In this user manual we have tried to describe the matters concerning the operation of this CNC system to the greatest extent. However, it is impossible to give particular descriptions for all unnecessary or unallowable operations due to length limitation and products application conditions；Therefore, the items not presented herein should be regarded as “impossible” or “unallowable”.

Preface

Your Excellency,

We are honored by your purchase of this GSK988TA/988TA1/988TB Turning CNC System made by GSK CNC Equipment Co., Ltd.

This book describes GSK988TA/988TA1/988TB Turning Center CNC System, Programming and Operation （software version: V1.12）, and concretely introduces the programming and operations.

To ensure safe and effective running, please read this manual carefully before installation and operation.

Warning

Accident may occur by improper connection and operation！

This system can only be operated by authorized and qualified personnel.

Special caution:

The power supply fixed on/in the cabinet is exclusively used for the CNC system made by GSK.

It can't be applied to other purposes, or else it may cause serious danger!

Cautions

■ Delivery and storage

- Packing box over 6 layers in pile is unallowed.
- Never climb the packing box, stand on it or place heavy objects on it.
- Do not move or drag the products by the cables connected to it.
- Forbid collision or scratch to the panel and display screen.
- Avoid dampness, insolation and drenching.

■ Open-package inspection

- Confirm that the products are the required ones.
- Check whether the products are damaged in transit.
- Confirm that the parts in packing box are in accordance with the packing list.
- Contact us in time if any inconsistence, shortage or damage is found.

■ Connection

- Only qualified personnel can connect the system or check the connection.
- The system must be earthed, and the earth resistance must be less than 0.1Ω. The earth wire cannot be replaced by zero wire.
- The connection must be correct and firm to avoid any fault or unexpected consequence.
- Connect with surge diode in the specified direction to avoid damage to the system.
- Switch off power supply before plugging out or opening electric cabinet.

■ Troubleshooting

- Switch off power supply before troubleshooting or changing components.
- Check the fault when short circuit or overload occurs. Restart can only be done after troubleshooting.
- Frequent switching on/off of the power is forbidden, and the interval time should be at least 1 min.

Announcement

- This manual describes various possibilities as much as possible. However, operations allowable or unallowable cannot be explained one by one due to so many possibilities that may involve with, so the contents that are not specially stated in this manual shall be considered as unallowable.

Warning

- Before installing, connecting, programming and operating, please carefully read the product user manual and the manual from the machine tool manufacturer and strictly operate accordance with the regulations in the manual; otherwise, the product or the machine tool may be damaged, the workpiece may get rejected, even the personal injury may occur.

Caution

- Functions, technical indexes (such as precision and speed) described in this user manual are only for this system. Actual function deployment and technical performance of the machine tool are designed by the machine tool manufacturer, so function configuration and technical indexes are subject to the user manual from the machine tool manufacturer.

  Refer to the user manual from the machine tool manufacturer for function and meaning of each button on the machine panel.

All specifications and designs herein are subject to change without notice.

Safety Responsibility

Manufacturer's Responsibility

——Be responsible for the danger which should be eliminated and/or controlled on design and configuration of the provided CNC systems and accessories.

——Be responsible for the safety of the provided CNC systems and accessories.

——Be responsible for the provided message and advice for the users.

User's Responsibility

——Be responsible for being familiar with and mastering the safety operation procedures through training with the safety operation of the CNC system.

——Be responsible for the dangers caused by adding, changing or altering the original CNC systems and the accessories.

——Be responsible for the dangers caused by failing to observe the provisions in the manual for operation, adjustment, maintenance, installation and storage.

This manual is kept by the end user.

Thank you for supporting us in the use of GSK's products!

Contents

Programming

PROGRAMMING

Chapter 1 Programming Fundamental

1.1 Product Introduction

With 6 feed axes (including Cs axis), 3 spindles, GSK988TA/GSK988TA1/GSK988TB is a new product aiming at the slant CNC machine and turning center，connected with a servo and I/O unit by GSK-Link bus. Its matched servo motor uses a high-resolution absolute encoder to realize 0.1μm-level position precision and meet high-precision turning-milling compound machining.

With a network interface, GSK988TA/GSK988TA1/GSK988TB supports a remote monitor and file transmission, and meets requirements of a networked teaching and workshop management. It is the best choice of a slant CNC turning machining and turning center.

Fig.1-1 GSK988TA/TB appearance

•Technical characteristics

- ■6 feed axes（including Cs axis）, 3-axis link and 3 spindles to realize the turning, milling compound machining
- ■Code unit 1μm and 0.1μm, up to 100 m/min
- ■The servo drive and I/O unit use connection control of GSKLink bus
- ■Nested many PLC programs, PLC ladder on-line editing
- ■Part programs edited on the background
- ■Network interface, remote monitoring and file transmission
- ■USB interface, U disc file operation, system allocation and software upgrading

•Technical specifications

■Controllable axes

- ◆Max. controllable axes：6（including Cs axis）
- ◆Up to link axes：3
- ◆PLC controllable axis number：6 axes in each path

■Feed axis function

- ◆Least code unit：0.001mm and 0.0001mm (optional)
- ◆Least code range：±99999999× least code unit

- ◆Rapid traverse speed：max. 100m/min in 0.001mm code unit, max. 60 m/min in 0.0001mm code unit
- ◆Rapid override：F0, 25%, 50%, 100% real-timing tuning
- ◆Cutting feedrate：0.01 mm/min～60000 mm/min or 0.01 inch/min～4000 inch/min（G98:feed per minute）； 0. 01 mm/rev～500 mm/rev or 0. 01 inch/rev～9.99 inch/rev（G99: feed per rev）
- ◆Feedrate override：0～150% 16-level real-time tuning
- ◆Interpolation mode: linear, circular, thread, polar, cylindrical interpolation, rigid tapping and polygon interpolation.

■Thread function

- ◆ Thread type: constant pitch straight thread/taper thread/end thread, variable pitch straight thread/taper thread/end thread
- ◆ Thread head：1~99 heads
- ◆ Thread cutting： linear, exponential type (optional)
- ◆ Initial speed, termination speed and time of acceleration/deceleration set by the parameter

■Acceleration/deceleration function

- ◆ Cutting feed: linear, exponential (optional)
- ◆ Rapid traverse: linear type
- ◆ Initial speed, terminate speed, time of acceleration/deceleration set by the parameter

■Spindle function

- ◆ 3-channel spindle control supporting multi-spindle spindle control
- ◆ Spindle speed: spindle speed specified by S or PLC signal, its range: 0rpm～20000rpm
- ◆ Spindle override：50%～120% 8-level real-time tuning
- ◆ Spindle constant surface control
- ◆ Rigid tapping

■Tool function

- ◆ Tool length compensation（tool offset）：99 groups
- ◆ Tool wear compensation：99 groups of tool wear compensation data
- ◆ Tool nose radius compensation（C type）
- ◆ Tool life management
- ◆ Toolsetting mode: fixed-point toolsetting, trial-cutting toolsetting, reference point return toolsetting
- ◆ Offset execution mode: modifying coordinate mode, tool traverse mode

■Precision compensation

- ◆Backlash compensation: compensation range (-9999~9999)× check unit
- ◆Memory pitch error compensation：1024 compensation points，compensation point number of each is set by the parameter, each point compensation range (-700~700) × check unit

■ PLC function

- ◆ 13 basic codes, 30 functional codes
- ◆ PLC ladder on-line edit, real-time monitoring
- ◆ 2-level PLC program, up to 12000 steps, the 1st level program refresh period 8ms
- ◆ Many PLC programs（up to 16 programs），the current running PLC program can be selected

■I/O unit

- ◆ Rapid I/O：16 input/8 output interface
- ◆ Operation panel I/O：118 input/96 output interface
- ◆ Up to 4 GSKLink remote I/O interfaces，each I/O has 48 input interfaces and 32 output interfaces

■Human-computer interface

- ◆Display in Chinese, English and others
- ◆Two-dimensional tool path and solid graph displa
- ◆Servo state monitoring
- ◆Servo parameter on-line allocation
- ◆System debugging, servo debugging
- ◆Real-time clock
- ◆On-line help
- ◆Counter

■Operation management

- ◆Operation mode: Auto, Manual, Edit, MDI, DNC, MPG, Reference point return
- ◆Multi-level operation Authorization Management
- ◆Alarm log
- ◆Timed stop

■Program edit

- ◆Program capacity：32M, 10000 programs（including subprogram and macro program）
- ◆Edit mode: full-screen edit, part program edit on the background
- ◆Edit function：searching, modifying and deleting programs/blocks/words, copying/ deleting blocks
- ◆Program format: ISO code(A set of G code, G code system B), word without blank space, relative coordinates, absolute coordinate compound programming
- ◆Macro code: statement macro code program
- ◆Program call: macro program call with parameters, 12-level subprogram nesting
- ◆Aided programming: common used cycle codes using graphic aided programming
- ◆Drawing dimension input: direct input contour angle, intersection point not to be counted
- ◆Grammar check: executing the rapid grammar check for the program(do not run the program) after it has been edit
- ◆Path preview: do not run programs, use the path preview function to ensure the program path is correct

■Communication function

- ◆USB：U disc file operation, U disc file directly machining, upgrading PLC program and system software U disc
- ◆LAN：remote monitoring, network DNC machining, file transmission, remotely upgrading PLC program, system software

■Safety function

- ◆Emergency stop
- ◆Hardware travel limit
- ◆Many stored travel checks
- ◆Data backup and recover

1.2 CNC System 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, lubricating, cooling 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 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 codes 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 codes in user programs to control spindle start/stop, tool selections, cooling 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.

The system has simultaneously motion control and logic control function to control two axes of CNC machine tool to move, and has PLC function. Edit PLC programs (ladder diagram) according to requirements of input and output control of machine tool and then download them to GSK988TA/988TA1/988TB Turning Machine CNC system, which realizes the required electric control requirements of machine tool, is convenient to electric design of machine tool and reduces cost of CNC machine tool.

Softwares realizing CNC control function 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 when the 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 GSK988TA/988TA1/988TB Turning CNC System User Manual. Refer to Operation Manual of machine manufacturer about functions and operations of PLC control because the machine manufacturer may modify or edit PLC programs again

Programming is a course of workpiece contours, machining technologies, technology parameters and tool parameters being edit into part programs according to special CNC programming G codes. CNC machining is a course of CNC controlling a machine tool to complete machining of workpiece according requirements of part programs. Technical flow of CNC machining is shown in Fig. 1-2.

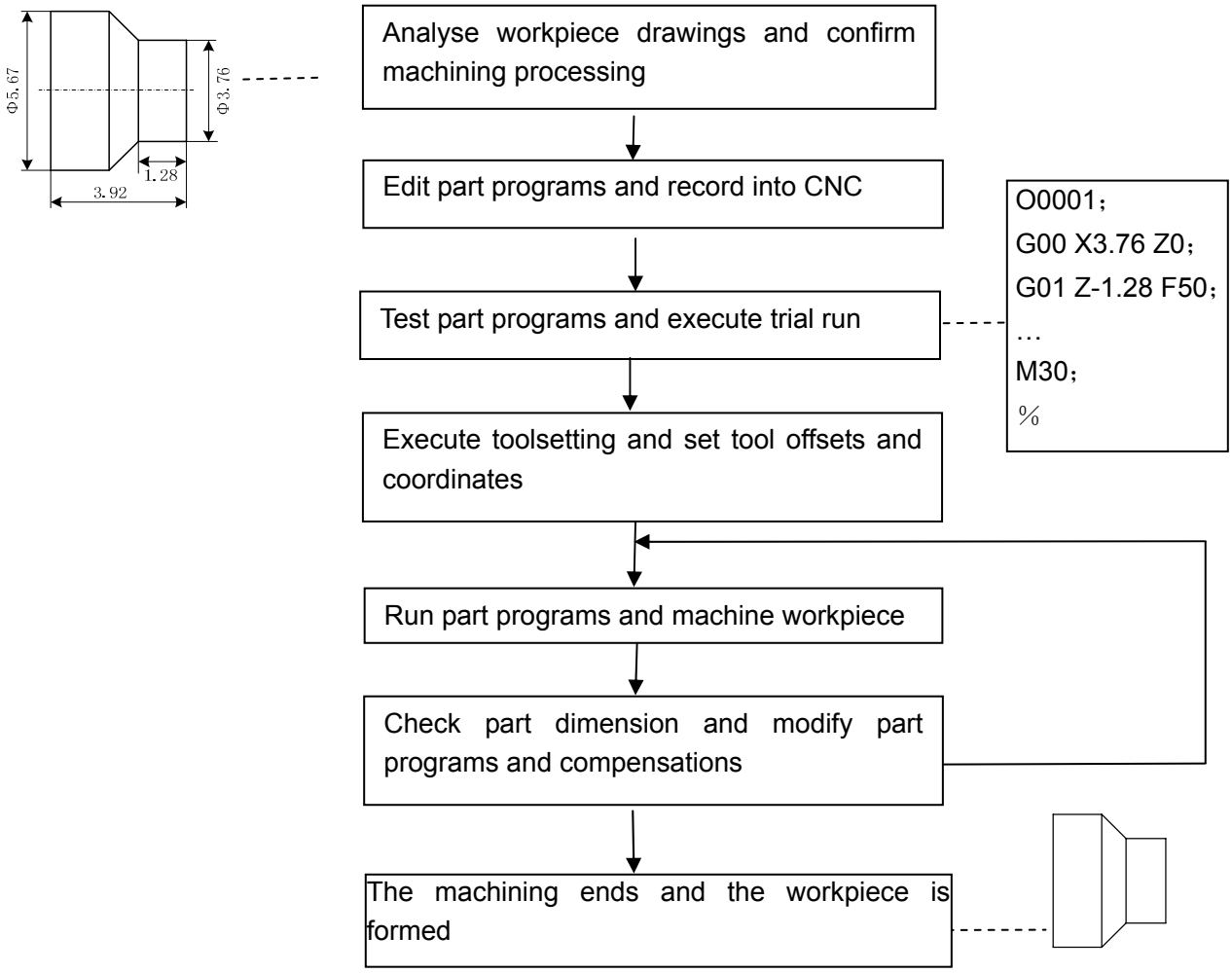


Fig. 1-2

1.3 Programming Fundamentals

1.3.1 Coordinates Definition

The following figure is the sketch of CNC turning:

Fig.1-3

GSK988TA/TB uses 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.
Parameter NO.1020 can set and modify program names for each axis and their corresponding relationship is shown below:

Table 1-3（a）

Axis name	Setting value	Axis name	Setting value
X	88	Z	90
Y	89	A	65
B	66	C	67
U	85	V	86
W	87		

Note: U, V, W is set only in G code system B.

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

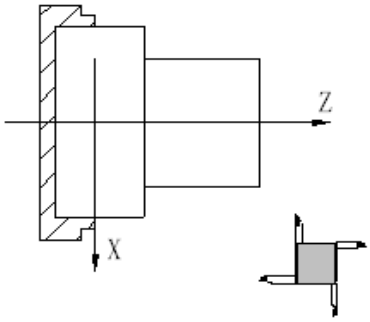


Fig.1-4 Front tool post coordinate system

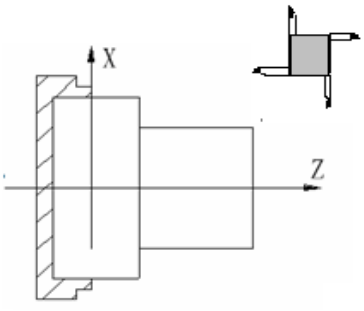


Fig.1-5 Rear tool post coordinate system

1.3.2 Increment System

Increment system includes least input increment (input) and least code increment (output). Least input increment is the least unit of programming movement distance. Least code increment is the least unit of tool movement on the machine tool. Their unit: mm, inch or degree.

Increment systems are separately IS-B and IS-C. No.1004 Bit1 decides to select IS-B or IS-C. No.1004 Bit1 is applied to all axes.

Table 1-3（b） increment system IS-B

		Least input increment	Least code increment
Metric machine	mm input	0.001mm（diameter） 0.001mm（radius） 0.001deg	0.0005mm 0.001mm 0.001deg

	inch input	0.0001inch（diameter） 0.0001inch（radius） 0.001deg	0.0005inch 0.001inch 0.001deg
Inch machine	mm input	0.001mm（diameter） 0.001mm（radius） 0.001deg	0.00005mm 0.0001mm 0.001deg
	inch input	0.0001inch（diameter） 0.0001inch（radius） 0.001deg	0.00005inch 0.0001inch 0.001deg

Table 1-3（c） increment system IS-C

		Least input increment	Least code increment
Metric machine	mm input	0.0001mm（diameter） 0.0001mm（radius） 0.0001deg	0.00005mm 0.0001mm 0.0001deg
	Inch input	0.00001inch（diameter） 0.00001inch（radius） 0.0001deg	0.00005inch 0.0001inch 0.0001deg
Inch machine	mm input	0.0001mm（diameter） 0.0001mm（radius） 0.0001deg	0.000005mm 0.00001mm 0.0001deg
	Inch input	0.00001inch（diameter） 0.00001inch（radius） 0.0001deg	0.000005inch 0.00001inch 0.0001deg

Whether the least input increment is mm or inch is determined by the machine based on the parameter INM(1001#0). The least input increment can be switched between the inch and the mm input, which is controlled by G codes(G20 or G21) or the set parameter.

