	MREF	MAFL	MSBK	MABSM	MMLK	MBDT1

MPST：Detection signal of Program reference point return

MREF：Detection signal of machine reference point return in JOG mode
MAFL：Detection signal of auxiliary function locked
MSBK：Detection signal of single Block mode
MABSM：Detection signal of absolute value in JOG mode
MMLK：Detection signal of machine locked
MBDT1：Detection signal of optional block jumping

F0007	***	***	***	***	TF	SF	***	MF

TF：Tool function selecting signal
SF：Spindle speed function selecting signal
MF：M function selecting signal

F0009	DM00	DM01	DM02	DM30	***	***	***	***

DM00：M decoding signal
DM01：M decoding signal
DM02：M decoding signal
DM30：M decoding signal

F0010	M7	M6	M5	M4	M3	M2	M1	M0

M7：M function code M07
M6：M function code M06
M5：M function code M05
M4：M function code M04
M3：M function code M03
M2：M function code M02
M1：M function code M01
M0：M function code M00

F0011	M15	M14	M13	M12	M11	M10	M9	M8

M15：M function code M15
M14：M function code M14
M13：M function code M13
M12：M function code M12
M11：M function code M11
M10：M function code M10
M9：M function code M09
M8：M function code M08

F0012	M23	M22	M21	M20	M19	M18	M17	M16

M23：M function code M23
M22：M function code M22
M21：M function code M21
M20：M function code M20
M19：M function code M19
M18：M function code M18
M17：M function code M17
M16：M function code M16

F0013	M31	M30	M29	M28	M27	M26	M25	M24

M31：M function code M31
M30：M function code M30
M29：M function code M29
M28：M function code M28
M27：M function code M27
M26：M function code M26
M25：M function code M25
M24：M function code M24

F0022	S7	S6	S5	S4	S3	S2	S1	S0

S7: spindle speed code signal S07
S6: spindle speed code signal S06
S5: spindle speed code signal S05
S4: spindle speed code signal S04
S3: spindle speed code signal S03
S2: spindle speed code signal S02
S1: spindle speed code signal S01
S0: spindle speed code signal S00

F0023	S15	S14	S13	S12	S11	S10	S9	S8

S15: spindle speed code signal S15
S14: spindle speed code signal S14
S13: spindle speed code signal S13
S12: spindle speed code signal S12
S11: spindle speed code signal S11
S10: spindle speed code signal S10
S9: spindle speed code signal S09
S8: spindle speed code signal S08

F0024	S23	S22	S21	S20	S19	S18	S17	S16

S23: spindle speed code signal S23
S22: spindle speed code signal S22
S21: spindle speed code signal S21
S20: spindle speed code signal S20
S19: spindle speed code signal S19
S18: spindle speed code signal S18
S17: spindle speed code signal S17
S16: spindle speed code signal S16

F0025	S31	S30	S29	S28	S27	S26	S25	S24

S31: spindle speed code signal S31
S30: spindle speed code signal S30
S29: spindle speed code signal S29
S28: spindle speed code signal S28

S27: spindle speed code signal S27

S26: spindle speed code signal S26

S25: spindle speed code signal S25

S24: spindle speed code signal S24

F0026	T7	T6	T5	T4	T3	T2	T1	T0

T7: T function code T07

T6: T function code T06

T5: T function code T05

T4: T function code T04

T3: T function code T03

T2: T function code T02

T1: T function code T01

T0: T function code T00

F0027	T15	T14	T13	T12	T11	T10	T9	T8

T15: T function code T15

T14: T function code T14

T13: T function code T13

T12: T function code T12

T11: T function code T11

T10: T function code T10

T9: T function code T09

T8: T function code T08

F0028	T23	T22	T21	T20	T19	T18	T17	T16

T23: T function code T23

T22: T function code T22

T21: T function code T21

T20: T function code T20

T19: T function code T19

T18: T function code T18

T17: T function code T17

T16: T function code T16

F0029	T31	T30	T29	T28	T27	T26	T25	T24

T31: T function code T31

T30: T function code T30

T29: T function code T29

T28: T function code T28

T27: T function code T27

T26: T function code T26

T25: T function code T25

T24: T function code T24

F0036	R080	R070	R060	R050	R040	R030	R020	R010

R08O: S 12-bit code signal R08O

R07O: S 12-bit code signal R07O

R06O: S 12-bit code signal R06O

R05O: S 12-bit code signal R05O

R04O: S 12-bit code signal R04O

R03O: S 12-bit code signal R03O

R02O: S 12-bit code signal R02O

R01O: S 12-bit code signal R01O

F0037	***	***	***	***	R12O	R11O	R10O	R09O

R12O: S 12-bit code signal R12O

R11O: S 12-bit code signal R11O

R10O: S 12-bit code signal R10O

R09O: S 12-bit code signal R09O

F0053	***	***	***	BGEACT	***	***	***	***

BGEACT：Background busy signal

F0054	U07	U06	U05	U04	U03	U02	U01	U00

UO7: Custom macro output signal UO07

UO6: Custom macro output signal UO06

UO5: Custom macro output signal UO05

UO4: Custom macro output signal UO04

UO3: Custom macro output signal UO03

UO2: Custom macro output signal UO02

UO1: Custom macro output signal UO01

UO0: Custom macro output signal UO00

F0055	U015	U014	U013	U012	U011	U010	U09	U08

UO15: Custom macro output signal UO15

UO14: Custom macro output signal UO14

UO13: Custom macro output signal UO13

UO12: Custom macro output signal UO12

UO11: Custom macro output signal UO11

UO10: Custom macro output signal UO10

UO9: Custom macro output signal UO09

UO8: Custom macro output signal UO08

F0056	U107	U106	U105	U104	U103	U102	U101	U100

U107: Custom macro output signal UO107

U106: Custom macro output signal UO106

U105: Custom macro output signal UO105

U104: Custom macro output signal UO104

U103: Custom macro output signal UO103

U102: Custom macro output signal UO102

U101: Custom macro output signal UO101

U100: Custom macro output signal UO100

F0057		U115	U114	U113	U112	U111	U110	U109	U108

U115: Custom macro output signal UO115
U114: Custom macro output signal UO114
U113: Custom macro output signal UO113
U112: Custom macro output signal UO112
U111: Custom macro output signal UO111
U110: Custom macro output signal UO110
U109: Custom macro output signal UO109
U108: Custom macro output signal UO108

F0058		U123	U122	U121	U120	U119	U118	U117	U116

U123: Custom macro output signal UO123
U122: Custom macro output signal UO122
U121: Custom macro output signal UO121
U120: Custom macro output signal UO120
U119: Custom macro output signal UO119
U118: Custom macro output signal UO118
U117: Custom macro output signal UO117
U116: Custom macro output signal UO116

F0059		U131	U130	U129	U128	U127	U126	U125	U124

U131: Custom macro output signal UO131
U130: Custom macro output signal UO130
U129: Custom macro output signal UO129
U128: Custom macro output signal UO128
U127: Custom macro output signal UO127
U126: Custom macro output signal UO126
U125: Custom macro output signal UO125
U124: Custom macro output signal UO124

F0065		***	***	***	***	***	***	RGSPM	RGSPP

RGSPM：Spindle rotating CCW in rigid tapping
RGSPP：Spindle rotating CW in rigid tapping

F0070		PSW8	PSW7	PSW6	PSW5	PSW4	PSW3	PSW2	PSW1

PSW8: Position switch signal PSW8
PSW7: Position switch signal PSW7
PSW6: Position switch signal PSW6
PSW5: Position switch signal PSW5
PSW4: Position switch signal PSW4
PSW3: Position switch signal PSW3
PSW2: Position switch signal PSW2
PSW1:Position switch signal PSW1

F0071		***	***	***	***	***	***	PSW10	PSW9

PSW10: Position switch signal PSW10
PSW9:Position switch signal PSW9

F0075	***	***	DRNO	MLKO	SBKO	BDTO	AFLO	***

DRNO：Soft key of dry run mode
MLKO：Soft key of machine locked
SBKO：Soft key of single block
BDTO：Soft key of optional block jumping
ALFO：Soft key of auxiliary function locked

F0076	***	***	***	***	RTAP	***	***	***

RTAP：Rigid tapping mode signal

F0094	***	***	***	***	ZP4	ZP3	ZP2	ZP1

ZP4：End signal of reference point return ZP4
ZP3：End signal of reference point return ZP3
ZP2：End signal of reference point return ZP2
ZP1：End signal of reference point return ZP1

F0102	***	***	***	***	MV4	MV3	MV2	MV1

MV4：Axis moving signal MV4
MV3：Axis moving signal MV3
MV2：Axis moving signal MV2
MV1：Axis moving signal MV1

F0106	***	***	***	***	MVD4	MVD3	MVD2	MVD1

MVD4：Axis direction signal MVD4
MVD3：Axis direction signal MVD3
MVD2：Axis direction signal MVD2
MVD1：Axis direction signal MVD1

F0120	***	***	***	***	ZRF4	ZRF3	ZRF2	ZRF1

ZRF4：Signal of reference point created ZRF4
ZRF3：Signal of reference point created ZRF3
ZRF2：Signal of reference point created ZRF2
ZRF1：Signal of reference point created ZRF1

F0197	***	***	***	***	***	***	MDOUT	***

MDOUT：OUT key in MDI mode is valid

F0198	***	***	***	***	PRO4	PRO3	PRO2	PRO1

PRO4：End signal of program reference point return PRO4
PRO3：End signal of program reference point return PRO3
PRO2：End signal of program reference point return PRO2
PRO1：End signal of program reference point return PRO1

F0199	***	***	***	***	***	***	***	MSPHD

MSPHD：Detection signal of spindle jogging

F0200		***	***	***	SIMSPL	***	***	***	***

SIMSPL: Analog spindle is valid

F0201		***	***	DECI	***	***	DCS	***	***

DECI: Level selecting of DEC signal in reference point return

DCS: Whether [OUT] key can start programs in MDI mode

F0203		***	***	***	OVRI	***	***	***	***

OVRI: Override canceling is valid

F0204		***	***	***	***	RSJG	***	***	***

RSJG: Function selecting of [RESET] for output port

F0205		***	***	***	***	***	ZNIK	TSGN	TCPS

ZNIK: Axis moving key held is valid as reference point return

TSGN: Level selecting of T* signal

TCPS: Level selecting of *TCP signal

F0207		***	***	***	***	***	***	***	***

Total tool numbers

Bit0: Bit 0 of tool number

Bit1: Bit 1 of tool number

Bit2: Bit 2 of tool number

Bit3: Bit 3 of tool number

Bit4: Bit 4 of tool number

Bit5: Bit 5 of tool number

Bit6: Bit 6 of tool number

Bit7: Bit 7 of tool number

F0208	***	AGER	AGIN	AGIM	***	SPEN	SLTW	SLSP	SLQP

SLQP: Chuck control is valid/invalid

SLSP: Whether chuck clamped is detected or not

SLTW: Tailstock control is valid/invalid

SPEN: Input signal of spindle rotation

AGIM: Valid level for M41I,M42I

AGIN: Detect gear shifting in-position signal

AGER: Spindle gears automatically

F0209	***	SPB4	PB4	SPB3	PB3	***	PB2	***	PB1

PB1: Select chuck mode

PB2: Detect chuck in-position signal

PB3: Valid function of checking low pressure

SPB3: Alarm level for low pressure

PB4: Safeguard is valid

SPB4: Level for closing safeguard

F0210		***	MST	MSP	***	MESP	***	***	SOVI

SOVI:External feedrate override switch(T05-T08)

MESP:Shield external emergency stop signal

MSP: Shield external pause signal

MST: Shield external start signal

F0211	***	***	***	***	***	***	SINC	SUOS

SUOS：Selection for S code/macro output

SINC：Selection for handwheel/increment 0.1

F0212	***	***	***	COVL	CINP	CDWL	***	***

CDWL: G04 is being executed

CINP: Bit checking is executed

COVL: Override is 0%

F0213	***	***	***	***	***	***	CTRD	CTPU

CTPU: RS232 interface is transmitting

CTRD: RS232 interface is receiving

F0214	SPHD	***	***	***	***	***	ZVAL	XVAL

XVAL: Selecting X direction

ZVAL: Selecting Z direction

SPHD：Spindle jogging/lubrication ON

5.2.4 Address G（PLC to CNC）

G0004					FIN			

FIN：Signal of auxiliary function ended

G0005		**AFL**						

AFL：Signal of auxiliary function locked

G0006		SKIPP		OVC		ABSM		SRN

SRN: Program restarting signal

ABSM: Absolute value signal in JOG

OVC: Feedrate override cancelling signal

SKIPP: Jumping signal

G0007	RLSOT	EXLM				ST	STLK	

STLK: Start function locking signal

ST: Cycle start signal

EXLM: Stored overtravel selecting signal

RLSOT: Overtravel releasing signal

G0008	ERS		SP	ESP				IT

IT: Interlocking signal

ESP: Emergency stop signal

SP: Feedrate pausing signal

ERS: External reset signal

G0009					PN8	PN4	PN2	PN1

PN1: Part no. searching signal PN1
PN2: Part No. searching signal PN2
PN4: Part No. searching signal PN4
PN8: Part No. searching signal PN8

G0010	JV07	JV06	JV05	JV04	JV03	JV02	JV01	JV00

JV00: JOG override signal JV00
JV01: JOG override signal JV01
JV02: JOG override signal JV02
JV03: JOG override signal JV03
JV04: JOG override signal JV04
JV05: JOG override signal JV05
JV06: JOG override signal JV06
JV07: JOG override signal JV07

G0011	JV15	JV14	JV13	JV12	JV11	JV10	JV09	JV08

JV15: JOG override signal JV15
JV14: JOG override signal JV14
JV13: JOG override signal JV13
JV12: JOG override signal JV12
JV11: JOG override signal JV11
JV10: JOG override signal JV10
JV09: JOG override signal JV09
JV08: JOG override signal JV08

G0012	FV07	FV06	FV05	FV04	FV03	FV02	FV01	FV00

FV00: Feedrate override signal FV00
FV01: Feedrate override signal FV01
FV02: Feedrate override signal FV02
FV03: Feedrate override signal FV03
FV04: Feedrate override signal FV04
FV05: Feedrate override signal FV05
FV06: Feedrate override signal FV06
FV07: Feedrate override signal FV07