1.3.3 Max. Travel

Max. travel=least code increment X（±）99999999

Table 1-3 (d) max. travel IS-C

Increment system		Max. travel
IS-B	Metric machine system	±99999.999mm ±99999.999deg
	Inch machine system	±9999.9999inch ±99999.999deg
IS-C	Metric machine system	±9999.9999mm ±9999.9999deg
	Inch machine system	±999.99999inch ±9999.9999deg

Note 1: The unit is diameter value in diameter programming, is radius value in radius programming in the above table.
Note 2: The input code cannot exceed max. travel code.
Note 3: The actual travel decides the machine tool.

1.3.4 Reference Point

A reference point is a fixed point on the machine tool. The tool can move to the position by executing the reference point return function. Generally, the reference point is used to tool change and setting coordinate system. GSK988TA/TB Turning CNC System can set 4 reference positions by parameters, which is shown in the following figure:

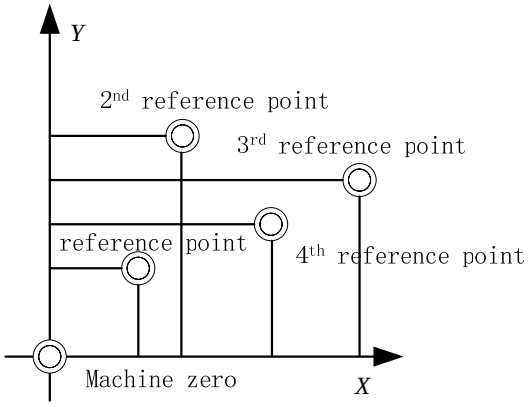


Fig.1-6 reference point

1.3.5 Machine Coordinate System

The machine tool coordinate system is a benchmark one used for the CNC counting coordinates and a fixed one on the machine tool. A machine tool zero is a fixed point which position is specified by zero switch or zero return switch on the machine tool. After the system is turned on, the reference point return is executed to set machine coordinate system. The machine coordinate system is not keeping until the system is turned off.

Note: For the machine with the incremental encoder, must execute the reference position return every time to set the machine coordinate system after power-off; for the machine with the multi-coil absolute encoder, need not execute the reference position return every time after power-off.

1.3.6 Workpice Coordinate System

The workpiece coordinate system is a rectangular coordinate system based on the part drawing, also called floating coordinate system. The workpiece coordinate system is set by the system in advance, can be changed by moving its coordinate origin point. The established workpiece is valid till it is replaced by a new one. The system has preset 6 workpice coordinate systems (G54-G59).

1.3.7 Local Coordinate System

When the system compiling programs in the workpiece coordinate system, sub-coordinate system of workpiece coordinate system can be set for easily programming, called local coordinate system as follows:

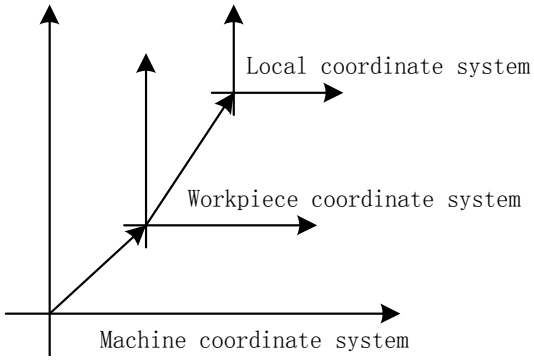


Fig.1-7 local coordinate system

1.3.8 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 Composite motion path. Positioning control is defined that a motion end point of one axis or multiple axes instead of the motion path in the course of running is controlled.

GSK988TA/TB has linear, arc and thread interpolation functions.

Linear interpolation: Composite motion path of Xp/Yp, and Zp axis is a straight line from start point to end point.

Circular interpolation: Composite motion path of Xp/Yp/Yp/Zp, and Zp/Xp axis is arc radius defined by R or the circle center (I, J, K) from start point to end point.

Thread interpolation: Moving distance of X or Z axis or X and Z axis 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.

Note 1:Xp, Yp, Zp are separately X or its parallel axis, Y or its parallel axis, Z or its parallel axis. The followings are the same as those.

Note 2: IP expresses the combination of X_Y_Z_(used in programming).

Example:

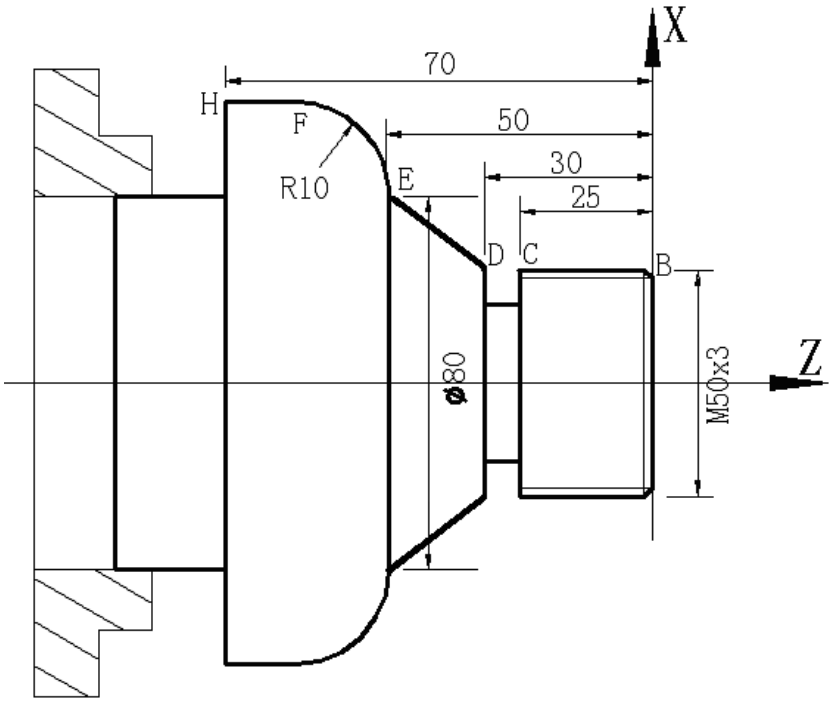


Fig.1-8

```
…
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;
```

1.4 Coordinate Value and Dimension

1.4.1 Absolute Programming and Incremental Programming

The system has two methods to code the tool traverse: absolute value and incremental value code. In the absolute programming, use the coordinate value programming of the end point; in the incremental programming, use the movement distance programming.

GSK988TA/TB system, the turning machine G codes are divided into two: A set of G code and B set of G code. In A set of G code system, a code's word determines to use the absolute value programming or incremental programming as the following Table 1-4(a); In B set of G code system, G90 and G91 determines to use the absolute value programming or incremental programming, G90 is an absolute code and G91 is an incremental code.

Table 1-4（a）

	Absolute value code	Incremental value code
X movement code	X	U
Y movement code	Y	V
Z movement code	Z	W
C movement code	C	H

A movement code	A	None
B movement code	B	None

In A set of G code system, the system can select the incremental programming or the absolute programming mode, or the incremental/absolute compound programming; the absolute code and the incremental code can be in the same block as follow:

X100.0 W100.0; compound programming

When the absolute code and the incremental code of one axis are in the same block, the following code value is valid.

In B set of G code system, the absolute value code and incremental value code cannot be in the same block. In one block, G90/G91 codes the absolute value code or incremental value code in the block as follows:

G90 X100.0； absolute programming

G91 X100.0; incremental programming

An axis word can exist repetitively in the same block and the later value is valid, but when No.3403 Bit 6 (AD2) is set to 1, the alarm occurs. U, W in other G code has bee specified to others. For example: in G73, the above conditions are described in G function codes.

1.4.2 Diameter Programming and Radius Programming

Because the workpiece section is the circle in CNC turning controlled program, X dimension can use two kind of method; diameter programming code and radius programming code.

1. The user can select the radius programming or diameter programming, which is set by No. 1006 Bit 3(DIAX)).
2. Parameters relevant with diameter/radius programming:

State parameter No.1006 BIT3 (DIAx):

0—radius programming；

1—diameter programming；

State parameter No.5004 Bit1(ORC):

0—offset value is expressed with diameter;

1—offset value is expressed with radius;

Pay more attention to the conditions in the following table when X uses diameter programming:

Table 1- 4 (b) addresses and data relevant with the diameter or radius programming

	Word	Explanation	Diameter programming	Radius programming
Addresses and data relevant with the diameter or radius programming	X	X coordinate, polar coordinate	Diameter value	Radius value
		G50 sets X coordinate	Diameter value	Radius value
	U	X increment	Diameter value	Radius value
		G71 infeed amount	Radius value	
		X finishing allowance in G71, G72, G73	Defined by a parameter	
		tool retraction amount in G73	Radius value	
	R	Clearance in G71, G72	Radius value	
		Clearance after cutting in G75	Diameter value	Radius value
		Clearance to end point in G74	Diameter value	Radius value

		Taper in G90, G92, G94, G76, radius in G02, G03, thread finishing amount in G76	Radius value
	I	X amount of circle center	Radius value
	F	Pitch long axis is X in G32,G34,G92,G76	Radius value
		X feedrate display	Radius/rev, radius /min

Note: Besides the above-mentioned addresses and data related to the diameter programming or the radius programming, other related to word and data related to X numerical value are expressed with radius value.

1.4.3 Decimal Programming

Value can be input by decimal programming. Distance, time and speed can be input by decimal programming. The following addresses can use decimal point: X, Y, Z, A, B, C, U, V, W, H, I, J, K, R and F, and other addresses cannot use decimal programming. An alarm occurs a word has more than one decimal point; an alarm occurs when an address which cannot be specified by a decimal point has a decimal point.

There are two types of decimal point usage which is decided by No. 3401 Bit0(DPI).

When NO.3401 Bit0(DPI) is set to 1, a value without a decimal point is with mm, inch.

When NO.3401 Bit0(DPI) is set to 0, an input value is specified by least input increment.

Parameter setting			Least code unit
ROTx=0 Rotary axis	The rotary axis is not related to parameter INI	ISC=0	0.001deg
		ISC=1	0.0001deg
ROTx=1 Linear axis	INI=0 Metric	ISC=0	0.001mm
		ISC=1	0.0001mm
	INI=1 Inch	ISC=0	0.0001inch
		ISC=1	0.00001inch

Example: when the metric input, the least input increment unit are set to 0.001:

Program code	The corresponding actual value when DPI is 1	The corresponding actual value when DPI is 0
X1000 without a decimal code value	1000mm	1 mm
X1000.0 with a decimal code value	1000mm	1000mm

The decimal which is less than the least input increment unit is discarded in course of program being executed.

Example: X2.34567. When the least unit of input increment is 0.001mm, X2.34567 becomes X2.345, when the least unit is 0.0001inch, it becomes X2.3456.

Note: An alarm occurs when the specified is more than 8-digit value.

1.4.4 Conversion between the Metric and the Inch

Metric input or inch input is set by NO.0000 Bit2(INI). G codes corresponding to metric/inch system is as follows:

G20： inch input ；

G21： mm input.

Input data unit becomes the inch or metric input unit when NO.0000 Bit2 (INI) setting is changed. But, the angle unit is not changed. It is suggested that the system should be turned on again when INI is modified. The unit of the following value is changed after metric/inch system is switched.

——F feedrate;

——position code;

——zero offset of workpiece;

——tool compensation value;

——graduation unit of MPG;

——movement distance in incremental feed.

NO.1001 Bit0 (INM) can be used to set machine's metric/inch output instead of metric/inch input.

1.4.5 Linear Axis and Rotary Axis

NO.1006 Bit0 (ROTx) can set each axis to a linear axis or rotary axis. NO. 1006 Bit 1 (ROSx) can be used to select the rotary type for each axis.

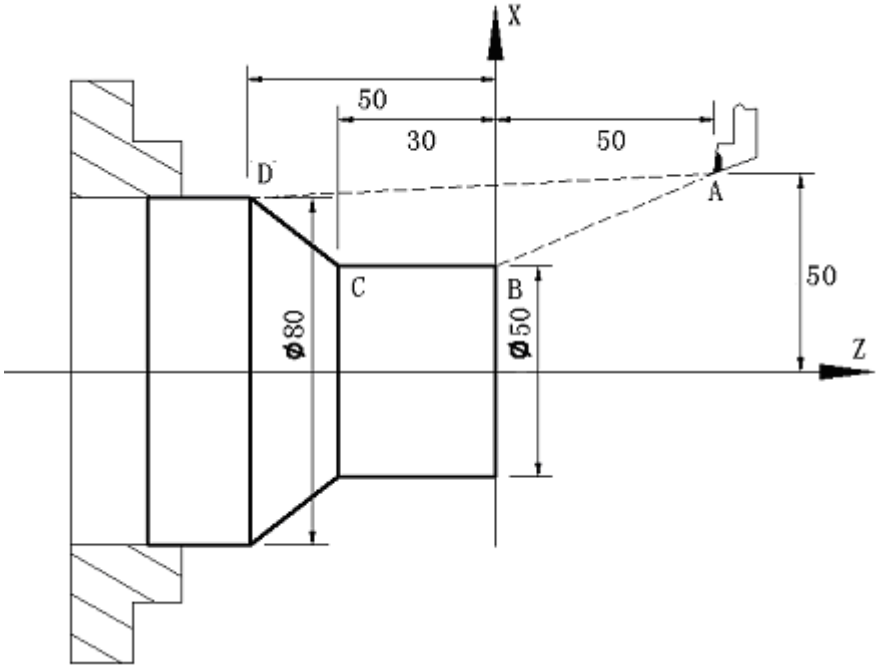
Absolute coordinate value is displayed circularly with the movement per rev set by NO.1260 when the cycle function is executed, which can prevent the rotary axis from overflowing. The cycle function is valid when NO.1008 Bit 0(ROAx) is set to 1.

For absolute value code, the coordinate value is the corresponding angle cycle value of per rev set by NO. 1260 after the machine moves. When NO.1008 Bit 1(RABx) is set to 0, the machine rotates according to the shortest distance (to the target point). For incremental code, the machine moves according to the angle defined by the code.

1.5 Structure of an NC Program

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

Program example:



O0001	;	(Program name)
N0005	G0 X100 Z50;	(Rapidly positioning to A point)
N0010	M12;	(Clamping workpiece)
N0015	T0101;	(Changing No.1 tool and executing its offset)
N0020	M3 S600;	(Starting the spindle with 600 r/min)
N0025	M8	(Cooling 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 Z50;	(Rapidly retracting to A point)
N0070	T0100;	(Canceling the tool offset)
N0080	M5 S0;	(Stopping the spindle)
N0090	M9;	(Cooling OFF)
N0100	M13;	(Releasing workpiece)
N0110	M30;	(End of program, spindle stopping and Cooling OFF)

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

A program consists of a sequence of blocks, beginning with “OXXXX”(program name)and ending with “M30”. A block consist of many words, beginning with block number (can be omitted) and ending with “；”. See the general structure of a program as Fig. 1-10:

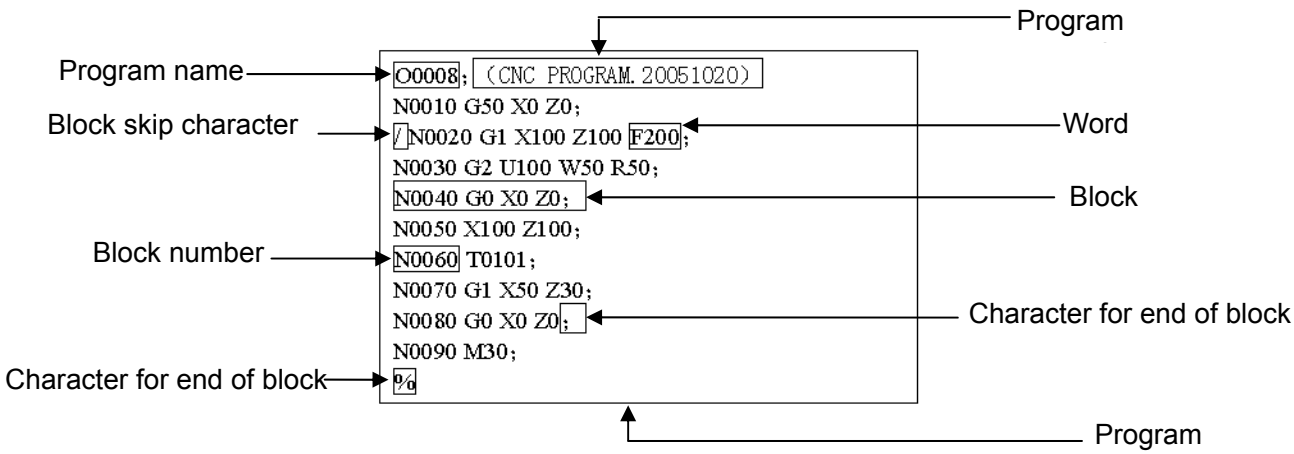


Fig.1-10 Structure of a program

1.5.1 Program Name

Format: O △△△△

Program number (0000～9999, the leading zero can be omitted)

Address O

△△△△ is number of a program name, its range is 4-digit integer 0000～9999, an alarm occurs when the negative program name is input.