G0014	*	*	*	*	*	*	RV2	RV1

RV1: Rapid feedrate override signal RV1
RV2: Rapid feedrate override signal RV2

G0018	*	*	*	*	*	*	HZ	HX

HX: X handwheel selecting signal
HZ: Z handwheel selecting signal

G0019	RT	*	MP2	MP1	*	*	*	*

MP1: Handwheel override signal MP1

MP2: Handwheel override signal MP2

RT: Rapid feedrate selecting signal in JOG mode

G0028	*	*	*	*	*	GR2	GR1	*

GR2：Gear selecting signal

GR1：Gear selecting signal

G0029		*SSTP		SAR				

*SSTP：Spindle speed reaching signal

SAR：Spindle stopping signal

G0030	SOV7	SOV6	SOV5	SOV4	SOV3	SOV2	SOV1	SOV0

SOV0: Spindle override signal

SOV1: Spindle override signal

SOV2: Spindle override signal

SOV3: Spindle override signal

SOV4: Spindle override signal

SOV5: Spindle override signal

SOV6: Spindle override signal

SOV7: Spindle override signal

G0032	R08I	R07I	R06I	R05I	R04I	R03I	R02I	R01I

R01I: Spindle speed instruction signal R01I

R02I: Spindle speed instruction signal R02I

R03I: Spindle speed instruction signal R03I

R04I: Spindle speed instruction signal R04I

R05I: Spindle speed instruction signal R05I

R06I: Spindle speed instruction signal R06I

R07I: Spindle speed instruction signal R07I

R08I: Spindle speed instruction signal R08I

G0033	SIND	SGN	*	*	R12I	R11I	R10I	R09I

R09I: Spindle speed instruction signal R09I

R10I: Spindle speed instruction signal R10I

R11I: Spindle speed instruction signal R11I

R12I: Spindle speed instruction signal R12I

SGN:Selecting signal of spindle speed instruction polarity

SIND:Spindle speed instruction selecting signal

G0043	ZRN	*	DNC1	*	*	MD4	MD2	MD1

MD1: Current operating mode selection

MD2: Current operating mode selection

MD3: Current operating mode selection

DNC1：DNC mode selecting signal

ZRN: Current operating mode selection

G0044	HDT	*	*	*	*	*	MLK	BDT

BDT: Optional block jumping signal

MLK: Machine locking signal

HDT:Manual tool change signal

G0046	DRN	*	*	*	KEY1	*	SBK	*

SBK: Single block signal

KEY1: Memory protecting signal

DRN: Dry run signal

G0053	CDZ	SMZ	*	*	*	*	*	*

SMZ: Error detection signal

CDZ: Chamfering signal

G0054	UI07	UI06	UI05	UI04	UI03	UI02	UI01	UI00

UI00: Custom macro inputting signal UI00

UI01: Custom macro inputting signal UI01

UI02: Custom macro inputting signal UI02

UI03: Custom macro inputting signal UI03

UI04: Custom macro inputting signal UI04

UI05: Custom macro inputting signal UI05

UI06: Custom macro inputting signal UI06

UI07: Custom macro inputting signal UI07

G0055	UI15	UI14	UI13	UI12	UI11	UI10	UI09	UI08

UI15: Custom macro inputting signal UI15

UI14: Custom macro inputting signal UI14

UI13: Custom macro inputting signal UI13

UI12: Custom macro inputting signal UI12

UI11: Custom macro inputting signal UI11

UI10: Custom macro inputting signal UI10

UI09: Custom macro inputting signal UI09

UI08: Custom macro inputting signal UI08

G0061	*	*	RGTSP2	RGTSP1	*	*	RGTAP	*

RGTAP: Rigid tapping signal

RGTSP1: Spindle selecting in rigid tapping

RGTSP2: Spindle selecting in rigid tapping

G0070	MRDYA	*	*	*	*	*	*	*

MRDYA: Machine ready signal

G0074	*	*	SFRB	SRVB	*	*	*	*

SRVB:CW rotating signal in rigid tapping

SFRB:CCW rotating signal in rigid tapping

G0100								
	*	*	*	*	+J4	+J3	+J2	+J1

+J1:Feedrate axis and direction selecting signal (+J1)
+J2:Feedrate axis and direction selecting signal (+J2)
+J3:Feedrate axis and direction selecting signal (+J3)
+J4:Feedrate axis and direction selecting signal (+J4)

G0102								
	*	*	*	*	-J4	-J3	-J2	-J1

-J1:Feedrate axis and direction selecting signal (-J1)
-J2:Feedrate axis and direction selecting signal (-J2)
-J3:Feedrate axis and direction selecting signal (-J3)
-J4:Feedrate axis and direction selecting signal (-J4)

G0198								
	*	*	*	*	NPOS4	NPOS3	NPOS2	NPOS1

NPOS1:Neglecting signal NPOS1 of position display
NPOS2:Neglecting signal NPOS2 of position display
NPOS3:Neglecting signal NPOS3 of position display
NPOS4:Neglecting signal NPOS4 of position display

G0200								
	*	*	*	*	*	*	*	SPD

SPD:Spindle jogging signal

G0201								
	NT07	NT06	NT05	NT04	NT03	NT02	NT01	NT00

NT00: Current tool number NT00
NT01: Current tool number NT01
NT02: Current tool number NT02
NT03: Current tool number NT03
NT04: Current tool number NT04
NT05: Current tool number NT05
NT06: Current tool number NT06
NT07: Current tool number NT07

G0202								
	NT15	NT14	NT13	NT12	NT11	NT10	NT09	NT08

NT08: Current tool number NT08
NT09: Current tool number NT09
NT10: Current tool number NT10
NT11: Current tool number NT11
NT12: Current tool number NT12
NT13: Current tool number NT13
NT14: Current tool number NT14
NT15: Current tool number NT15

G0203								
	NT23	NT22	NT21	NT20	NT19	NT18	NT17	NT16

NT16: Current tool number NT16
NT17: Current tool number NT17
NT18: Current tool number NT18
NT19: Current tool number NT19

NT20: Current tool number NT20

NT21: Current tool number NT21

NT22: Current tool number NT22

NT23: Current tool number NT23

G0204	NT31	NT30	NT29	NT28	NT27	NT26	NT25	NT24

NT24: Current tool number NT24

NT25: Current tool number NT25

NT26: Current tool number NT26

NT27: Current tool number NT27

NT28: Current tool number NT28

NT29: Current tool number NT29

NT30: Current tool number NT30

NT31: Current tool number NT31

5.2.5 Address A（information-asking for signal， defined by standard PLC）

地址	内 容
A0000.0	Tool change time is too long
A0000.1	Alarm of tool being not in-position at the end of tool change
A0000.2	Alarm of unfinished tool change
A0001.0	Tailstock function is invalid and M10、M11 can’t be executed
A0001.1	Tailstock can’t be receded under spindle rotating
A0001.4	Cycle start enabling is closed and cycle can’t be started
A0001.5	Spindle start enabling is closed and spindle can’t be started
A0002.0	Safeguard opening alarm
A0002.1	Low pressure alarm of chuck
A0002.3	Chuck can’t be unclamped under spindle rotating
A0002.4	Alarm of invalid clamping in-position signal under spindle rotation
A0002.5	Spindle can’t be started when clamping in-position signal of chuck is invalid
A0002.6	Spindle can’t be started when chuck is unclamped
A0003.0	Chuck function is invalid and M12、M13 can’t be executed
A0004.0	Illegal M code
A0004.1	Spindle jogging is invalid under non-analogous spindle
A0004.2	Setting error for M03 and M04
A0005.0	Setting error for No.164、168、172 parameters

5.3 PLC DATA

5.3.1 Timer Address（T， defined by standard PLC）

Address	**Meaning**
T0002	Timing for M3，4，5，8，9，10，11，32，33 executed

Address	Meaning
T0003	Timing for liberation/jogging output
T0004	Timing for from TL+ ceased to TL- output
T0005	Timing for TL- output
T0006	Timing for S code executed
T0007	Timing for M01，02，30executed
T0020	Timing for tool change
T0021	Delay for M05 output
T0022	Timing for M05 ceased to SPZD output
T0023	Timing for SPZD output
T0025	Timing for closing former gear-shift time
T0026	Timing for new gear-shift output to FIN output
T0027	Timing for spindle CW jogging
T0028	Timing for spindle CCW jogging
T0040	Timing for low pressure alarm
T0050	Timing for M12，M13 executed

5.3.2 Counter Address（C，defined by standard PLC）

Address	Meaning
C0001	Counting for rapid override in MDI panel decreased
C0002	Counting for rapid override in MDI panel increased
C0003	Counting turn for coolant input signal
C0004	Counting for feedrate override in MDI panel decreased
C0005	Counting for feedrate override in MDI panel increased
C0006	Counting for spindle override in MDI panel decreased
C0007	Counting for spindle override in MDI panel increased
C0008	Counting turn for lubrication input signal
C009	Counting turn for manual rapid traverse input signal
C0010	Counting turn for auxiliary function locked input signal
C0011	Counting turn for machine locked input signal
C0012	Counting turn for dry run input signal
C0013	Counting turn for single block input signal
C0014	Counting turn for optional block jumping input signal
C0015	Counting turn for spindle jogging input signal
C0016	Counting for manual feed in MDI panel decreased
C0017	Counting for manual feed in MDI panel increased
C0018	Counting turn for chuck control input signal
C0019	Counting turn for tailstock control input signal
C0020	Counting for two time reset under alarm of unfinished tool change

5.3.3 Counter Pre-setting Value Address（DT，defined by standard PLC）

Address	Meaning
DT000	Occupied by CNC and modified by No.65 data parameter

Book3 connection

Address	Meaning
DT001	Occupied by CNC and modified by No.66 data parameter
DT002	Occupied by CNC and modified by No.69 data parameter
DT003	Occupied by CNC and modified by No.76 data parameter
DT004	Occupied by CNC and modified by No.78 data parameter
DT005	Occupied by CNC and modified by No.80 data parameter
DT006	Occupied by CNC and modified by No.81 data parameter
DT007	Occupied by CNC and modified by No.82 data parameter
DT008	Occupied by CNC and modified by No.83 data parameter
DT009	Occupied by CNC and modified by No.85 data parameter
DT010	Occupied by CNC and modified by No.87 data parameter
DT011	Occupied by CNC and modified by No.89 data parameter
DT012	Occupied by CNC and modified by No.108 data parameter
DT013	Occupied by CNC and modified by No.112 data parameter
DT021	Spindle stopped and chuck enabling delayed

5.3.4 Counter Pre-setting Value Address（DC，defined by standard PLC）

（unused）

Chapter6   STORED PITCH ERROR COMPENSATION

6.1   FUNCTION DESCRIPTION

Stored pitch error compensation function of GSK980TD can reduce the accuracy error caused by the error screw pitch.