1.5.2 Block Format

1. Format: / N△△△△△countless words;

/: skip character. A block can have or not it, generally, it is placed in the initial position of a program; user can press "SKIP" on the operation panel to execute the operation when the skip function is valid, otherwise, the "SKIP" key on the operation panel is valid, i.e. the skip character in the block is invalid;

N△△△△△: block number. A block can have or not it; number △△△△△ following N is 5-digit non-negative integer 00001～99999, and the system alarms when the input number is decimal.

Countless words: one block can input countless words, and one block can have one or more words or have no words.

;: "EOB" is a end character when one block is completed, ";" is displayed in LCD, there must be have one end character for one block;

2. Format requirements

（1）In one block, there can be no blank space between block number and word, and can be countless blank space(the total characters of one block is within 255);

（2）In one block, there can be not or be countless space between skip character and block number or words;

（3）In one block, there can be not or be countless space between end character of block and its front word or blocks;

Each block can be up to 256 characters, including skip character, block number, code, space, end character of block“;”;

（4）The system automatically ignores the content with small bracket “(”,“)”.

Explanations of program annotation:

Note: The annotation of program home as the total annotation of a program is displayed in the program catalog window, the created program automatically creates the small brackets“(”, “)”, if they are deleted, the system has no them and they can be replaced by “;”.

Sprit(/) explanations:

Slash sign (/) explanations:

Note 1：When the slash sign (/) is taken as a skip sign, it is placed at the beginning of a block. When it is placed at other places and the skip switch is open, the messages between the slant and EOB code are ignored. Example: U10. G00/04；when the skip switch is open, the system executes U10. G00;(G00 U10.); when the skip switch is closed, it executes U10. G0004;(G04 U10.);

Note 2：When the cycle code buffer is executed and the block is read to the buffer memory from the memory, the system ensures the skip function is valid. After the block is read into the buffer memory, even if the skip switch state is changed, the block read into the buffer is not influenced;

Note 3：The slash (/) symbol in <Expression> (closed in the brackets[]) and that at the right of assignment statement “=” are taken as a division operator, which is not taken as a skip symbol.

3. Parameters relevant with a block number:

（1）Automatically creates a block number：

The user can set the system automatically creates a block number by No.0000 bit5(SEQ);

（2）The user can set a interval value by No.3216 when the system automatically creates bock numbers.

1.5.3 Word

1. Format: address + number. There must not be a space between address and number.

Presently, the system permissively input addresses: G, M, S, T, F, X, Y, Z, U, V, W, P, Q, I, J, K, R, L, A, B, C, H, N, O，and will add other.

Code number range following address is referred to the following table.

Table 1-5-1 word and key word table A

Address	Function	mm input	Inch input	Related G codes
O	Program name	0～9999	0～9999	
N	Line label	0～99999	0～99999	
G	Preparatory function	See G command explanation	See G command explanation	
M	Miscellaneous function	0～9999	0～9999	
S	Spindle speed	（G96）0～32767 (m/min)	（G96）0～3276（feet/min）	
		（G97）0～32767 (r/min)	（G97）0～32767 (r/min)	
T	Tool offset	0000～9999	0000～9999	
F	Feedrate per minute（G98）	（ISB system）0．00１～60000 （mm/min）	（ISB system）0.00001～2400 (inch/min)	G98
		（ISC system）0．00１～24000 (mm/min)	（ISC system）0.0001～9600 (inch/min)	

Address	Function	mm input	Inch input	Related G codes
	Feedrate per rev（G99）	（ISB system）0.001～500 (mm/r)	（ISB system）0.0001～9.99 (inch/r)	G99
		（ISC system）0.001～500 (mm/r)	（ISC system）0.0001～9.99 (inch/r)	
	Pitch	0.01 ～500 (mm)	0.01～9.99 (inch)	Codes relevant with the thread
X	X absolute coordinate value((linear axis), delay time (*1)	（ISB system）-99999.999～99999.999 (mm)	（ISB system）-9999.9999～9999.9999 (inch)	Codes relevant with the thread , G04
		（ISC system）-9999.9999～9999.9999 (mm)	（ISC system）-999.99999～999.99999 (inch)	
Y	Y absolute coordinate value(linear axis) (*1)	（ISB system）-99999.999～99999.999 (mm)	（ISB system）-9999.9999～9999.9999 (inch)	Codes relevant with the axis
		（ISC system）-9999.9999～9999.9999 (mm)	（ISC system）-999.99999～999.99999 inch	
Z	Z absolute coordinate value (linear axis) (*1)	（ISB system）-99999.999～99999.999 (mm)	（ISB system）-9999.9999～9999.9999 (inch)	Codes relevant with the thread
		（ISC system）-9999.9999～9999.9999 (mm)	（ISC system）-999.99999～999.99999 (inch)	
A	A absolute coordinate value(linear axis) (*1)	（ISB system）-99999.999～99999.999 (mm)	（ISB system）-9999.9999～9999.9999 (inch)	Codes relevant with the thread
		（ISC system）-9999.9999～9999.9999 (mm)	（ISC system）-999.99999～999.99999 (inch)	
B	B absolute coordinate value(linear axis) (*1)	（ISB system）-99999.999～99999.999 (mm)	（ISB system）-9999.9999～9999.9999 (inch)	Codes relevant with the thread
		（ISC system）-9999.9999～9999.9999 (mm)	（ISC system）-999.99999～999.99999 (inch)	
C	C absolute coordinate value (rotary axis) (*1)	（ISB system）-99999.999～99999.999 (deg)	（ISB system）-99999.999～99999.999 (deg)	Codes relevant with the thread
		（ISC system）-9999.9999～9999.9999 deg	（ISC system）-9999.9999～9999.9999 (deg)	
U	X relative coordinate value, finishing allowance in G71, G72, G73, X tool retraction distance and specified delay time(*1) in G73	（ISB system）-99999.999～99999.999 (mm)	（ISB system）-9999.9999～9999.9999 (inch)	Codes relevant with the thread, G71, G04
		（ISC system）-9999.9999～9999.9999 (mm)	（ISC system）-999.99999～999.99999 (inch)	
	Cut depth in G71(modify parameter manual) (*2)	（ISB system）0.001~99999.999 (mm)	（ISB system）0.0001~9999.9999 (inch)	G71
		（ISC system）0.0001~9999.9999 (mm)	（ISC system）0.00001~999.99999 (inch)	
V	Y relative coordinate value(linear axis) (*1)	（ISB system）-99999.999～99999.999 (mm)	（ISB system）-9999.9999～9999.9999 (inch)	Codes relevant with the thread
		（ISC system）-9999.9999～9999.9999 (mm)	（ISC system）-999.99999～999.99999 (inch)	

Address	Function	mm input	Inch input	Related G codes
W	Z relative coordinate value, Z finishing allowance in G71, G72, G73, Z tool retraction distance (*1) in G73	（ISB system） -99999.999～99999.999 (mm)	（ISB system） -9999.9999～9999.9999 (inch)	Axis' relevant code，G71, G72，G73
		（ISC system） -9999.9999～9999.9999 (mm)	（ISC system） -999.99999～999.99999 (inch)	
	Cut depth (*2) in G72	（ISB system） 0.001~99999.999 (mm)	（ISB system） 0.0001~9999.9999 (inch)	G72
		（ISC system） 0.0001~9999.9999 (mm)	0.00001~999.99999 (inch)	
R	Arc radius (*1)	（ISB system） -99999.999～99999.999 (mm)	（ISB system） -9999.9999～9999.9999 (inch)	G02，G03
		（ISC system） -9999.9999～9999.9999 (mm)	（ISC system） -999.99999～999.99999 (inch)	
	Taper and thread taper (*1) in G90, G92, G94, G76	（ISB system） -99999.999～99999.999 (mm)	（ISB system） -9999.9999～9999.9999 (inch)	G90，G92 G94，G76
		（ISC system） -9999.9999～9999.9999 (mm)	（ISC system） -999.99999～999.99999 (inch)	
	Tool retraction（*2） in G71,G72	（ISB system） 0～99999.999 (mm)	（ISB system） 0～9999.9999 (inch)	G71，G72
		（ISC system） 0～9999.9999 (mm)	（ISC system） 0～999.99999 (inch)	
	Roughing times in G73	1～999（times）	1～999（times）	G73
	Thread increment in variable pitch cutting	0.01～499.99 (mm) -0.01～-499.99 (mm)	0.01～9.99(inch) -0.01～-9.99(inch)	G34
	Tool retract amount （*2） after cutting in G74, G75 and tool retraction after cutting to end point	（ISB system） 0～99999.999 (mm)	（ISB system） 0～9999.9999 (inch)	G74，G75
		（ISC system） 0～9999.9999 (mm)	（ISC system） 0～999.99999 (inch)	
	Finishing amount（*2） in G76	（ISB system） 0.001～99999.999 (mm)	（ISB system） 0.0001～9999.9999 (inch)	G76
		（ISC system） 0.0001～9999.9999 (mm)	（ISC system） 0.00001～999.99999 (inch)	
P	Dwell time	0~99999999ms	0~99999999 ms	G04
	G30 returning to No.n reference position	2， 3，4	2，3，4	G30（default to 2）
	Codes for macro program number, subprogram and subprogram call times	1～9999	1～9999	G65，G66 M98（times is defaulted to 1）
	Line number assignment in G70, G71, G72,G73	0～99999	0～99999	G70，G71 G72，G73
	X cycle movement（*3）in G74, G75	1～99999999× least code unit	1～99999999× least code unit	G74，G75

Address	Function	mm input	Inch input	Related G codes
	Thread cutting parameter in G76	Including 3 parameters: Thread finishing times：1～99 Thread run-out length：00～99（*0.1 pitch） Angle between two teeth：0°~99°	Including 3 parameters: Thread finishing times：1～99 Thread run-out length：00～99（*0.1 pitch） Angle between two teeth：0°~99°	G76
	Thread tooth height（*3）in G76	1～99999999× least code increment	1～99999999× least code increment	G76
	Spindle selection in multi-spindle	0～3	0～3	Spindle speed S code
Q	Line number assignment in G70, G71, G72, G73	0～99999	0～99999	G70, G71 G72, G73
	Tool infeed amount(*3) in Z brokenly infeed in G74,G75	1～99999999× least code increment	1～99999999× least code increment	G74, G75
	Min. cutting amount (*3) in G76 thread roughing	0～99999999× least code increment	0～99999999× least code increment	G76
	1st thread cutting depth（*3）in G76 thread roughing	1～99999999× least code increment	1～99999999× least code increment	G76
	Initial angle（*3）of 1st circle in thread cutting	0～99999999× least code increment（default to 0）	0～99999999× least code increment（default to 0）	G32, G34 G92
L	Macro program call times assignment	1～9999（default to 1）	1～9999（default to 1）	G65, G66
	Head quantity of multi-thread	1～99（default to 1）	1～99（default to 1）	G92
I	X vector of arc center relative to start point (*1)	（ISB system） -99999.999～99999.999(mm)	（ISB system） -9999.9999～9999.9999(inch)	G02, G03
		（ISC system） -9999.9999～9999.9999 (mm)	（ISC system） -999.99999～999.99999 (inch)	
J	Y vector of arc center relative to start point (*1)	（ISB system） -99999.999～99999.999 (mm)	（ISB system） -9999.9999～9999.9999 (inch)	G02, G03
		（ISC system） -9999.9999～9999.9999 (mm)	（ISC system） -999.99999～999.99999 (inch)	
	Movement in short axis when thread run-out (*1)	（ISB system） -99999.999～99999.999 (mm)	（ISB system） -9999.9999～9999.9999 (inch)	G32, G34 G92
		（ISC system） -9999.9999～9999.9999 (mm)	（ISC system） -999.99999～999.99999 (inch)	
K	Z Vector of arc center relative to start point (*1)	（ISB system） -99999.999～99999.999 (mm)	（ISB system） -9999.9999～9999.9999 (inch)	G02, G03
		（ISC system） -9999.9999～9999.9999 (mm)	（ISC system） -999.99999～999.99999 (inch)	
	Length in long axis when thread run-out is executed（*2）	（ISB system） 0～99999.999 (mm)	（ISB system） 0～9999.9999 (inch)	G32, G34 G92
		（ISC system） 0～9999.9999 (mm)	（ISC system） 0～999.99999 (inch)	

Table 1-5-1 word and key word table B

Sign	Abbrev	Function description	Remark
IF		Conditional judgement	
THEN	TH	Execution after IF conditional judgment is completed successfully	
GOTO	GO	Non-conditional skip	
WHILE	WH	Cycle judgment	
DO		Start to execute cycle	
END	EN	Return to WHILE	
EQ	==	Equal to	Jugement logic, only used to brackets after IF,WHILE
NE	<>	Not equal to	
GT	>	Greater than	
GE	>=	Greater than or equal to	
LT	<	Less than	
LE	<=	Less than or equal to	
SIN	SI	Sine	Functional function, used to coumt a expression value
ASIN	AS	Anti-sine	
COS	CO	Cosine	
ACOS	AC	Anti-cosine	
TAN	TA	Tangent	
ATAN	AT	Anti-tangent	
SQRT	SQ	Square root	
ABS	AB	Absolute value	
ROUN	RO	Rounding-off	
FIX	FI	Down integer	
FUP	FU	Up integer	
LN		Nature logarithm	
EXP	EX	Exponential function	
OR		OR	
XOR	XO	OR AND	
AND	AN	AND	
BIN	BI	Converse from BCD to BIN	
BCD	BC	Converse from BIN to BCD	
01234 56789		With to compose the value of word, the leading 0 can be omitted	
0		Word is 0 and is different with Null value	
+		Number count and number expression	
-			
*			
/			
		Skip code, selectively skip the codes following the character	
.		Floating point number with number	
=		Variable assignment	
[		Prior operation of expression and conditional judgement prompt	
]			
#		Variable	
;		End of program in the block, following annotation	

Sign	Abbrev	Function description	Remark
(		Annotation start in the block. Example: (X20.)W-10.; not execute X20.	
)		Annotation end in the block	
%		End of program	

Note 1: The 2-digit following the decimal point of F value is value, and the more following the two-digit is ignored.

Note 2: The expression can follow the word, the value counted by the expression is taken as the value of the word, and the expression should have[] , and there must not be the space between the word and the expression. For example X[#1-#110] Z[#1+SIN[#120]].

Ⅰ. When the address values in the above table, X, Y, Z, C, A, B, C, U, V, W, H, I, J, K, R are taken as word address, their value ranges are controlled by the following 4 parameters:

（1）No.0000#2 INI

INI input unit

0：metric

1：inch

（2）No.1006#0 ROTx

ROTx set linear axis or rotary axis

0：linear axis

1：rotary axis

（3）No.0004#1 ISC

ISC setting least input increment and least code increment

ISC	Least input increment	Abbrev
0	0.001mm, 0.001deg or 0.0001inch	IS-B
1	0.0001mm, 0.0001deg or 0.00001inch	IS-C

Table 1-5-2 least code increment and value range

Address	Parameter setting			Least code increment	Range
X, Y, Z, C, A, B, C, U, V, W, H	ROTx=0 Rotary axis	The Rotary axis is not related to INI	ISC=0 ISB	0.001deg	-99999.999～99999.999（deg）
			ISC=1 ISC	0.0001deg	-9999.9999～9999.9999（deg）
X, Y, Z, C, A, B, C, U, V, W, H, I, J, K, R	ROTx=1 Linear axis	INI=0 Metric system	ISC=0 ISB	0.001mm	-99999.999～99999.999 （mm）
			ISC=1 ISC	0.0001mm	-9999.9999～9999.9999 （mm）
		INI=1 Inch system	ISC=0 ISB	0.0001inch	-9999.9999～9999.9999 （inch）
			ISC=1 ISC	0.00001inch	-999.99999～999.99999 （inch）

When these word addresses follow data (with positive/negative sign), data precision is least code increment, and excessive data is ignored. When a word address follows variable number or has [ ] expression, the word value has decimal data, and its precision is the least code increment, but its excessive data rounds.

（4）No.3401#0 DPI

DPI can use decimal address. When the decimal is omitted, its setting is as follows:
0：least setting unit
1：unit: mm，inch，s

When parameter DPI is set to 1, word range is referred to Table 1-5-2;
When DPI is set to 0, and word omits its decimal, its value range is -99999999～99999999, data unit is the least code increment in Table 1-5-2.

Ⅱ. Code value calculation method specified by U, W, R, K is the same that of *1）, they meet the value range described in *1） and limit value range according to preparatory function.

Ⅲ. Position specified value coded by P, Q is 0～99999999, data unit is the least code increment in Table 1-5-2. value range is limit by specific preparatory function.

2. Word value and state will change when the system runs, the following table 1-5-3 separately explains each word omittance and state in the next block when the system is ON, resets.

Table 1-5-3 word state

Address	Function	Initial value when power-on	Default value	Keep in the next block?	Value after pressing reset key	Relevant explanation
O	Program name	Value reserved by last power-on	Current value	Yes	Yes	None
G	Preparatory function	Initial mode in each group	Modal value	No	（CLR） NO.3402#6	None
M	Miscellaneous function M00, M01, M02, M30, M98, M99	Null	Null	No （ function reserved）	Null	Specified by PLC, set by parameter
S	Analog spindle speed	0	Current value	Yes	Current value，output is invalid	
T	Tool offset	The tool number is the value reserved by last power-on and the tool offset value is 0	Current value	Yes	Current value	
IF	Feedrate	Parameter value	Current value	Yes	CLR） NO.3402#6	
	Pitch	Null	Current value	Yes	Current value	
X	Specify delay time	Null	0	No	0	

Address	Function	Initial value when power-on	Default value	Keep in the next block?	Value after pressing reset key	Relevant explanation
	X absolute coordinate value	0	Current value	Yes	Current value	
Y	Y absolute coordinate value	0	Current value	Yes	Current value	
Z	Z absolute coordinate value	0	Current value	Yes	Current value	
C	C absolute coordinate value	0	Current value	Yes	Current value	
U	Specify delay time	Null	0	No	Null	
	X relative coordinate value	0	0	No	Current value	
	X allowance in finishing	Null	0	No	Null	
	Cutting depth in G71	Parameter value	Parameter value	Yes	Parameter value	
V	Y relative coordinate value	0	0	No	Current value	
W	Z relative coordinate value	0	0	No	Current value	
	Z allowance in finishing	Null	0	No	Null	
	Cutting depth in G72	Parameter value	Parameter value	Yes	Parameter value	
H	C increment value	0	0	No	Current value	G00
		0	0	No	Current value	Polar coordinate interpolation
R	Arc radius	0	0	No	Current value	
	Taper G90, G92, G94 and thread taper	0	0	Yes	Current value	
	Tool retraction in G71, G72	Parameter value	Parameter value	Yes	Parameter value	
	Roughing times in G73	Parameter value	Parameter value	Yes	Parameter value	
	Clearance in G74,G75	Parameter value	Parameter value	Yes	Parameter value	
	Clearance to end point in G74,G75	0	0	No	Null	
	Finishing cutting amount in G76	Parameter value	Parameter value	Yes	Parameter value	
P	Dwell time	Null	0	No	Null	
	G30 returning to No. n reference position	Null	2	No	Null	

Address	Function	Initial value when power-on	Default value	Keep in the next block?	Value after pressing reset key	Relevant explanation
	Macro program number, subprogram, subprogram call times	Null	Alarm	No	Null	
	Line assignment in G70, G71, G72, G73	Null	Alarm	No	Null	
	X cycle movement in G74,G75	Null	0	No	Null	
	Thread cutting in G76	Parameter value	Parameter value	Yes	Parameter value	
	Thread tooth height in G76	0	Alarm	No	Null	
Q	Line assignment in G70, G71, G72, G73	Null	Alarm	No	Null	
	Z broken tool infeed amount in G74, G75	Null	0	No	Null	
	Least cutting amount in G76 roughing	Parameter value	Parameter value	Yes	Parameter value	
	1st thread cutting depth in G76 thread roughing	Null	Alarm	No	Null	
	1st circle start angle in thread cutting	Null	0	No	0	
L	Macro program call times assignment	1	1	No	Null	
I	X vector of circle center corresponding to start point	0	0	No	Current value	
	X calculation direction in cancelling radius compensation	Null	Null	No	Null	
J	Y vector of circle center corresponding to start point	0	0	No	Current value	
	Y calculation direction in cancelling radius compensation	Null	Null	No	Null	
K	Z vector of circle center corresponding to start point	0	0	No	Current value	
	Pitch increment in variable pitch thread cutting	Null	0	Yes	Current value	

Address	Function	Initial value when power-on	Default value	Keep in the next block?	Value after pressing reset key	Relevant explanation
	Z calculation direction in cancelling radius compensation	Null	Null	No	Null	

1.5.4 Block Number

1. Format: **N** △△△△△

△△△△△ is 5-digit integer 00001～99999, and its leading zero can be omitted.

（1）Can or not input a block number in one block (must input block number in target block in which program skips), when many block number are input in one block, only the last block number is valid;

（2）A block number can be placed any position of block but it is suggested that it should be placed at the initial position in order to search and read;

（3）There can be many same block number in one program, but the block number of target block of program skip has only one; otherwise, the program skips to the nearest block to the block;

（4）Block numbers can be placed at will.(it is suggested that it should be placed by the rising or falling monotonously;

Note: When the block number exceeds the range, and the program runs or the grammatical check is done, the relevant alarm occurs.

1.6 Program Run

1.6.1 Sequence of Program Run

Running the current open program must be in Auto mode. GSK988TA/TB cannot open two or more programs at the same time, and runs only one program any time. When one program is open, the cursor is located at display line of the program name and can be moved in Edit mode. In the run stop state in Auto mode, the program starts to run by the cycle start signal (CYCLE START key) is pressed or external cycle start signal)from a block pointed by current cursor, usually blocks are executed one by one according to their programming sequence, the program stops running till executing M02 or M30. The cursor moves along with program running and is located at the heading of the current block. Sequence and state of program running are changed in the followings:

- ◆The program stops run after pressing RESET or EMERGENCY STOP button;
- ◆The program stops running when the CNC system or PLC alarms;
- ◆The program runs and single block stops (the program run stops after the current block runs completely) in Edit, MDI mode, and then a block pointed by the current cursor starts running after the system switches into Auto mode, the CYCLE START key is pressed or external cycle start signal is switched on;
- ◆The program stops run in Manual(Jog), Handwheel (MPG), Single Block, Program Reference position Return, Machine Reference position Return mode and it continuously runs from the current stop position after the system is switched into Auto mode and the CYCLE START key is pressed or the external cycle start signal is switched on;
- ◆The program pauses after pressing the FEED HOLD key or the external cycle start signal

is switched off, and it continuously runs from current position after pressing the CYCLE START key 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 the CYCLE START key 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 Codes about execution sequence of G70~73;
- ◆Call corresponding subprograms or macro program to run when executing M98; 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 returns to the first block to run and the current program is executed repetitively when M99 is executed in a main program.

1.6.2 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 are transmitted to PLC by NC explaining and others are directly executed by NC. M98, M99, S word for specifying spindle speed (r/min, m/min) is directly executed by NC.

The NC firstly executes G and then M codes when G codes and M00, M01, M02 and M30 are in the same block.

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

When G codes 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 codes. Please see User Manual of machine manufacturer for execution sequence of codes.

Execution sequence of G, M (except for the above M codes), S, T defined by GSK988TA/TB PLC in the same block is determined by PLC, which is divided into two methods:

a) Movement codes and M miscellaneous code are executed simultaneously.

b) Execute miscellaneous codes after executing movement codes.

Refer to the machine manufacture's user manual for the concrete execution method.

The second method is executed when there is M9, M99, M13, M33 or M5 for our GSK ladder.

Note: When G28 or G30 and M01 are in the same block, the pause after zero return is done. When there is a single block stop code without M01, the single block stop is executed at the middle point and zero return completion position.

Chapter 2 G Codes

2.1 Summary

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

G □□
└ Command value
└ Command address

Note 1: The leading zero of the code value can be omitted. Example: G02 is equivalent to G2, G01 to G1.
Note 2: The leading zero following the code value cannot be omitted. Example: G20 and G2 are different G codes in the different group; G12.1 is legal but G12.10 is illegal.
Note 3: The code value can be up to 8-bit digit. Example: G00000002 is correct and valid, equivalent to G02.
Note 4: Except for G12.1/G13.1/G7.1/G50.2/G51.2/G50.4/G50.5/G50.6/G51.4/G51.5/G51.6/G54.1, other G code cannot be with the decimal point, otherwise, the alarm occurs. For example: G20.0, G00.0, G18.are illegal.

2.1.1 G code Classification

G codes are divided into: modal G code and non-modal G code.

After a G code is executed, its defined function or state remains valid till other G code is specified in the same group, this G code is called the modal. After the modal G code is executed, before its defined function or state is changed, the G code is not input again when the following block executes the G word.

After a G code is executed, its defined function or state is valid once, its word must be input again when it is executed, and so the G code is called the non-modal.

Example 1: G01 and G00 are modal.

```
G01 X_;
    Z_;      } G01 is valid in the range
    X_;
G00 Z_;
    X_;      } G00 is valid in the range
G01 X_;
```

Example 2: G04 is non-modal.

```
O0002;
G0 X50 Z5;      (Rapid traverse to X50 Z5)
G04 X4;         (Delay 4s)
G04 X5;         (Delay 5s again, G04 is non-modal and must be input again)
M30;
```

Table 2-1 G code list

G code system		Group	Function	Classification
A	B			
*G00	*G00	01	Positioning(rapid traverse)	Modal
G01	G01		Linear interpolation	
G02	G02		Circular interpolation(CW)	
G03	G03		Circular interpolation(CCW)	
G04	G04	00	Dwell, Exact stop	Non-modal
G7.1 (G107)	G7.1 (G107)		Cylindrical interpolation	
G10	G10		Programmable data input	
G11	G11		Programmable data input cancel	
G12.1 （G112）	G12.1 （G112）	21	Polar coordinate interpolation mode	Modal
*G13.1 （G113）	*G13.1 （G113）		Polar coordinate interpolation mode cancel	
G17	G17	16	XpYp plane selection	Modal
*G18	*G18		ZpXp plane selection	
G19	G19		YpZp plane selection	
G20	G20	06	Inch input	Modal
*G21	*G21		mm input	
*G22	*G22	09	Stored travel check ON	Modal
G23	G23		Stored travel check OFF	
G28	G28	00	Return to reference point	Non-modal
G30	G30		Return to 2nd, 3rd, 4th reference point	
G31	G31		Skip interpolation	
G36	G36		Automatic tool offset (X)	
G37	G37		Automatic tool offset (Z)	
G32	G32	01	Constant pitch thread cutting	Modal
G34	G34		Variable pitch thread cutting	
*G40	*G40	07	Tool radius compensation cancel	Modal
G41	G41		Cutter compensation left	
G42	G42		Cutter compensation right	
G50	G92	00	Workpiece setting or max. spindle speed setting	Non-modal
G52	G52		Local coordinate system setting	
G53	G53		Machine coordinate system setting	
*G54	*G54	14	Select workpiece coordinate system 1	Modal
G55	G55		Select additional workpiece coordinate system	
G56	G56		Select workpiece coordinate system 2	
G57	G57		Select workpiece coordinate system 3	
G58	G58		Select workpiece coordinate system 4	
G59	G59		Select workpiece coordinate system 5	
G61	G61	15	Select workpiece coordinate system 6	Modal

*G64	*G64		Cutting mode	
G65	G65	00	Non-modal macro program call	Non-modal
G66	G66	12	Macro program mode call	Modal
*G67	*G67		Cancel macro program mode call	
G70	G70	00	Finishing cycle	Non-modal
G71	G71		Axial roughing cycle	
G72	G72		Radial roughing cycle	
G73	G73		Closed cutting cycle	
G74	G74		Axial grooving cycle	
G75	G75		Radial cutting multi-cycle	
G76	G76		Multi thread cutting cycle	
*G80	*G80	10	Cancel drilling fixed cycle	Modal
G83	G83		End drilling cycle	
G84	G84		End rigid/common tapping cycle	
G85	G85		End boring cycle	
G87	G87		Side drilling cycle	
G88	G88		Side rigid/common tapping cycle	
G89	G89		Side boring cycle	
G90	G77	01	Axial cutting cycle	Modal
G92	G78		Thread cutting cycle	
G94	G79		Radial cutting cycle	
G96	G96	02	Constant surface speed control	Modal
*G97	*G97		Constant speed control	
*G98	*G94	05	Feed per minute	Modal
G99	G95		Feed per revolution	
—	*G90	03	Absolute code	Modal
—	G91		Incremental code	
—	*G98	11	Fixed cycle return to initial plane	Modal
—	G99		Fixed cycle return to point R plane	

Note 1: No. 3401 Bit6 sets A system or G code system B.

Note 2: G codes in Group 01, 05, 09 separately set their state in No.3402 Bit0(G01), Bit4 (FPM), Bit7 (G23) when the system is power-on, the G codes in Group 06 in No.0000 Bit2(INI); when the system is turned on, the modal G code in other groups are at the state designated by *.

Note 3: When the system resets, No.3402 Bit6 (CLR) is set to 0, the modal of the G code remains unchanged; when it is set to 1, the modal is changed to the one which is at the power-on, but G22 and G23 in Group 09 and G20 and G21 in Group 06 remain unchanged.

Note 4: G codes in Group 00 are non-modal.

Note 5: G codes in Group 00 and ones in Group 01 are specified in the same block, G codes in Group 00 are valid, G codes in Group 01 only change their modal.

Note 6: Codes in Group 06, 09, 21 and ones in other groups cannot be in the same block, codes in Group 12 and G65 are specified only in a separate block.

Note 7: When No.3403 Bit6(AD2) is set to 0, many G codes in the different groups can be specified in the same block, and the G code specified at last is valid; when it is set to 11, the alarm occurs.

Note 7: When compiling a G code in one block needs a word, and the compiled cannot use the word, the word is ignored(for example: G00 X_ Z_ R_ , R_ is ignored); when the ignored word format is not correct, the alarm occurs (For example: G00 X_ Z_ R2.3.1）.

Note 9: When compiling No.1020 does not have the axis word including the absolute address or incremental address, an alarm occurs.

2.1.2 Omitting Word Input

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

Table 2-2

Code address	Function	Initial value when power-on
U	Cutting depth in G71	NO.5132 value
	Move distance of X tool retraction in G73	NO.5135 value
W	Cutting depth in G72	NO.5132 value
	Move distance of X tool retraction in G73	NO.5136 value
R	Move distance of tool retraction in G71, G72 cycle	NO.5133 value
	Cycle times of stock removal in turning in G73	NO.5137 value
	Move distance of tool retraction after cutting in G74, G75	NO.5139 value
	Allowance of finishing in G76	NO.5141 value
	Taper in G90, G92, G94, G96	0
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.5142 value NO.5130 value NO.5143 value
Q	Least cutting value in G76	NO.5140 value
F	Metric pitch(G32, G92, G76)	0
	Feedrate per minute(G98)	NO.1411 value
	Feedrate per rotation (G99)	NO.1411 value multiplying 0.001
S	Spindle speed specified(G97)	0
	Spindle surface speed specified(G96)	0

Note 1: For the code 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 code, the pitch must be input with F word when machining metric thread.

Note 2: When the words in the above table (except for F, S) are not omitted, the input new code value is written to the corresponding parameter.

Note 3: When X（U）, Y（V）, Z（W）, A, B or C（H）are used to the end point coordinates of the specified block and their words in the block are not input, the system takes the absolute coordinates of the current X, Y, Z, A, B or C as the coordinates of the end point.

Example 1:（run after the first power-on）:

```
O0003;
G98 F500 G01 X100 Z100;        （G98: feed/minute，500mm/min）
G92 X50 W-20 F2 ;              （thread cutting, F must be input when it is the pitch）
```

```
G99 G01 U10 F0.01 ;          （G99: feed/minute, F is input again）
G00 X80 Z50 ;
M30;
```

Example 2:

```
O0001;
G0 X100 Z100;      （rapidly traverse to X100 Z100; the modal G0 is valid）
X20 Z30;           （rapidly traverse to X20 Z30; the modal G0 can be omitted）
G1 X50 Z50 F300;   （linear interpolation to X50 Z50， 300mm/min; the modal G1 is valid）
X100;              （linear interpolation to X100 Z50，300mm/min; When Z coordinate is
                     not input, the current coordinate value Z50 is used; F300 is kept,
                     G01 can be omitted when it is modal.）
G0 X0 Z0;          （rapidly traverse to X0 Z0，the modal G0 is valid）
M30;
```

2.1.3 Relevant Definitions

Definitions of word are as follows except for the especial explanations:

Start point: position before the current block runs;
End point: position after the current block ends;
Start point of cutting: initial position of cutting feed;
End point of cutting: end position of cutting feed;
X: X absolute coordinates of end point;
Xp：absolute coordinate of X end point or one which is parallel to X;
U: different value of X absolute coordinate between start point and end point;
Y：Y absolute coordinate of end point;
Yp：absolute coordinate of Y end point or one which is parallel to Y;
V：different value of Y absolute coordinate;
Z: Z absolute coordinates of end point;
Zp：absolute coordinate of Z end point or one which is parallel to Z;
W: different value of absolute coordinates between start point and end point;
C：C absolute coordinate of end point;
H：different value of C absolute coordinate between end point and start point;
A：A absolute coordinate of end point;
B：B absolute coordinate of end point;
F：cutting feedrate.
IP：a combination of axis word is called IP_.