6.2   SPECIFICATION

1、 The compensation value is related with compensation origin、compensation interval、compensation point and moving direction etc.
2、 Take the reference point as compensation origin and specify the parameters with compensation value on each interval of each axis.
3、 Compensation point number is 256.
4 、Compensation range of each point is from 0 to±255um.
5 、Compensation interval is from 1000 to 9999999um.
6 、Compensation value of point N is decided by the machine error between point N and point N-1.
7、 Factual compensation interval is decided by the max. compensation range and machine travel.
8 、The setting method is same as the parameters setting of CNC，please read OPERATION MANUAL for details.

6.3   PARAMETER SETTING

6.3.1   Pitch Error Compensation

state parameter

0	0	3			PEC					

Bit5 is 1：pitch error compensation is valid
is 0：pitch error compensation is invalid

6.3.2   Compensation Origin

Compensation origin is the compensation starting position at reference point specified by No.98、No.99 parameter and 255 position points can be set for each axis at most.

data parameter

0	9	8	Position No. of pitch error compensation(X)

0	9	9	Position No. of pitch error compensation(Z)

Book3 connection

6.3.3 Compensation Interval

1	0	2	Pitch error interval of X axis
1	0	3	Pitch error interval of Z axis

Input unit is 0.001mm and setting range is from 1000 to 9999999.

6.3.4 Compensation Value

Each compensation value is diameter input and input unit is 0.001mm.Please set compensation value as the following table.

Sequence Number	X	Z
000	…	…
001	5	-2
002	-3	4
…	…	…
255	…	…

6.4 NOTES for SETTING COMPENSATION VALUE

1、Pitch error compensation can be specified only in the second level authority.

2、No compensation for zero compensation interval.

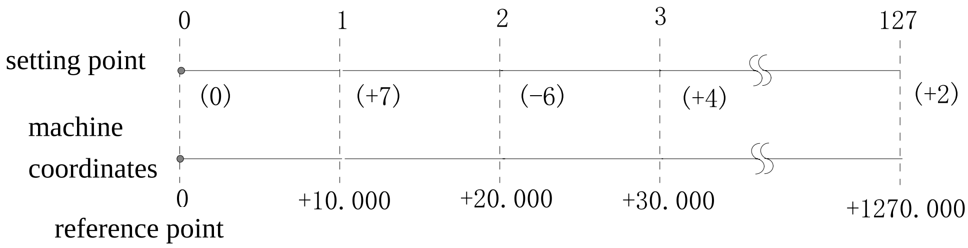
3、When specifying the parameters，compensation can be realized correctly only after returning reference point.

Book3 connection

6.5 EXAMPLE for SETTING COMPENSATION PARAMETERS

1、 When No.99 parameter is 0 and No.103 parameter is 10.00

Under this condition，the compensation value of N segment can be set in the No.000＋N position.Because the reference point is compensation origin corresponding with No.001 compensation value，pitch error compensation can be realized in the positive moving direction from the reference point.



In above diagram，No.000 position is compensation origin and No.001 position is the point 10.000 away from the origin in positive moving direction，and the rest may be deduced by analogy.That is to say，compensation value at N point is set for moving from（N-1）×compensation interval to N×compensation interval.

Example as following：

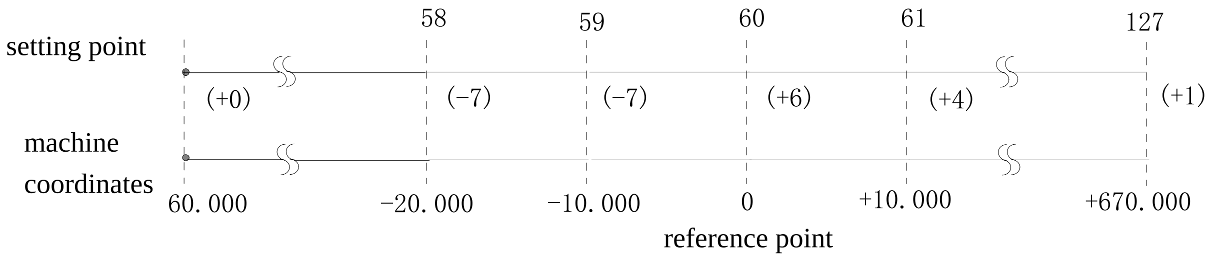
compensation range	compensation value
0～10.000	+7
10.000～20.000	-6
20.000～30.000	+4

machine coordinates	compensation parameter No.	compensation value	current pulse No. before compensation	current pulse No. after compensation
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	…		

In fact，the compensation value for from the origin to +30.000 is that (+7)+(-6)+(+4).

2、 when No.099 parameter is 60 and No.103 parameter is 10.000

Under this condition，the compensation value of N segment can be set in the No.060＋N position in moving positive direction and the compensation value of N segment can be set in the No.061－N position in moving negative direction. Therefore，pitch error compensation can be realized in two directions.



In above diagram，No.060 position is compensation origin and No.061 position is the point 10.000 away from the origin in positive moving direction，and No.059 position is the point 10.000 away from the origin in negative moving direction.That is to say，compensation value at N point is set for moving from（N-61）×compensation interval to（N－60）×compensation interval.

Example as following：

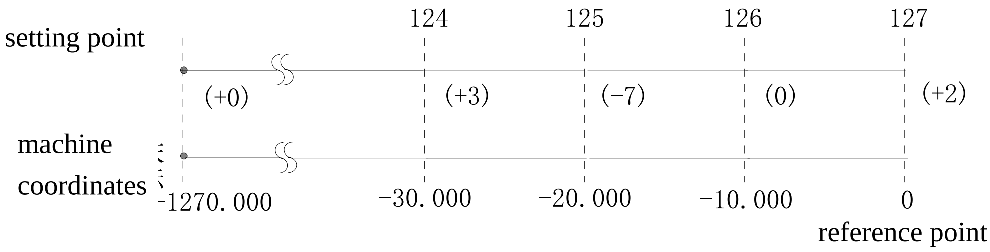
compensation range	compensation value
0～10.000	+4
—10.000～0	+6
—20.000～—10.000	—7
—30.000～—20.000	—7

machine coordinates	compensation parameter no.	value	current pulse no. before compensation	current pulse no. after compensation
-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	…		

In fact，the compensation value for from –30.000 to +10.000 is that (-7)+(-7)+(+6)+(+4).

3、 when No.099 parameter is 127 and No.103 parameter is 10.000

Under this condition，the compensation value of N segment can be set in the No.128-N position.Because the reference point is compensation origin corresponding with No.127 compensation value，pitch error compensation can be realized in the negative moving direction from the reference point.



In above diagram，No.127 position is compensation origin and No.126 position is the point 10.000 away from the origin in negative moving direction，and the rest may be deduced by analogy.That is to say，compensation value at N point is set for moving from（N-128）×compensation interval to（N-127）×compensation interval.

Example as following：

compensation range	compensation value
0～—10.000	+2
—20.000～——10.000	0
—30.000～—20.000	—7
—40.000～—30.000	+3

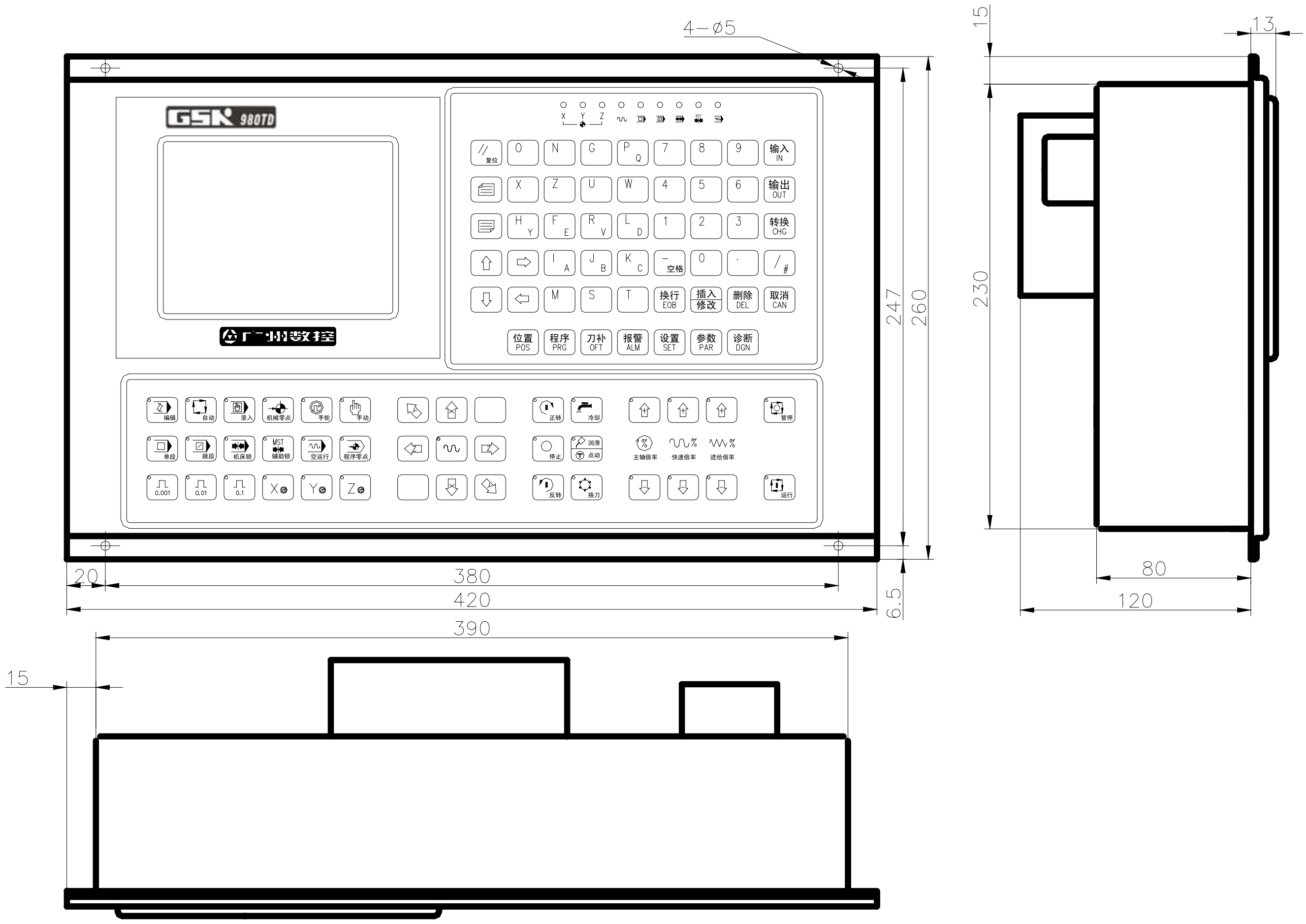
machine coordinates	compensation parameter no.	value	current pulse no. before com.	current pulse no. after com.
reference point 0			0	0
-10.000	127	2	10000	10002
-20.000	126	0	20000	20002
-30.000	125	-7	30000	29995
-40.000	124	3	40000	39998

In fact，the compensation value for from –40.000 to the origin is that (+3)+(-7)+(0)+(+2).