2.2 Rapid Traverse (Positioning) G00

Command function: In the absolute code, the tool rapidly traverses to the position specified by the workpiece coordinate system; in the incremental code, the tool rapidly traverses to the position which offsets the specified value of the current position.

Command format: G00 IP__;

Command explanation: IP: it is the end point coordinate value of the tool traversing for the absolute code; it is the tool traversing distance for the incremental code.

Command path:

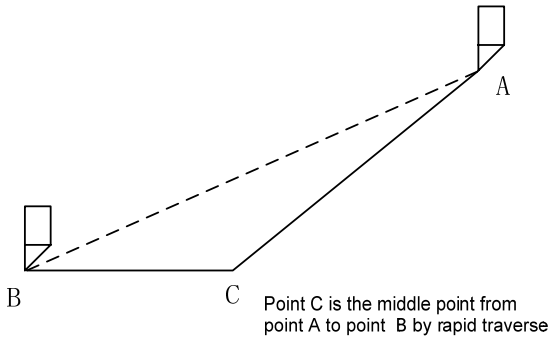
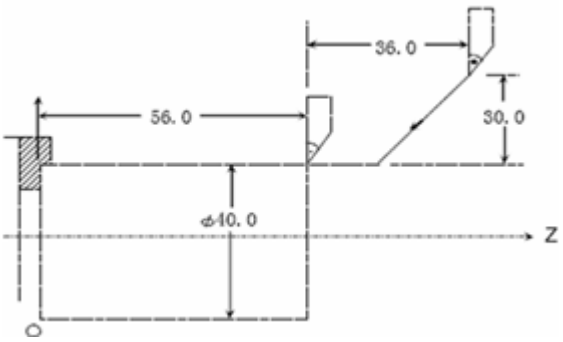


Fig.2-1 rapid traverse(positioning)

Execution process:



Program: (diameter programming）
G00 X40.0 Z56.0；(Absolute coordinate programming) or
G00 U60.0 W-36.0；(Incremental coordinate programming) or
G00 X40.0 W-36.0；（Compound programming）or
G00 U60.0 Z56.0；（Compound programming）

Fig. 2-2 positioning example

Note 1: The rapid traverse speed (G00) is set in No.1420 and is not related to the executed feedrate F value in the block.

Note 2: Whether the initial mode of Group 01 when power-on is G00 or G01 is determined by No.3402 Bit0 (G01).

2.3 Linear Interpolation G01

Command function: the tool executes the linear traverse.

Command format: G01 IP__ F__; it can be omitted to G1

Command explanation: IP_: it is the end point coordinate value of tool traversing for the absolute code; it is the tool traversing distance for the incremental code.

F_: it is the feedrate of the tool and its ranges is shown below.

Feed mode		Metric（mm）input	Inch (inch) input
G98	ISB system	1～60000 mm/min	0.01～2400 inch/min
	ISC system	1～24000 mm/min	0.01～960 inch/min

G99	ISB system	0.01～500mm/r	0.01～9.99inch/r
	ISC system	0.01～500mm/r	0.01～9.99 inch/r

Note: G98, G99 are separately feed per minute and feed per rotation. G94, G95 are separately feed per minute and feed per rotation in G code system B.

Command path:

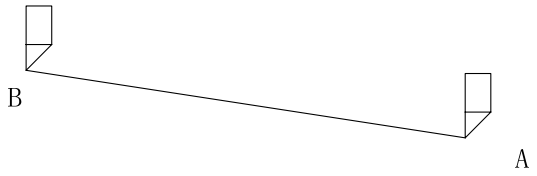
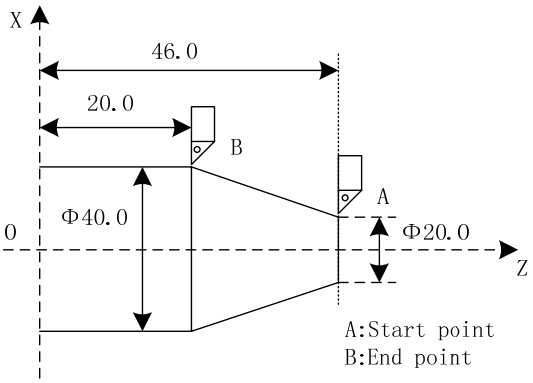


Fig. 2-3 linear interpolation

Execution process:



Program: (diameter programming）
G01 X40.0 Z20.0 F500;（(Absolute coordinate programming)
or
G01 U20.0 W-26.0; (Incremental coordinate programming) or
G01 X40.0 W-26.0;（Compound programming）or
G01 U20.0 Z20.0;（Compound programming）

Fig. 2-4 Linear interpolation example

Note 1: The tool traverses to the specified position along the linear at the speed specified by F. Before the new value is specified, each program is not needed to specify.

Note 2: The actual cutting feedrate is the product between the feedrate override and F code value.

Note 3: The actual cutting feedrate is limited by max. cutting feedrate MFR of No. 1422.

Note 4: G01 supports the synchronous interpolation of linear axis and rotary axis. The code speed includes the speeds of linear axis and rotary axis.

2.4 Circular Interpolation G02, G03

Command function: The tool traverses along an arc on the specified plane.

Command format:

$$G17\begin{Bmatrix}G02\\G03\end{Bmatrix}Xp\_Yp\_\begin{Bmatrix}R\_\\I\_J\_\end{Bmatrix}F\_$$

$$G18\begin{Bmatrix}G02\\G03\end{Bmatrix}Xp\_Zp\_\begin{Bmatrix}R\_\\I\_J\_\end{Bmatrix}F\_$$

$$G19\begin{Bmatrix}G02\\G03\end{Bmatrix}Yp\_Zp\_\begin{Bmatrix}R\_\\J\_K\_\end{Bmatrix}F\_$$

Command explanations:

Code	Description

G17	XpYp plane selection
G18	ZpXp plane selection
G19	YpZp plane selection
G02	Circular interpolation (CW)
G03	Circular interpolation (CCW)
Xp_	Movement of X or an axis parallel to it (set by No.1022)
Yp_	Movement of Y or an axis parallel to it (set by No.1022)
Zp_	Movement of Z or an axis parallel to it (set by No.1022)
I_	Distance between start point of Xp axis to center of arc (with sign, its range referred to the following table)
J_	Distance between start point of Yp axis to center of arc (with sign, its range referred to the following table)
K_	Distance between start point of Zp axis to center of arc (with sign, its range referred to the following table)
R_	Arc radius (with sign, it is the radius value when machining, range referred to the following table)
F_	Feedrate along arc（its range is the same that of G01）

Address	Incremental system	Metric input（mm）	Inch input (inch)
I，J，K，R	ISB system	-99999.999～99999.999	-9999.9999～9999.9999
	ISC system	-9999.9999～9999.9999	-999.99999～999.99999

I, J, K have sign symbols according their directions, they are positive when their directions are the same those of Xp, Yp, Zp, otherwise, they are negative.

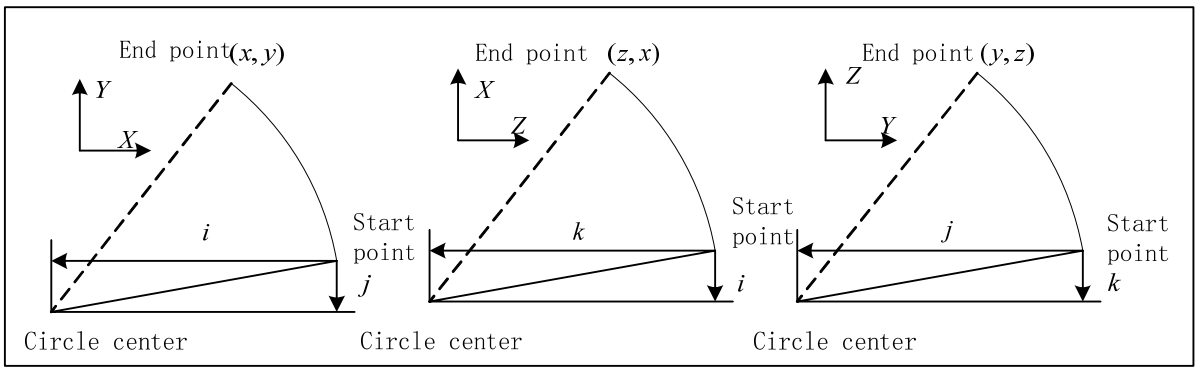


Fig. 2-5

Command path（arc direction）:

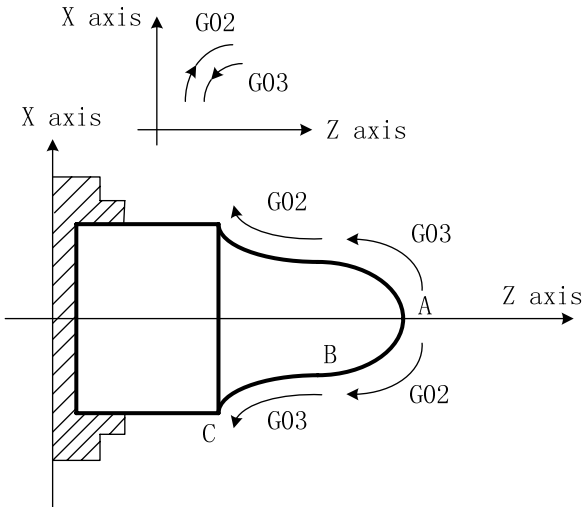
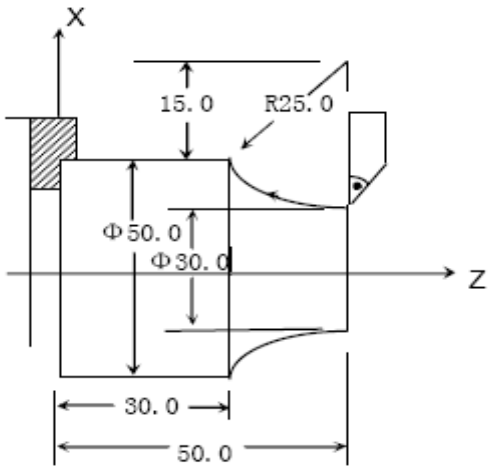


Fig. 2-6 circular interpolation

Execution process：（taking an example of G02）



Program: (diameter programming）
G02 X50.0 Z30.0 R25.0. F30 ； or
G02 U20.0 W-20.0 R25.0 F30 ； or
G02 X50.0 Z30.0 I25.0 F30（K=0）; or
G02 U20.0 W-20.0 I25.0 F30（K=0）;

Fig. 2-7 G02 circular interpolation

Note 1: One or all of Xp, Yp, Zp can be omitted. When one of them is omitted, it means the coordinate values of the start point and the end point of the axis is consistent; when all are omitted, it means the two points are in the same position.

Note 2: When I, J, K is omitted and the code is executed, the tool executes the linear movement according to No. 3403 setting value or alarms to stop move.

Note 3: When I = 0, J = 0 or K = 0, and the code is executed, the tool linearly traverses to the end point.

Note 4: When I and J, J and K, I and K, are input with R, only R is valid, I, J, K are invalid.

Note 5: When the start point and the end point are the same one, I, K are the center value, G02/G03 path is a full circle （360°）; When R is the arc radius, it means the circle is 0 degree.

Note 6: When R is the arc radius, it is more than or less than 180°, and it is more than 180° arc when R is negative; it is less than or equal to 180° when R is positive.

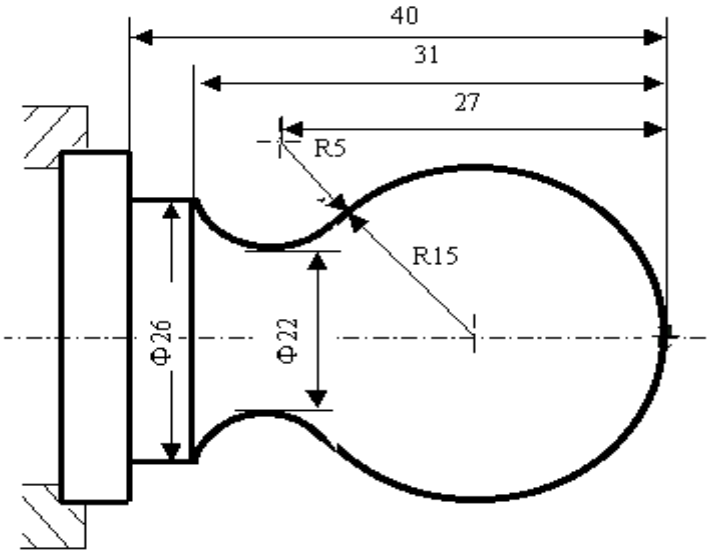
Note 7: The alarm occurs when the radius difference between the start point and the end point of arc exceeds the set value(except for 0) of No. 3410. When the difference does not exceed the setting value or the set value is 0, the tool firstly executes the arc interpolation along the radius value between the arc and the center, and traverse linearly to the end point; in using R programming, R should be equal to or more than the half between the start point and the end point; when the end point is not in the arc defined by R, the user can set whether the system alarms according to No. 3403 Bit4 (RER). It is suggested that the user should use R programming.

Note 8: In G02/G03 mode, the system alarms when the other axes exceeding the current plane are executed in G02/G03.

Note 9: The feedrate along the arc is related to not only F value and the override, but also the machining precision (ISB, ISC) and the machining radius. For example, when the arc radius is smaller, the machining cannot be executed at the set feedrate to get the machining precision.

Note 10: The actual cutting feedrate is limited to max. cutting feedrate MFR of No.1422

G02/G03 compound programming example：



Program：O0001

N001 G0 X40 Z5；（Rapidly traverse）

N002 M03 S200；（Start the spindle）

N003 G01 X0 Z0 F900；(Approach the workpiece)

N005 G03 U24 W-24 R15；（Cut arc R15）

N006 G02 X26 Z-31 R5；（Cut arc R5）

N007 G01 Z-40；(Cut Φ26)

N008 X40 Z5；(Return to start point)

N009 M30；（End of program）

Fig. 2-8 arc programming example

2.5 Spiral Interpolation G02, G03

Command function: when an circular interpolation is specified and the movement 1 axis or 2 axes of the axis exceeding the specified plane is executed, the spiral interpolation to make the tool spiral movement can be executed.

Command format:

$$G17\begin{Bmatrix}G02\\G03\end{Bmatrix}Xp\_Yp\_\begin{Bmatrix}R\_\\I\_J\_\end{Bmatrix}\alpha\_(\beta\_)F\_$$

$$G18\begin{Bmatrix}G02\\G03\end{Bmatrix}Xp\_Zp\_\begin{Bmatrix}R\_\\I\_J\_\end{Bmatrix}\alpha\_(\beta\_)F\_$$

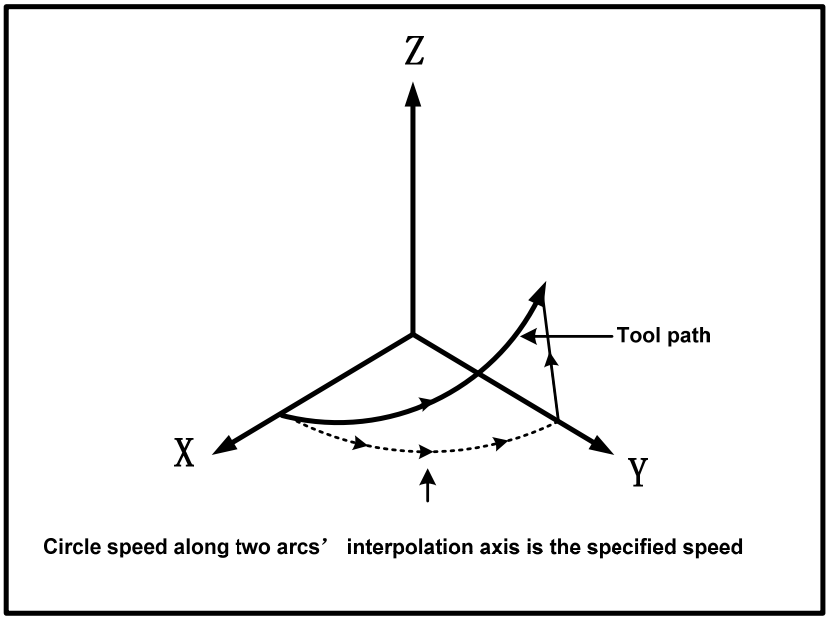
$$G19\begin{Bmatrix}G02\\G03\end{Bmatrix}Yp\_Zp\_\begin{Bmatrix}R\_\\J\_K\_\end{Bmatrix}\alpha\_(\beta\_)F\_$$

A, β: can specify any one linear axis, up to 2 axes exceeding the circular interpolation axis.

Command explanation: The speed code can be set by HTG (No.1403#5), and it can be specified by arc's tangent speed or the tangent speed containing a linear axis.

HTG=0: F specifies the feedrate along the arc. So, the linear axis speed is:

$$F\times\frac{linear\ axis\ length}{arc's\ arc\ length}$$

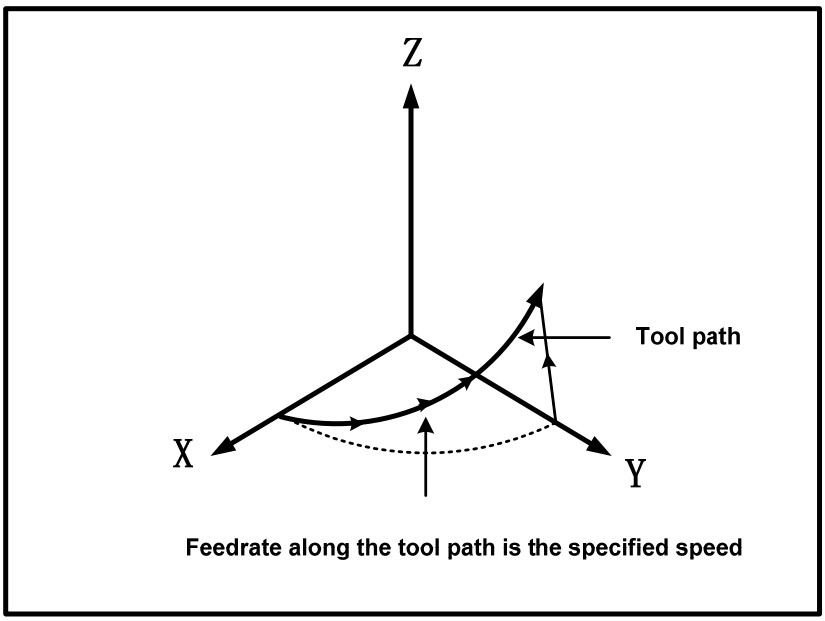


HTG=1: F is the resultant speed containing arc and linear feedate, so the arc tangent speed is :

$$F\times\frac{arc's\ arc\ length}{\sqrt{(arc's\ arc\ length)^2+(linear\ axis\ length)^2}}$$

Linear axis speed:

$$F\times\frac{linear\ axis\ length}{\sqrt{(arc's\ arc\ length)^2+(linear\ axis\ length)^2}}$$



Note: the tool nose radius compensation is applied to only arc.

2.6 Dwell G04

Command function: it can delay the next block to execute in the defined time.

Command format: G04 P__ ; or
G04 X__ ; or
G04 U__ ; or
G04;

Code specification: G04 is non-modal.
The dwell time is defined by the word P__, X__ or U__.
X, U value can specify the decimal.
P value cannot have the decimal, otherwise, the system alarms.

Time of P__, X__ or U__ is shown below.

Address	P			U	X
Unit	DWT=1	0.001s		s	s
	DWT=0	ISB	0.001s		
		ISC	0.0001s		

Note: DWT is the setting value of No. 1015 Bit 7(DWT).

Value range of P__, X__ or U__ is shown below.

Address	Incremental system	Metric input	Inch input
X, U	ISB system	-99999.999～99999.999	-9999.9999～9999.9999
	ISC system	-9999.9999～9999.9999	-999.99999～999.99999
P	ISB, ISC	0~99999999	0~99999999

Note 1: The system exactly stop between blocks when P, X, U are not input or zero is specified..

Note 2: P cannot code a negative value, otherwise, an alarm occurs.

Note 3: P time unit is set by No. 1015 Bit 7(DWT).

Note 4: P, X, U are in the same block, P is valid; X, U are in the same block, the later specified code is valid.

Note 5: The dwell can be executed after the current delay time is completed in executing the feed hold in G04.

Note 6: When G04 and subprogram M98 /M99 P__ are in the same block, the number following P is the time value of G04 dwell, and is also the message of M98/M99, i.e. subprogram skip message error.

Note 7: G04 and the interpolation code in Group 1(such as G00, G01) are in the same block, G04 is valid, G0, G01 only change the modal value of G codes in Group 1.

Note 8: When No.3403 Bit 6(AD2) is 0, G04 and G codes in Group 00 are in the same block, and the later specified code is valid.

2.7 Cylindrical Interpolation 7.1

Command function: the cylindrical interpolation is defined that the movement amount of rotary axis specified by angle is converted into the movement distance of linear axis along the surface in the CNC inside, which makes the rotary axis and other axis execute the linear interpolation or circular interpolation. After interpolation, the distance is converted into the movement amount of the rotary axis, which is shown below:

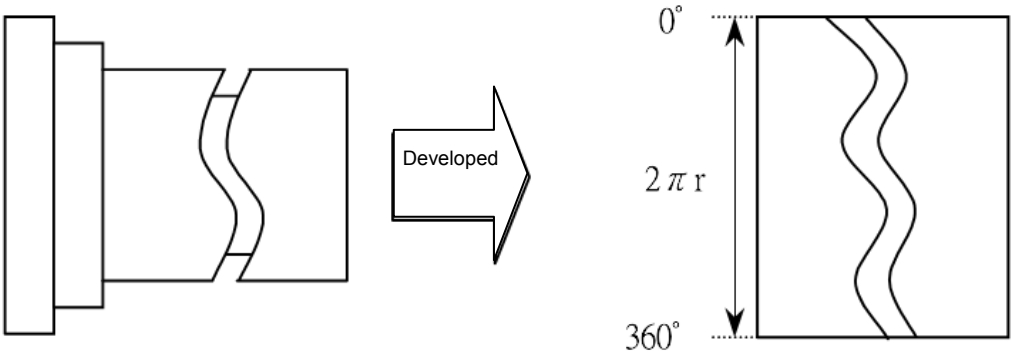


Fig. 2-9

Command format:

$$G07.1\begin{Bmatrix} X(U) \\ Y(V) \\ Z(W) \\ A \\ B \\ C \end{Bmatrix} \underline{r};$$

Activate the cylindrical interpolation code. G07.1 can be written to G107 or G7.1, but must not be with other code in a line;

……;

……;

$$G07.1\begin{Bmatrix} X(U) \\ Y(V) \\ Z(W) \\ A \\ B \\ C \end{Bmatrix}\underline{0};$$ Disable the cylindrical interpolation mode. It must not be with other code in a line;

Command explanation: G7.1 is non-modal;
r is the cylindrical radius.
Unit of the rotary axis is not degree, but mm (metric input) or inch(inch input)

Note 1: The rotary axis in the cylindrical interpolation mode is specified by No. 1022, X, Y, Z or an axis parallel with it is also done. G17～G19 is specified to select the plane for which the rotary axis is the specified linear axis. For example, when the rotary axis is X, G17 must specify XY plane which is determined by the rotary axis and Y axis.

Note 2: Before the cylindrical interpolation, the plane for cylindrical interpolation must be specified firstly, otherwise, the alarm occurs;

The alarm does when G17~G19 is specified to select the plane when the cylindrical interpolation is being executed;

G17~G19 must be specified alone with the rotary axis in the same block, otherwise, an alarm occurs.

Note 3: The system must specify a plane again after G7.1 C0 exits the cylindrical interpolation, otherwise, the machining plane is still in the plane selected by the cylindrical interpolation;

Note 4: For the axis which is not specified by a parameter, its movement value is executed in the cylincrial interpolation mode, it does not execute the cylindrical interpolation;

Note 5: The specified feedrate in the cylindrical interpolation mode is the speed on the circumference, i.e., the extended cylindrical surface's speed;

Note 6：One rotary axis and another linear axis can execute the circular interpolation in the cylindrical interpolation mode. But the arc radius can be specified by only R instead of I, J and K. The usage of the radius R is the same that of the circular interpolation;

For example, when the circular interpolation is executed between Z and C axis, No. 1022 is set to 1 (X axis) for C axis; at the moment, the circular interpolation code is:

G18 Z__ C__;
G02(G03) Z__ C__ R__;

For C axis, when No. 1022 is set to 2, the arc code is :

G19 C__ Z__;
G02(G03) Z__ C__ R__;

Note 7：Any tool radius compensation mode being executed must be cleared before the system enters the

cylindrical interpolation mode. Start and end the tool offset in the cylindrical interpolation mode; an alarm occurs when the cylindrical interpolation is enabled in the used tool radius compensation mode;

Note 8: In cylindrical interpolation mode, the movement amount of rotary axis specified by the angle is converted into the movement distance of linear axis along outside surface, which makes rotary axis and another axis execute the linear interpolation or circular interpolation. After interpolation, the distance is converted into the angle, and the movement amount for the conversion is rounded to least input increment. So, when the diameter of the cylindrical is lesser, the actual movement amount is not equal to the specified movement amount, but the error does not accumulate.

$$\text{Actual motion amount} = \frac{\text{MOTION\_REV}}{2\times 2\pi R}\times\left[\text{command value}\times\frac{2\times 2\pi R}{\text{MOTION\_REV}}\right]$$

MOTION_REV：movement amount per rotation of rotary axis（its value is set by No.1260）；
R：Radius of workpiece；
[ ]：Round to least input increment；

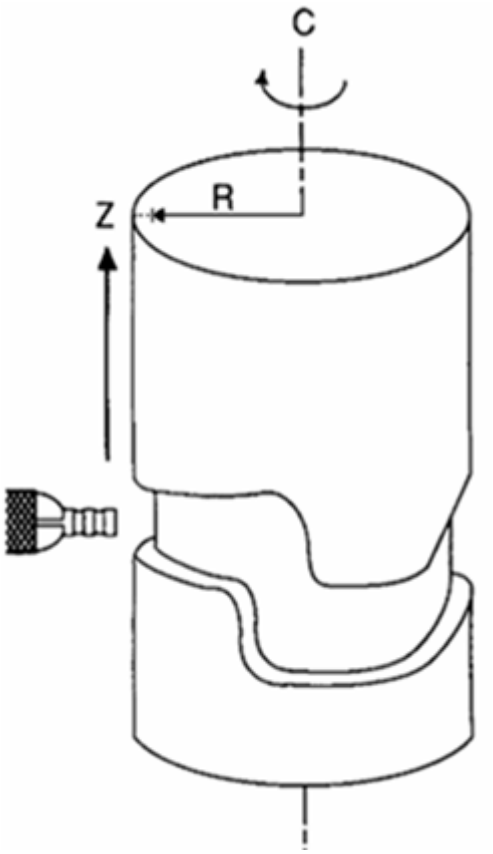
Note 9: In the cylindrical interpolation mode, an alarm occurs when the positioning operation (rapid movement code G00 and other codes to create rapid traverse, including G28, G53, G73, G74, G76, G80～G89) is specified;

Note 10: In the cylindrical interpolation mode, an alarm occurs when the workpiece coordinate system（G50，G54～G59）or the local coordinate system is specified;

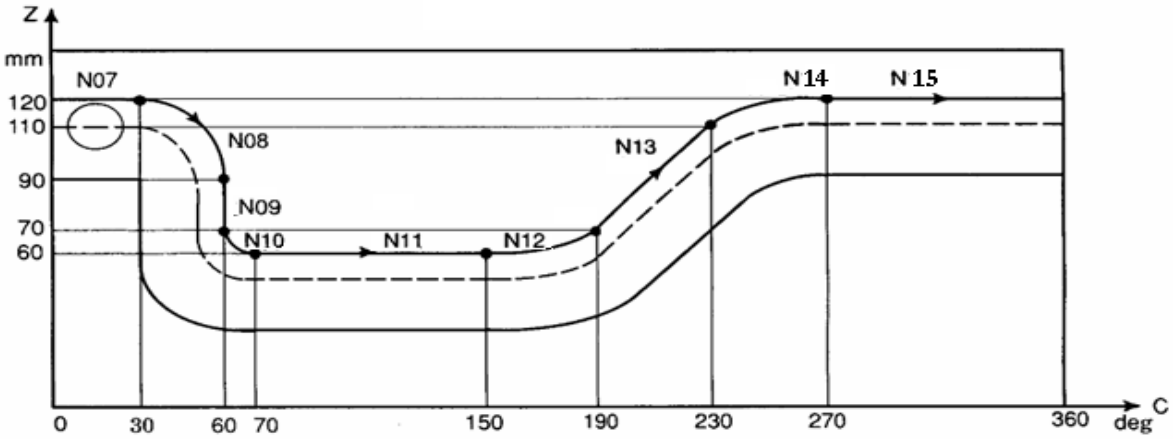
Note 11: In the cylindrical interpolation mode, the system resets to clear the cylindrical interpolation mode. It must be specified again when the system enters the cylindrical interpolation mode again;

Note 12：An offset value cannot be changed in the cylindrical interpolation mode, otherwise, an alarm occurs.

Example：




```
00001(CYLINDRICAL INTERPOLATION);
N00001 G0 Z100.0;
N00002 M14;（the spindle is switched into
                position control mode）
N00003 G28 H0;（C axis returns to zero）
N00004 G18 C0;
N00005 G7.1 C67.299;
N00006 G01 G42 Z120.0 F300;
N00007 C30.0;
N00008 G03 Z90.0 C60.0 R30.0;
N00009 G01 Z70.0;
N00010 G02 Z60.0 C70.0 R10.0;
N00011 G01 C150.0;
N00012 G02 Z70.0 C190.0 R75.0;
N00013 G01 Z110.0 C230.0;
N00014 G03 Z120.0 C270.0 R75.0;
N00015 G01 C360.0;
N00016 G40 Z100.0;
N00017 G7.1 C0;
N00018 M15;（the spindle is switched into speed control mode）
N00019 M30;
```



The above figure is the side unfolded cylindrical in the program. In the figure, when the movement amount of rotary axis (C axis) specified by the angle is converted into the distance of linear axis of outside surface of the cylindrical, and the rotary axis and another linear axis (Z axis) together execute interpolation, which is taken as the interpolation of Z-X plane coordinate system in G18 plane.

2.8 Programmable Parameter Input G10

2.8.1 Workpiece Coordinate System Offset

Command function: the assumed workpiece during programming deviates from the coordinate system actually set by G50. The expected offset amount set by the workpiece coordinate system makes the set coordinate system offset.

Command format：G10 P0 IP_;

Command explanation：P0：workpiece coordinate system's offset code

IP_：setting value of axis address and workpiece coordinate system's offset amount

An absolute code is the offset amount of the previous workpiece coordinate system; an incremental code is the offset amount of the current workpiece coordinate system.

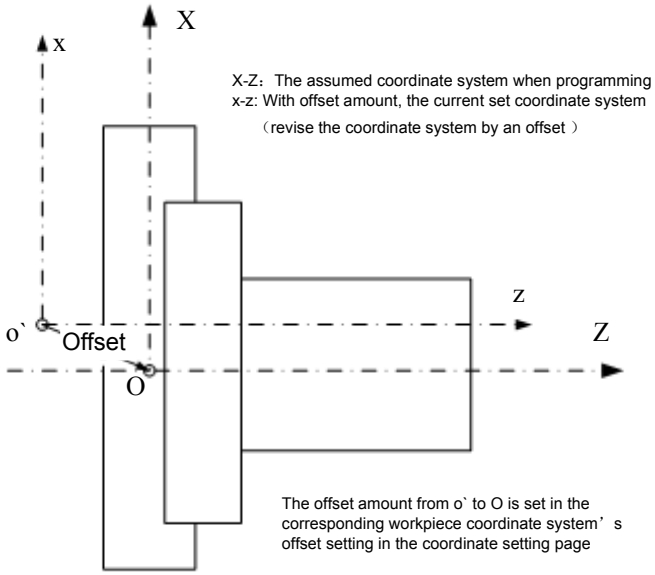
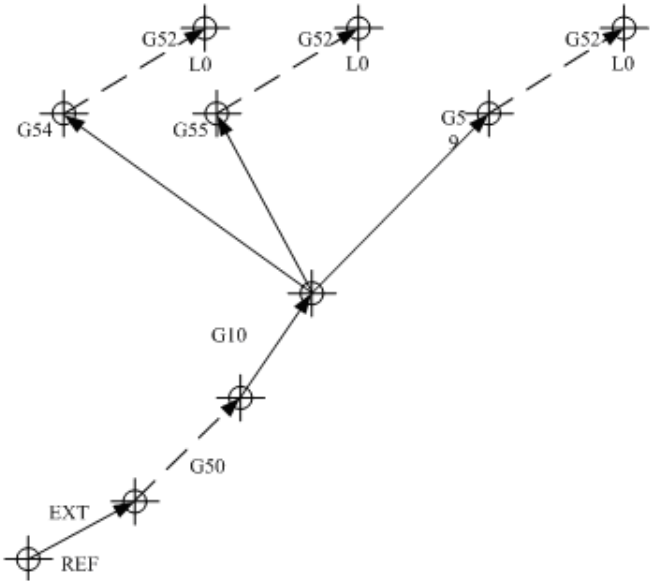


Fig. 2-10 Workpiece coordinate system offset

Relationship between coordinate offset and each coordinate:



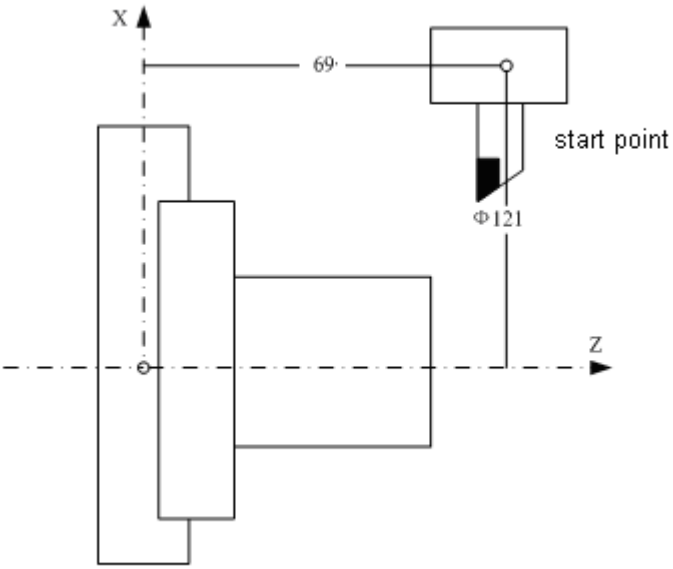
Note 1：In a program, even if X，Y，Z，C，U，V，W，H are in the same block，the later specified code is

valid for the same axis' codes.