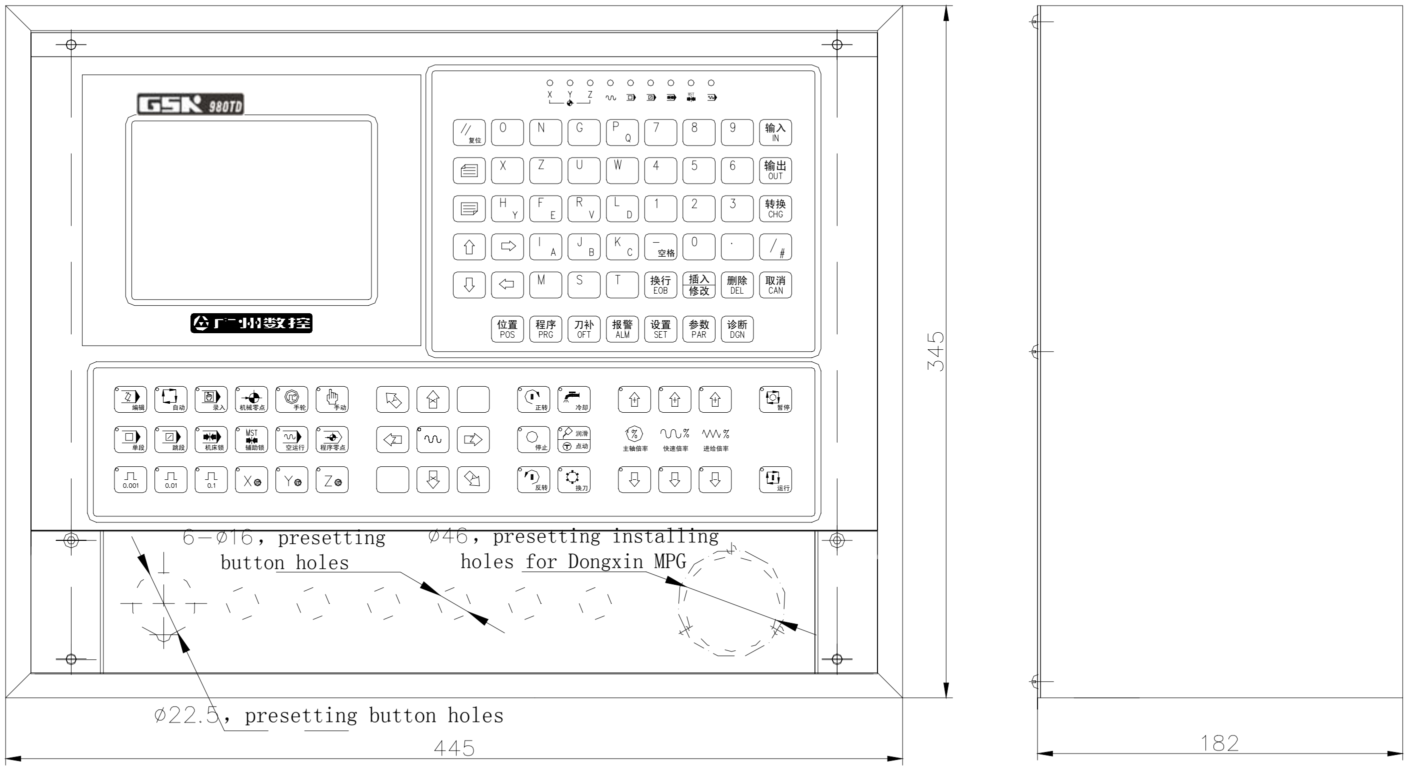
BOOK APPENDIX

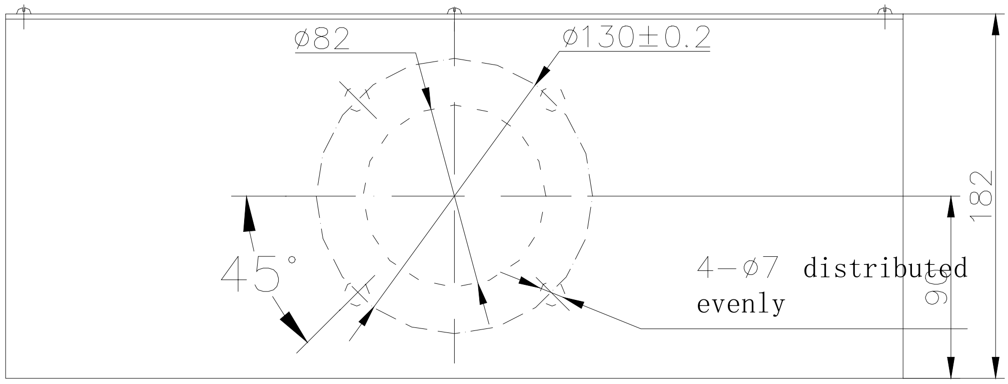
APPENDIX

Appendix Ⅰ Outline Dimension of GSK980TD



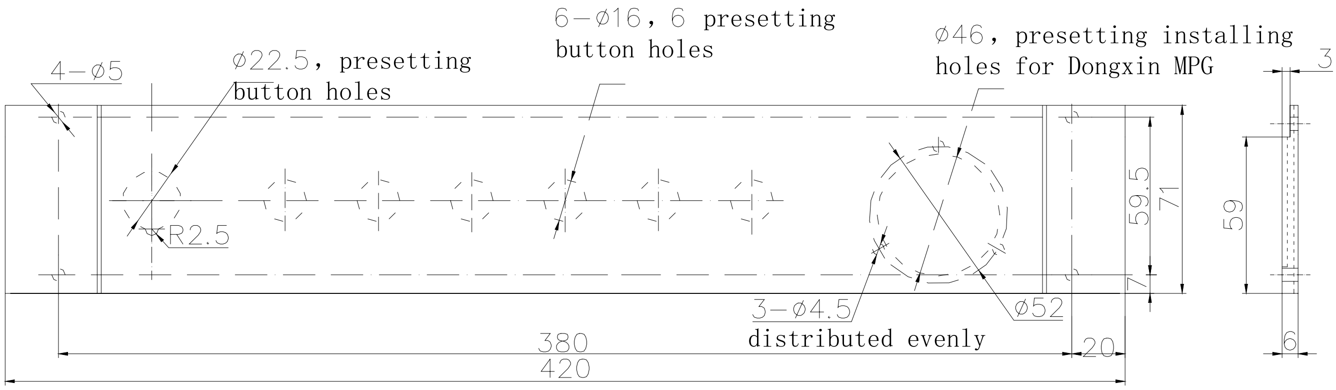
Appendix Ⅱ Outline Dimension of GSK980TD-B





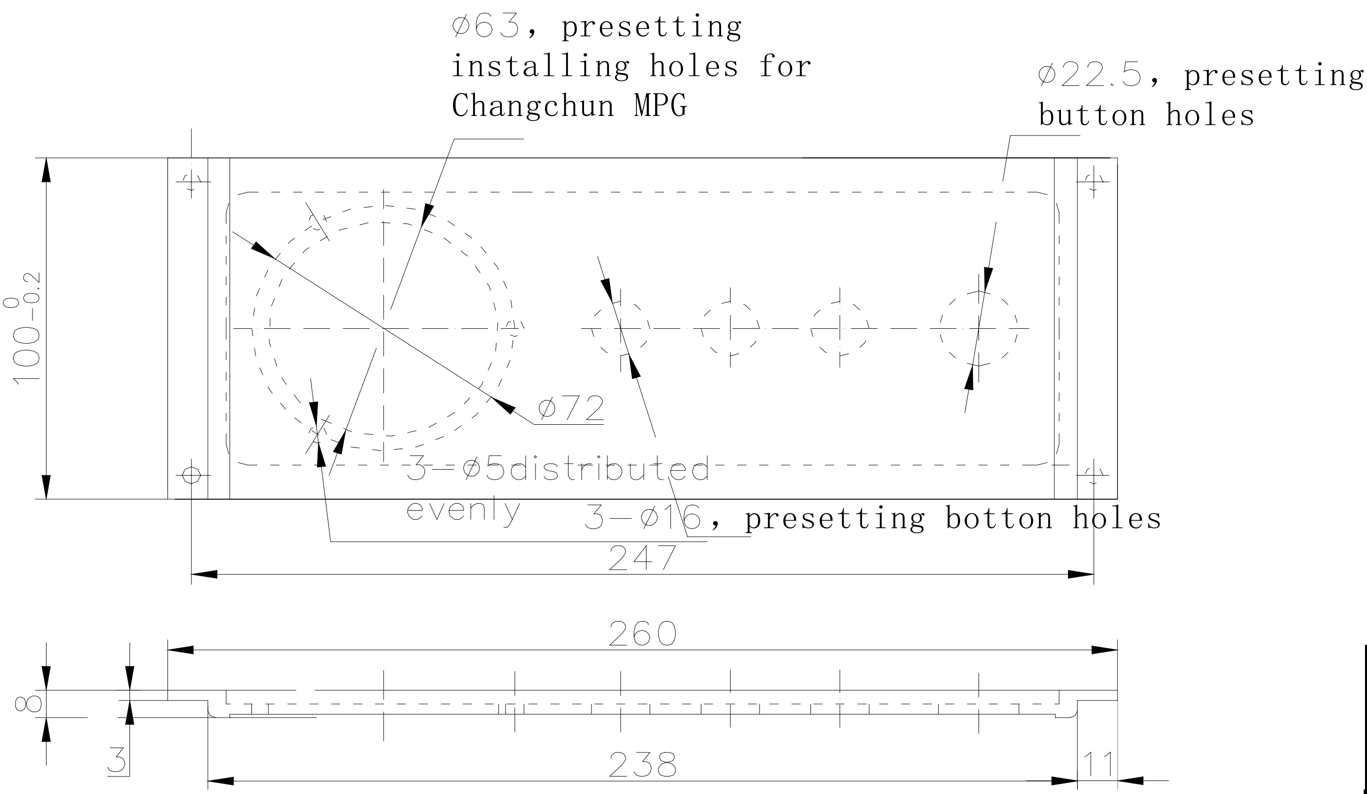
Appendix Ⅲ Outline Dimension of Accessional Panel AP01

AP01：aluminum alloy 420×71（mm），can be fixed below the panel and the outline dimension as following：



Appendix Ⅳ Outline Dimension of Accessional Panel AP02

AP02：aluminum alloy100×260（mm），can be fixed at the side of panel and the outline dimension as following：



Appendix Ⅴ Standard Parameter

Bit par.	Par.1 (test)	Par.2 (step)	Par.3 (servo)	User par. (backup)
001	00000000	00000000	00000000	
002	00100010	00100010	00100010	
003	00110000	00110000	00110000	
004	01000000	00000000	00000000	
005	00010011	00010011	00010011	
006	00000000	00000000	00000000	
*007	00001000	00000000	00000000	
008	00000011	00000011	00000011	
009	00000000	00000011	00000011	
010	00000000	00000111	00001111	
011	10000110	10000010	00000010	
012	01101011	10101011	10101011	
013	00000000	00000000	00000000	
014	00000011	00000011	00000011	
164	11100101	00000000	00000000	
168	00000000	00000000	00000000	
172	00000000	00100000	00100000	
173	00000000	00000000	00000000	
174	00001000	00001000	00001000	
*175	00000000	00000000	00000000	
176	00000000	00000000	00000000	
177	00000000	00000000	00000000	
178	00000000	00000000	00000000	
179	00000000	00000000	00000000	
180	00000000	00000000	00000000	
181	00000000	00000000	00000000	
182	00000000	00000100	00000100	
183	00000000	00000000	00000000	
184	00000000	00000000	00000000	
185	00000000	00000000	00000000	

Data par.	Par.1 (test)	Par.2 (step)	Par.3 (servo)	User par. (backup)
015	1	1	1	
016	1	1	1	
017	1	1	1	
018	1	1	1	
019	10	5	5	
020	0	0	0	
021	0	0	0	
022	3800	2500	3800	
023	7600	5000	7600	
*024	200	240	100	
*025	200	240	100	
*026	100	200	100	
*027	8000	8000	8000	
*028	500	50	200	