Note 2：When the code used to setting the coordinate system is set, the set offset amount is invalid. Example） When G50X100.0Z80.0; is specified，in spite of the workpiece coordinate system's offset amount, only one coordinate system is set, which current tool's reference position is X=100.0，Z=80.0.

Note 3：After the offset amount is set, when the manual reference point return is executed, an offset amount is valid and the set coordinate system immediately offsets.

Note 4：A workpiece coordinate system's offset amount is determined by the diameter/radius assignation. Example）it is expected that the reference point should be positioned from the workpiece's origin X=Φ120.0（diameter value），Z=70.0 to the current reference point X=Φ121.0，Z=69.0，the set offset is shown below: X=1.0，Z=-1.0



2.8.2 Setting a Workpiece Coordinate System's Offset Amount

Command function: the function can replace the direct input on the MDI panel to modify the workpiece origin's offset and the workpiece coordinate system's offset value in the coordinate setting page.

Command format：G10 L2 Pp IP_;

Command explanation:p = 0：specify the external workpiece origin's offset amount；

p = 1~6：specify the workpiece origin 's offset amount relative to the workpiece coordinate system 1~6；

IP_：setting of axis' address and workpiece origin's offset amount.

An absolute code is the offset amount of each axis' workpiece origin. An incremental code is to add its value to the previous set workpiece origin's offset amount of each axis (its result is the offset amount of workpiece origin).

Note: when G10 is executed, the corresponding workpiece coordinate system's offset value is refreshed real-time, and #1220~#1226 setting values corresponded to the workpiece coordinate system are simultaneously modified.

2.8.3 Additional Workpiece Coordinate System Setting

Command function: the function can replace the direct input on the MDI panel to modify the additional workpiece coordinate system's offset value in the coordinate setting page.

Command format：G10 L20 Pn IP_;

Command explanation:Pn :set the specified code of the workpiece origin offset amount's workpiece coordinate system;

n：1~48;

IP_：setting of axis' address and workpiece origin's offset amount.

When the workpiece orgin offset amount is an absolute value, the specified value is a new offset amount. When it is an incremental value, it adding a value specified in the current set offset amount becomes a new offset amount.

Note: When G10 is executed, the offset value of the corresponding additional workpiece coordinate system is refreshed real-time.

2.8.4 Automatically Inputting a Tool Life

Command function: executing G10/G11 inputs a tool life management data

Command format：G10 L3 P_;

Command explanation: L3：tool life management function

P_：the tool life management mode cannot be mixed with the internal executed P tool life group number in G10.

P is omitted：delete all groups and log in the tool life group

P1：refresh the group data

P2：delete the group data

Delete all groups' data (P is omitted) when log in:

Format	Symbol description
G10 L3;	G10 L3: Delete all groups' data
P- L-;	P-：group number
T-;	L-：tool life value
T-;	T-：tool number and tool offset number
…….	G11：the log-in ends
P- L-;	
T-;	
T-;	
……..	
G11;	
M02(M30);	

After deleting logged-in all tool life management data, the system logs-in the programmed tool life management data.

Change the tool life management data（P1）

Format	Symbol description
G10 L3 P1； P- L-； T-； T-； …… P- L-； T-； T-； …….. G11； M02(M30)；	G10 L3: start to change the group' data P-：group number L-：tool life value T-：tool number and tool offset number G11：the log-in ends

In the non logged-in tool life management data group, set the tool life management data or change the logged-in tool life manage data.

Delete the tool life management data （P2）

Format	Symbol description
G10 L3 P2； P- ； P- ； P- ； P- ； ……. G11； M02(M30)；	G10 L3 P2: start to delete the group data P-：group number G11：the deletion ends

Set the tool life group's count type

Format	Symbol description
G10 L3 ； （G10 L3 P1）； P- L- Q-； T-； T-； ……. G11； M02(M30)；	Q: life count type （1：times, 2：time）

Note: when Q is omitted, the life count type is based on LTN(No. 6800#2) seting value.

2.8.5 Setting a Tool Offset Value

Command function：the function can replace the direct input on the MDI panel to modify the tool wear and all tool geometry offset in the tool offset setting page.

Command format：G10 P_ X_ Y_ Z_ ；

G10 P_ U_ V_ W_;

Command explanation:

Code	Description
P	The code value is an offset number P = 1~99 ： the tool wear offset value code P = 10001~10099 ： the tool geometry offset value code
X	X offset value (absolute);
Y	Y offset value (incremental);
Z	Z offset value (absolute);
U	X offset value (absolute);
V	Y offset value (incremental);
W	Z offset value (incremental);

In the absolute code, values specified in the address X，Y，Z and R are taken as the offsetvalue corresponded to the offset number specified by P.

In the incremental code, values specified in the address X，Y，Z and R should be added to the current offset value corresponding to the offset number.

Note 1: X, Y, Z, U, V and W are specified in the same block.

Note 2: Using the code in a program can permit the tool to feed point by point. Also, using the code can input an offset value one at a time from continuously specifying the code's program instead of inputting these offset value one at a time from MDI mode.

2.9 Polar Coordinate Interpolation G12.1, G13.1

Command function: the contour is controlled by the programming code in the rectangle coordinate system being switched into one linear motion (tool motion) and one turn motion (workpiece turn motion). The function is used to end face cutting.

Command format: G12.1; enter the polar coordinate interpolation mode, also be written to G112;

```
----------;
----------;
----------;
----------;
----------;
----------;
----------;
----------;
```

The following commands can be commanded in polar coordinate interpolation mode:
G01：linear inerpolation;
G02, G03：circular interpolation;
G04：dewell ;
G40, G41, G42：tool nose radius compensation;
G65, G66, G67：user macro program command;
G98, G99：feed per rev, feed per minute;

G13.1; cancel the polar coordinate interpolation mode, also be written to G113;

Command explanation: G12.1, G13.1, are specified by an single block.

Before executing the polar coordinate interpolation, the system must firstly set a linear axis androtary axis performing the polar coordinate interpolation by NO.5460, NO.5461. The polar coordinate interpolation plane consists of a linear axis and a rotary axis, and the polar coordinate interpolation is performed in the plane, which is shown in Fig. 2-11.

In the polar coordinate interpolation mode, the system codes the linear interpolation or the circular interpolation by absolute programming or incremental programming, and also by the tool nose radius compensation.

In the polar coordinate interpolation mode, F feederate is the speed which is tangent with the polar coordinate interpolation plane (rectangular coordinate system).

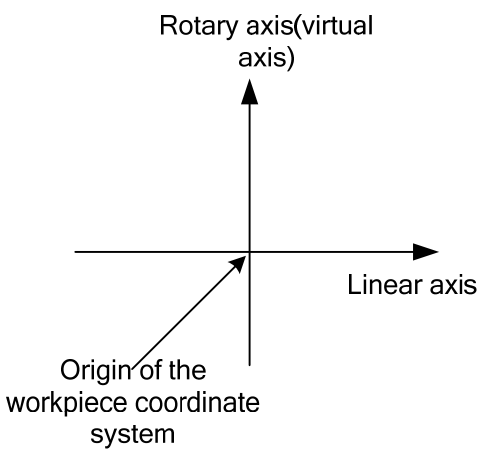


Fig. 2-11 polar coordinate interpolation plan

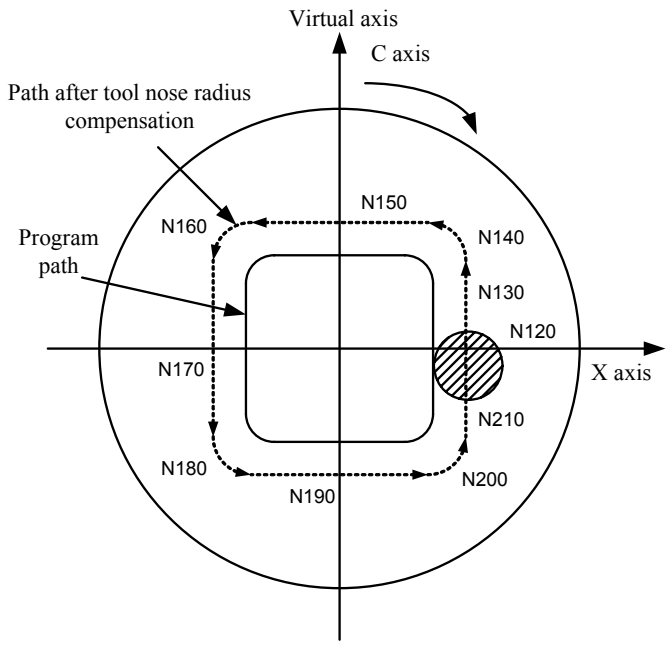


Fig. 2-12

Example: a polar coordinate interpolation program based on X axis（linear axis）and C axis (a rotary axis) (Fig.2-12)

X axis uses the diameter programming and C axis uses the radius programming (programming unit: mm, and the display unit: degree.)

```
O0001;
N10   T0202
……..;
N100 G00 X150. C0 Z0;
N110 G12.1;
N120 G42 G01 X80. F200;
N130 C20.0;
N140 G03 X40.0 C40.0 R20.0;
N150 G01 X-40.0;
N160 G03 X-80.0 C20.0 R20.0;
N170 G01 C-20.0;
N180 G03 X-40.0 C-40.0 R20.0;
N190 G01 X40.0;
N200 G03 X80.0 C-20.0 R20.0;
N210 G01 C0;
N220 G40 X150.0;
N230 G13.1;
N240 Z100.0;
```

…….;
N500 M30;

Note 1: When the system is turned on or resets, the polar coordinate interpolation is cancelled (G13.1); G12.1 and G13.1 are modal;

Note 2: The axis undefined by the parameter does not execute the polar coordinate interpolation in spite of specifying the movement value in the polar coordinate interpolation mode;

Note 3: The used plane (selected by G17, G18 or G19) before G12.1 is cancelled; after G13.1 cancels the polar coordinate interpolation, the plane recovers; when the system resets, the polar coordinate interpolation is cancelled and the system uses the plane;

Note 4: In the polar coordinate interpolation mode, the program codes use the rectangular coordinate code in the polar coordinate plane. The linear axis in the plane uses the diameter or radius programming and the turn axis uses the radius programming;

Note 5: The arc interpolation executing the arc radius address is determined by the linear axis of the interpolation plane in the polar coordinate interpolation plane as follows:
Use I and J when the linear axis is X or its parallel axis, and the turn axis uses J;
Use J and K when the linear axis is X or its parallel axis, and the turn axis uses J;
Use K and I when the linear axis is Z or its parallel axis, and the turn axis uses I;
Also use R code;

Note 6: Must set a workpiece coordinate system before using G12.1, the center of the turn axis is the origin of the coordinate system. The coordinate system must not be changed in G12.1 mode.

Note 7: Cannot start or cancel the polar coordinate interpolation mode; code G12.1 or G13.1 in G40; otherwise, an alarm occurs;

Note 8: When the tool traverses near to the workpiece center in the polar coordinate interpolation mode, C weight of feedrate changes, which exceeds max. C cutting speed to cause an alarm;

Note 9: The program code uses the rectangular coordinate code in the polar coordinate plane. The axis address of the turn axis is taken as the one of the 2nd axis (imaginary axis) in the plane.

Note 10: The current position displays the actual coordinates in the polar coordinate interpolation. However, the remainder distance is displayed according to the coordinates in the polar coordinate interpolation plane (rectangular coordinate plane);

Note 11: When the system executes G12.1, the tool position of the polar coordinate interpolation starts from the angle 0. So, the spindle must be positioned before the polar coordinate interpolation is executed;

Note 12: Must not switch the spindle gear in the polar coordinate interpolation. The system must be in the spindle speed control mode when the gear shifting is needed.

2.10 Metric/Inch Switch G20, G21

Command function: G code selects the metric or inch system.
Command format: G20; inch input
G21; metric input
Command explanation: G20/G21 must be specified in a single block before a program begins to set a coordinate system.
After metric/inch switch G code is specified, unit of input data is changed into least inch/metric

input increment of incremental system ISB or ISC. Angle unit does not change.

The units of the following value will change after they switch between the metric and the inch.

——F feedrate;
——position code;
——zero offset of workpiece;
——tool compensation value;
——scale unit of MPG;
——movement in incremental feed;
——some parameters.

Note 1: The modal G20/G21 in group 06 can be set to initial mode by NO. 0000 BIT2 (INI).

Note 2: The tool compensation value must input the incremental unit and set it again in metric/inch switch. The tool compensation value can automatically change and cannot be set again when NO.5006 Bit0 is 1.

Note 4: It modifies NO.0000 Bit2 (INI) when the system executes G20/G21. The displayed mode also changes when NO.0000 Bit 2 (INI) is changed.

Note 5: Display digit number and mode of absolute coordinate and relative coordinate are set by No.0000 Bit2 (INI), and display digit number and mode of machine coordinate are separately determined by No.1001 Bit0 (INM) AND No. 3104 Bit0 (MCN).

2.11 Stored Travel Check G22, G23

Command function: Create the forbidden area of stored travel limit check 2 and limit the tool traverse range in one area.

Command format: G22; stored travel 2 check is turned on
---------;
---------;
G23; stored travel 2 check is turned off

Command explanation: G22: stored travel check is turned on;
G23: stored travel check is turned off;
Positive coordinates of the stored travel area is set by No.1322;
Negative coordinates of the stored travel area is set by No. 1323;

Limit area figure: taking examples of X, Y, Z limit area are as follows. X, Y, Z are positive coordinates, I, J, K are negative.

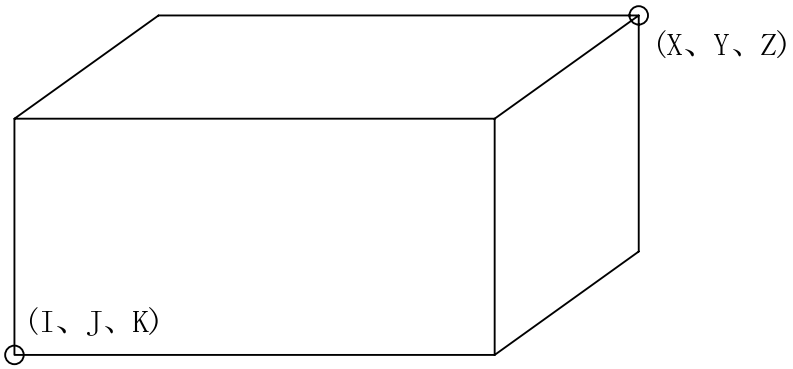


Fig.2-13

Note 1: The modal G22/G23 in group 9 can be set by No. 3402 Bit 7(G23);

Note 2: G22 stored travel check is limited to the stored travel limit check 2, and the detailed is referred to OPERATION;

Note 3: The data is set by the distance (least code increment is taken as the unit) to the reference position when the parameter sets the top point of the forbidden area;

Note 4: Whether the limit range is the inner side or outer side of the area is set by No. 1300 Bit0 (OUT) , and it is the inner side when it is set to 0;

Note 5: The limit is valid after the system executes the reference position return; the system alarms when the reference position is in the limit area in G22 mode;

Note 6: The tool reversely traverses when the travel alarm occurs, and the alarm is cleared after reset again;

Note 7: G22/G23 is executed in an alone block, and an alarm occurs when it and other G codes or MST are in the same block;

Note 8: When the system is switched from G23 to G22 in the forbidden area, an alarm occurs;

Note 9: When No.1310 Bit 0(OT2x) of the stored travel limit check 2 is set to 1(executing the stored travel limit 2 check), the system executes G22 and then the check; the system does not execute the check when it is G23;

2.12 Skip Interpolation G31

Command function: In the course of executing the code, when the outside skip signal (X0.4) is input, the system stops the code to execute the next block.

The function is used to the dynamic measure (such as milling machine), toolsetting measure and so on of workpiece measure.