Data par.	Par.1 (test)	Par.2 (step)	Par.3 (servo)	User par. (backup)
*029	100	160	100	
*030	10	10	10	
031	0	0	0	
032	400	400	400	
033	200	200	200	
034	0	0	0	
035	0	0	0	
036	0	0	0	
037	9999	9999	9999	
038	9999	9999	9999	
039	9999	9999	9999	
040	9999	9999	9999	
*041	300	200	400	
042	10	10	10	
043	99	99	99	
044	115200	115200	115200	
045	9999999	9999999	9999999	
046	9999999	9999999	9999999	
047	-9999999	-9999999	-9999999	
048	-9999999	-9999999	-9999999	
049	0	0	0	
050	0	0	0	
051	0	0	0	
052	0	0	0	
053	0	0	0	
054	0	0	0	
055	0	0	0	
056	0	0	0	
057	0	0	0	
058	0	0	0	
059	0	0	0	
060	0	0	0	
061	0	0	0	
062	0	0	0	
063	0	0	0	
064	0	0	0	
065	1000	1000	1000	
066	1000	1000	1000	
067	100	100	100	
068	0	0	0	
069	0	0	0	
070	1024	1024	1024	
071	32	32	32	
072	0	0	0	
073	0	0	0	
074	0	0	0	
075	0	0	0	
076	1000	1000	1000	
077	0	0	0	
078	15000	15000	15000	
079	0	0	0	

Data par.	Par.1 (test)	Par.2 (step)	Par.3 (servo)	User par. (backup)
080	500	500	500	
081	500	500	500	
082	0	0	0	
083	500	500	500	
084	4	4	4	
085	1000	1000	1000	
086	0	0	0	
087	0	0	0	
088	0	0	0	
089	50	50	50	
090	0	0	0	
091	0	0	0	
092	0	0	0	
093	0	0	0	
094	0	0	0	
095	0	0	0	
096	0	0	0	
097	0	0	0	
098	0	0	0	
099	0	0	0	
100	0	0	0	
101	0	0	0	
102	10000	10000	10000	
103	10000	10000	10000	
104	0	0	0	
105	0	0	0	
106	0	0	0	
107	0	0	0	
108	3000	3000	3000	
109	40	40	40	
110	1	1	1	
111	1	1	1	
112	0	0	0	
113	7600	5000	7600	
114	0	0	0	
115	0	0	0	
116	0	0	0	
117	0	0	0	
118	0	0	0	
119	3	3	3	
120	0	0	0	
121	0	0	0	
122	0	0	0	
123	0	0	0	
124	0	0	0	

Note: the parameters with “*” will infect acceleration and deceleration characteristic, and user needs to adjust them according to system configuration and load characteristic.

Appendix Ⅵ Alarm Table

1、CNC alarm

Number	Content	Troubleshooting
000	Emergent stop alarm and ESP open circuit	Resume ESP signal input and press【Reset】key
001	There are no part programs or cannot open part programs	Press 【Reset】 key and modify programs
002	G instruction values are negative or with decimals	Press 【Reset】 key and modify programs
003	Characters are less than 2 or more than 11 for one word	Press 【Reset】 key and modify programs
004	Address error (address is A~Z)	Press 【Reset】 key and modify programs
005	Illegal instruction value	Press 【Reset】 key and modify programs
006	Block numbers are negative or with decimals	Press 【Reset】 key and modify programs
007	Illegal G instructions	Press 【Reset】 key and modify programs
008	Execute G96 when the spindle analog voltage control is invalid	Press 【Reset】 key and modify programs or parameter No.001
009	Command movement distance when G instructions in 00 and 01 groups are not input and there are invalid instructions in 01 group	Press 【Reset】 key and modify programs
010	There are the same addresses in one block	Press 【Reset】 key and modify programs
011	There are more than 20 words in one block	Press 【Reset】 key and modify programs
012	Instruction values exceed their valid range	Press 【Reset】 key and modify programs
013	Input S instructions except for S00~S99 when the spindle analog voltage control is invalid	Press 【Reset】 key and modify illegal S instructions
014	Input G instructions in 00 and 01 groups in one block	Press 【Reset】 key and modify programs
015	Execute M instructions for spindle automatic gear shifting when the spindle analog voltage control is invalid	Press 【Reset】 key and modify programs
016	Tool offset numbers exceed their valid range(0~32)	Press 【Reset】 key and modify programs or parameters
017	Tool number exceeds the range of No.084 parameter	Press 【Reset】 key and modify programs or No.084 parameters
018	Data in G02 or G03 cannot form a correct arc	Press 【Reset】 key and modify programs
030	Movement distance in X direction isn't zero in G33	Press 【Reset】 key and modify programs
031	G02 or G03 has changed monotony of corresponding coordinates in G71~G73	Press 【Reset】 key and modify programs
032	Absolute value of R is more than that of U/2 in G90, G92	Press 【Reset】 key and modify programs
033	Absolute value of R is more than that of W in G94	Press 【Reset】 key and modify programs
034	There are more than 100 blocks in G70~G73	Press 【Reset】 key and modify programs
035	Ns and Nf are reversed each other in G70~G73	Press 【Reset】 key and modify programs
036	There is no Ns or Nf or they exceed their allowed range in G70~G73	Press 【Reset】 key and modify programs
037	There is no Ns or Nf in G70~G73	Press 【Reset】 key and modify programs
038	Single infeed exceeds its allowed range in G71 or G72	Press 【Reset】 key and modify programs

Number	Content	Troubleshooting
039	Single tool retraction exceeds its allowed range in G71 or G72	Press 【Reset】 key and modify programs
040	Total cutting travel exceeds its allowed range in G73	Press 【Reset】 key and modify programs
041	Cycle times is less than 1 or more than 99999 in G73	Press 【Reset】 key and modify programs
042	Single tool retraction R(e) exceeds its allowed range in G74 or G75	Press 【Reset】 key and modify programs
043	The tool retraction is negative at the end of cutting in G74 or G75	Press 【Reset】 key and modify programs
044	Single cutting value exceeds its allowed range in X or Z direction in G74 or G75	Press 【Reset】 key and modify programs
045	Starting point of cutting taper thread is between thread starting point and its end point in G76	Press 【Reset】 key and modify programs
046	Min. cutting value exceeds its allowed range in G76	Press 【Reset】 key and modify programs
047	Finishing allowance exceeds its allowed range in G76	Press 【Reset】 key and modify programs
048	Tooth height is less than finishing allowance or 0 in G76	Press 【Reset】 key and modify programs
049	Cycle times exceeds its allowed range in G76	Press 【Reset】 key and modify programs
050	Chamfer exceeds its allowed range in G76	Press 【Reset】 key and modify programs
051	Angle of tool nose exceeds its allowed range in G76	Press 【Reset】 key and modify programs
052	Movement distance in X, Z direction in G76 is zero	Press 【Reset】 key and modify programs
053	There is no specified tooth height P in G76	Press 【Reset】 key and modify programs
054	There is no the first cutting depth Q or Q is 0 or Q is not input	Press 【Reset】 key and modify programs
055	Call subprograms in G70~G73	Press 【Reset】 key and modify programs
056	Ns does not command G00 or G01 in G70~G73	Press 【Reset】 key and modify programs
057	X value is not specified in the first block or the movement is 0 in G71	Press 【Reset】 key and modify programs
058	Z value is not specified in the first block or the movement is 0 in G72	Press 【Reset】 key and modify programs
059	Z value is not specified in G74	Press 【Reset】 key and modify programs
060	Q value is 0 or is not input in G74	Press 【Reset】 key and modify programs
061	X value is not specified in G75	Press 【Reset】 key and modify programs
062	P value is 0 or is not input in G75	Press 【Reset】 key and modify programs
063	Initial blocks are employed with the forbidden G instructions in G70~G73	Press 【Reset】 key and modify programs
064	End blocks are employed with the forbidden G instructions in G70~G73	Press 【Reset】 key and modify programs
065	Execute G70~G73 in MDI mode	G70~G73 can't be executed in MDI mode and press 【Reset】 key
095	Subprogram numbers are not input or are illegal when M98 calls them	Press 【Reset】 key and modify programs
096	Layers of nested subprograms are more than 4	Press 【Reset】 key and modify programs
097	Calling programs in M98 is the current one(main program)	Press 【Reset】 key and modify programs
098	Use M98 or M99 in MDI mode	Press 【Reset】 key and modify programs
099	Use M98 or M99 in the state of tool radius compensation	Press 【Reset】 key and modify programs

Number	Content	Troubleshooting
101	Operation values of H11, H12, H13, H25 are not binary in G65	Press 【Reset】 key and modify programs
102	Operation value of H24 is more than 1023 in G65	Press 【Reset】 key and modify programs
103	Denominator is 0 for division operation in G65	Press 【Reset】 key and modify programs
104	G65 commands illegal H instruction	Press 【Reset】 key and modify programs
105	Macro variable number of G65 is illegal(error)	Press 【Reset】 key and modify programs
106	Macro variable P is not commanded or P value is zero in G65	Press 【Reset】 key and modify programs
107	Variable Q is not commanded or Q value is zero when H instructions except for H80 or H99 are commanded	Press 【Reset】 key and modify programs
108	Do not command variable R or R is illegal	Press 【Reset】 key and modify programs
109	P instruction value isn't variable in G65	Press 【Reset】 key and modify programs
110	Number with H21 in G65 is negative	Press 【Reset】 key and modify programs
111	H99 user alarm number in G65 exceeds its range	Press 【Reset】 key and modify programs
112	Block number of macro instruction (G65)jumping or M99 returning exceeds their range	Press 【Reset】 key and modify programs
113	There is no block number for block jumping or subprogram returning	Press 【Reset】 key and modify programs
251	Mistake in programming causes an error of tool nose radius compensation	Press 【Reset】 key and modify programs
252	Mistake in programming causes an end point of arc machining is not on the arc in the course of tool nose radius compensation	Press 【Reset】 key and modify programs
253	Mistake in programming causes there are the same coordinates for two neighbouring points not to execute tool nose radius compensation	Press 【Reset】 key and modify programs
254	Mistake in programming causes there are the same coordinates between center point and starting point of arc not to execute tool nose radius compensation	Press 【Reset】 key and modify programs
255	Mistake in programming causes there are the same coordinates between center point and end point of arc not to execute tool nose radius compensation	Press 【Reset】 key and modify programs
256	Arc radius is less than that of tool nose to cause not to execute tool nose radius compensation	Press 【Reset】 key and modify programs
257	Mistake in programming causes there is not intersection of two arcs of current tool radius in the course of tool nose radius compensation	Press 【Reset】 key and modify programs
258	Specify one arc instruction as executing tool nose radius compensation	Press 【Reset】 key and modify programs
259	Specify one arc instruction as canceling tool nose radius compensation	Press 【Reset】 key and modify programs

Number	Content	Troubleshooting
260	There is excessive cutting as checking tool nose radius compensation	Press 【Reset】 key and modify programs
261	Mistake in programming causes there is not intersection between straight line and arc of current tool radius in the course of tool nose radius compensation	Press 【Reset】 key and modify programs
262	Mistake in programming causes there is not intersection between arc and straight line of current tool radius in the course of tool nose radius compensation	Press 【Reset】 key and modify programs
301	Parameter switch has been on	Press【Reset】and【Cancel】key simultaneously or close parameter switch
302	CNC initialization is failure	Power off and restart
303	Cannot open part programs	Reset or power on again
304	Fail to save part programs	Reset or power on again
305	Total lines of part program exceed its range(69993)	Reset
306	Illegal dictates have been input	Reset and input correct dictates
307	Memory capacity is not enough	Reset and delete excrescent part programs
308	Program numbers exceed the range	Reset or power on again
309	Editing macro program is forbidden under its current operation authority	Reset and modify operation password
310	Cannot open PLC programs (ladder)	Download PLC programs again(ladder)
311	Edit software version of PLC programs (ladder) is inconsistent	Update software version of GSKCC
312	First-grade PLC program is too long	Modify PLC programs(ladder)
313	The edit keyboard or operator panel is failure	Press【Reset】or【Cancel】key
314	The memorizer is failure, check it or power on again	Press【Reset】key, check it and power on again
401	The program reference point is not specified	Press 【Reset】 key and set program reference point with G50
402	Max. spindle speed at some gear is not specified and check No.037~No.040	Press【Reset】key and modify the parameter corresponding to the current gear
403	Run speed is too rapid	Press 【Reset】 key and modify program or parameter
404	Feedrate is cancelled owing to spindle stopping	Press 【Reset】 key and check the spindle
405	Spindle speed is too low when machining thread	Press 【Reset】 key and change the spindle speed
406	Spindle speed is too high when machining thread	Press 【Reset】 key and change the spindle speed
407	Fluctuation of spindle speed is excessive when machining thread	Press 【Reset】 key and modify No.106 parameter

Appendix

Number	Content	Troubleshooting
411	Software overtravel in X positive direction	Press 【Reset】 key and move X axis in negative direction
412	Software overtravel in X negative direction	Press 【Reset】 key and move X axis in positive direction
413	Software overtravel in Z positive direction	Press 【Reset】 key and move Z axis in negative direction
414	Software overtravel in Z negative direction	Press 【Reset】 key and move Z axis in positive direction
421	Driver is not ready in X direction	Press 【Reset】 key after fault clearance
422	Driver is not ready in Z direction	Press 【Reset】 key after fault clearance
426	Driver alarms in X direction	Press【Reset】 key after fault clearance
427	Driver alarms in Z direction	Press【Reset】 key after fault clearance
440	Emergent stop is failure	Power on again

2、Operation prompt

Content	Operation with prompt	Remark
Memory full	Program number exceeds 384 or total memory capacity exceeds 6144KB	All prompts will be displayed in bottom left corner of display interface
Error data	Input data is out of range	
Block exceeding	Input block exceeds 255 characters	
Unallowed input	Input data includes unrecognizable characters	
Serial interface not connected	Doing communication under unconnected serial interface	
Communication error	Error data transferring	
Fail to delete blocks	Not find object characters during deleting blocks	
Fail to search	Not find object characters in cursor searching up or down	
Line exceeding	Limit to the max. lines （69993）of part programs and forbid to add lines	
Illegal G	Illegal dictates have been input	
File not existed	Not search object part program	
File existed	Files with same name exist when saving or renaming file	
Modify in parameter page	When modifying parameters in diagnosis page	

3、PLC alarm（defined in standard PLC）

Number	Content	Address
1000	Tool change time is too long	A0000.0
1001	Alarm of toolpost not in-position at the end of tool change	A0000.1
1002	Alarm of tool change not finished	A0000.2
1008	Can’t execute M10 and M11 under invalid tailstock function	A0001.0
1009	Can’t retreating tailstock when spindle rotating	A0001.1
1012	Can’t start cycle when cycle start enabling is closed	A0001.4
1013	Can’t start spindle when spindle start enabling is closed	A0001.5

Number	Content	Address
1016	Alarm of protection door not closed	A0002.0
1017	Chuck low pressure alarm	A0002.1
1019	Can’t unclamp chuck when spindle rotating	A0002.3
1020	Alarm of invalid clamping in-position signal when spindle rotating	A0002.4
1021	Can’t start spindle when clamping in-position signal	A0002.5
1022	Can’t start spindle when chuck is unclamped	A0002.6
1024	Can’t execute M12 or M13 under invalid chuck function	A0003.0
1032	Illegal M code	A0004.0
1033	Spindle jogging function is invalid under invalid analog spindle	A0004.1
1034	Setting error for M03 and M04	A0004.2
1040	Setting error for No.164,168,172 parameters	A0005.0

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: sale1@gsk.com.cn

Tel: 86-20-81796410/81797922 Fax: 86-20-81993683

All specification and designs are subject to change without notice July. 2006/Edition 2

Suit for software version:V06.03.23 July. 2006/Printing 2