Command format: G31 IP_ F_;

Command explanations: non-modal G code (00 group); its address format is same that of G01; Cancel the tool nose radius compensation before using it; feedrate should not be set to too big to get a precise stop position;

The following block execution when skipping:

1. The next block of G31 is the incremental coordinate programming, which is shown in Fig. 2-14.

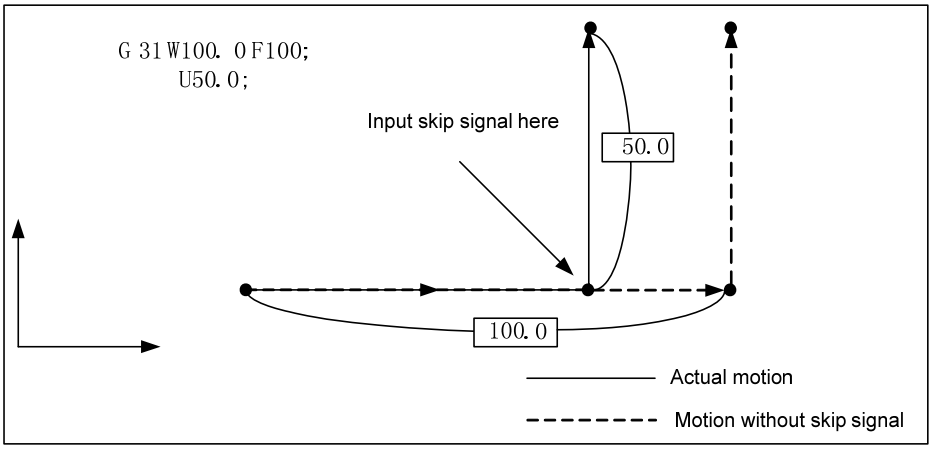


Fig.2-14

2. The next block of G31 is the absolute coordinate programming of one axis, which is shown in Fig. 2-15.

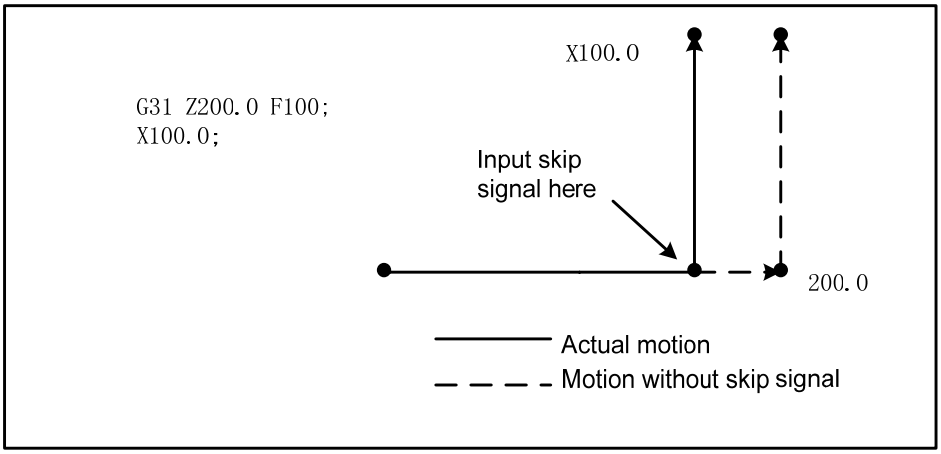


Fig.2-15

3. The next block of G31 is the absolute coordinate programming of two axes as Fig. 2-16.

Programming：G31 Z200 F100
G01 X100 Z300

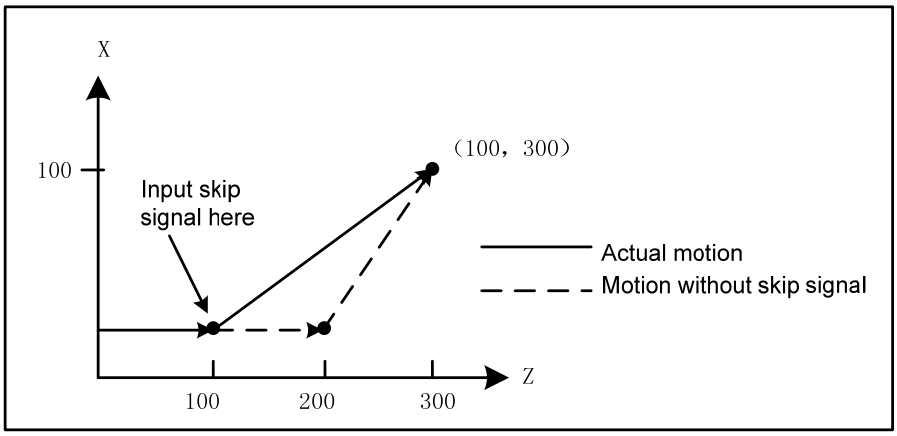


Fig. 2-16

Skip signal explanation:

SKIP signal (SKIP): X0.4

Type: input signal

Function: X0.4 ends the skip cutting. I.e. in a block containing G31, the skip signal becoming the absolute coordinate position of "1" is to be stored in the macro variable (#5061～#5065, its last bit digit corresponds to the No. n axis of the system), at the same time, the movement in G31 block ends. No. 6200 Bit1 (SK0) sets the invalid input state of the skip signal, and when it is set to 0, the input signal 1 is valid.

Operation: When the skip signal becomes "1", CNC executes as follows: When the block is executing G31, CNC stores the current absolute coordinates for each axis. CNC stops G31 to execute the next block, the skip signal detects its state instead of its RISING EDGE. So when the skip signal is "1", it meets the skip conditions.

Note 1: When the skip signal is input, the feedrate override, the dry run, and automatic acceleration/ deceleration are invalid in the course of movement by the skip function, which is to improve the tool positioning precision. Set No. 6200 Bit7(SKF) to 1 to make these function valid.

Note 2: The skip signal is valid, the CNC immediately stops the feed axis (without acceleration/ deceleration execution), and G31 feedrate should be as low as possible to get the precise stop position.

2.13 Automatic Tool Offset G36, G37

Command function: When the code is executed to make the tool move to the measured position, the CNC system automatically measures the difference between the current actual coordinates and the code coordinates to be the tool offset value. The function is used to the automatic toolsetting.

Command format: G36 X__;
G37 Z__;

Explanations: X absolute coordinate(only used to G36), Z absolute coordinate (only used to G37);
Non-modal G code (00 group);
Cancel the tool nose radius compensation before using it;
Only use the absolute programming;
Specify the tool offset number before using the code;

Measure position arrival signal:

XAE(X0.6) ————corresponding to G36

ZAE(X0.7) ————corresponding to G37

Type: input signal

Function: When the position measured by the program code is different from that where the tool actually reaches (i.e. at the time, the measured position arrival signal becomes the state set by No.6240#0), the difference of the coordinates is added to the current tool compensation value to update the compensation value. When G36X_(or G37Z_) is executed, the tool firstly rapidly traverses to the position γ measured by the code, and decelerates and temporarily stop the position before the measured position, and then, reaches to the measured position at the speed set by No.6241 (or No.6242). When the measured position arrival signal corresponding to G code becomes the state set by No. 6240#0, and the tool is in the measured position range ±ε, the system updates the offset

compensation value and ends the block. When the measured position arrival signal does not become "1", and after the tool reaches the measured position distance ε, the CNC alarms, ends the block and does not update the offset compensation value.

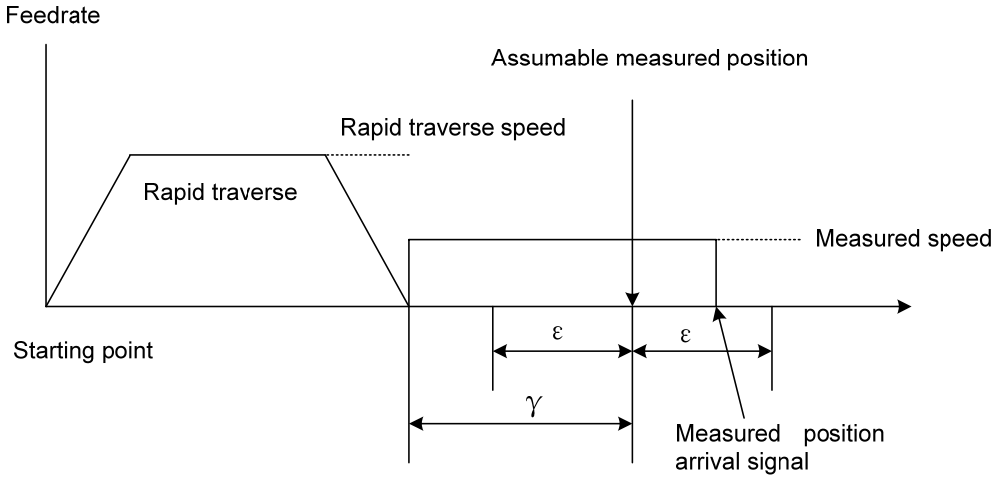
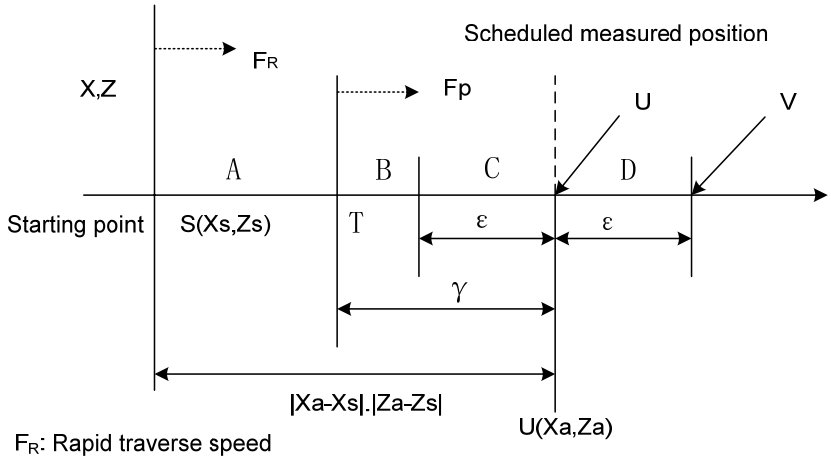


Fig.2-17

G36, G37 automatic tool offset code use

From the initial position to the measured position specified by Xa (or Za) in G36(or G37), the tool rapidly traverses to A zone and stops at T point (Xa–γx (or Za–γz) , and then traverses to B, C and D at the feedrate set by No.6241(or No.6242). The system alarms when the tool traverses in B zone and the measured point arrival signal of the end point is set to. The system alarms when the tool stops at V point. No. 6241, No. 6242, No.6254 and No.6255 are set by the radius value.



F_R: Rapid traverse speed
Fp: Feedrate set by No. 6241
γ: Parameter (No.6251, No.6252)
ε : Parameter (No.6254, No.6255)

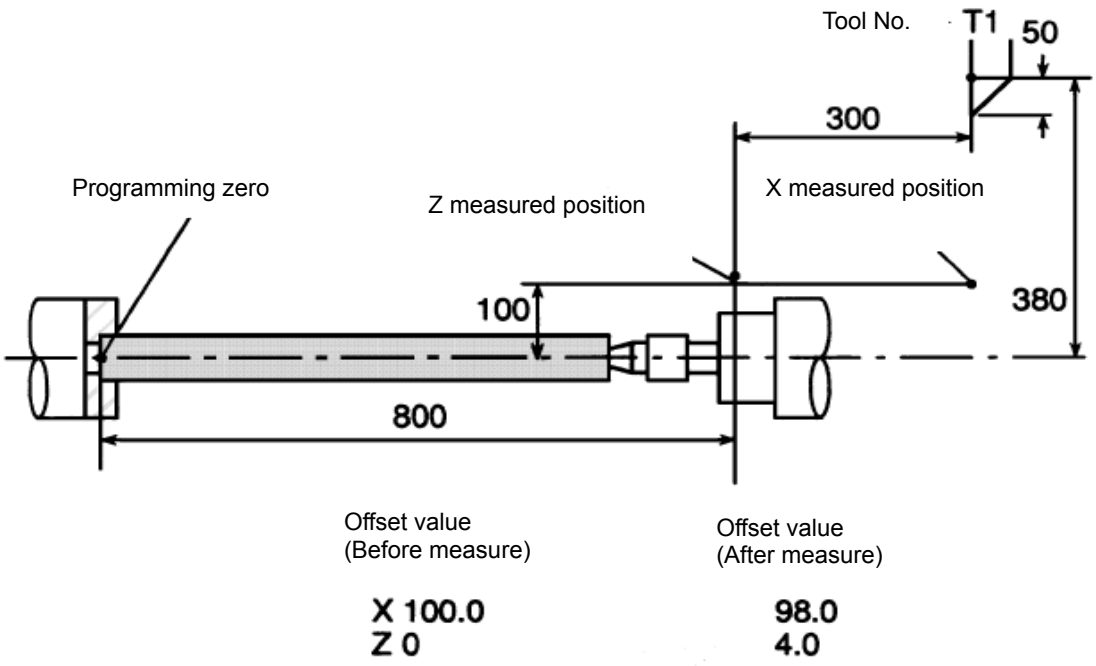
Fig.2-18

Example:

G50 X760 Z1100;	must have created a workpiece coordinate system
T0101;	define No. 1 tool and execute its tool compensation
G36 X200;	traverse to X toolsetting point (X toolsetting point coordinate: 200)
G00 X204；	retract a little
G37 Z800;	traverse to Z toolsetting point (Z toolsetting point coordinate: 800)

T0101; again get an offset value

M30;



2.14 Reference Point Function

2.14.1 Reference Point Return G28

Command function: move from the start point at the rapid traverse speed to the middle position specified by IP_ and then return to the reference point.

Command format: G28 IP__ ;

Command explanation: G28 is non-modal.

IP_: it is the middle point code (absolute value code /incremental value code). The system can omit one or all code address for each axis. Omitting some axis means the axis does not return to the reference point, omitting all axes means the tool does not move.

Code execution process: (as Fig. 2-19):

（1）Rapidly position from the current position to the middle point of the code axis(A→B);

（2）Rapidly position from the middle point to the reference position (B→R);

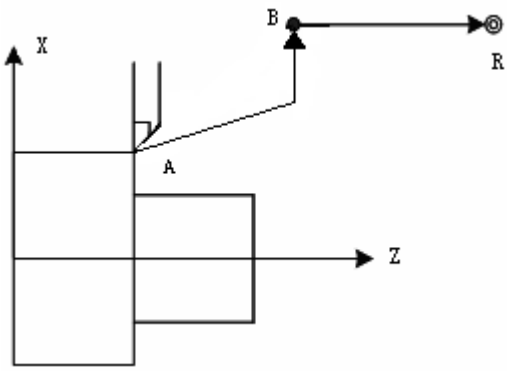


Fig.2-19

Note 1: After the system is turned on, it does not execute the manual reference position return; when the system executes G28 reference position return, it judges it alarms or executes like the manual reference position return according to No. 1002 Bit 3(AZR) to use the deceleration block to execute the reference position return. But, when the reference position setting function without the block(No.1002 Bit1 (DLZ)) is set to 1 or NO.1005 Bit 1(DLZx) is set to 1, it is unrelated to AZR setting, an alarm occurs when the system executes G28 before the reference position is created.

Note 2: Each axis separately moves at the rapid traverse speed from the start point through the middle point to the reference position.

Note 3: G28 or G30 in the tool radius compensation mode automatically cancels the tool radius compensation, and automatically recovers it in the next movement code.

Note 4: No. 5003 Bit2 (CCN) selects executing G28 cancel tool compensation mode on the middle point or after the reference point return.

Note 5: Generally, G28 is specified in an alone line; when the system specifies simultaneously the same parameter address word of G00 or G01, IP_ is specified to G28 parameter, G00 or G01 only change the modal value of the corresponding G groups and does not create a motion.

2.14.2 2nd, 3rd, 4th Reference Point Return

Command function: traverse at the rapidly traverse speed to the middle point specified IP_ and then to the 2nd, 3rd and 4th reference position.

Command format: G30 P2 IP__ ; return to the 2nd reference position

G30 P3 IP__ ; return to the 3rd reference position

G30 P4 IP__ ; return to the 4th reference position

Command explanation: G30 is non-modal;

IP_: it is the middle point code (absolute value code /incremental value code). The system can omit one or all code address for each axis. Omitting some axis means the axis does not return to the reference position, omitting all axes means the tool does not move.

Code execution process (as Fig.2-19):

（1）Rapidly position from the current position to the middle position of the code axis(A→B);

（2）Rapidly position from the middle point to the reference position (B→R);

Note 1: A reference point position is set in NO.1241～NO.1243;

Note 2: After the system is turned on, it executes the reference position return once before executing G30; do not execute the reference position return firstly before executing G30 after the system with the absolute encoder is turned on;

Note 3: When P is omitted, the system executes it as P2 and returns to the 2nd reference position;

Note 4: Each axis rapidly moves at individual speed from the current start point to the middle, and from the middle point to the reference point.

2.15 Relevant Functions of Coordinate System

The tool position is expressed with a coordinate value of the coordinate system. GSK988TA/TB system has three kind of coordinate system:

1. machine coordinate system, 2. workpiece coordinate system, 3. local coordinate system

Fig.2-20 describes the relationship of the three coordinate systems:

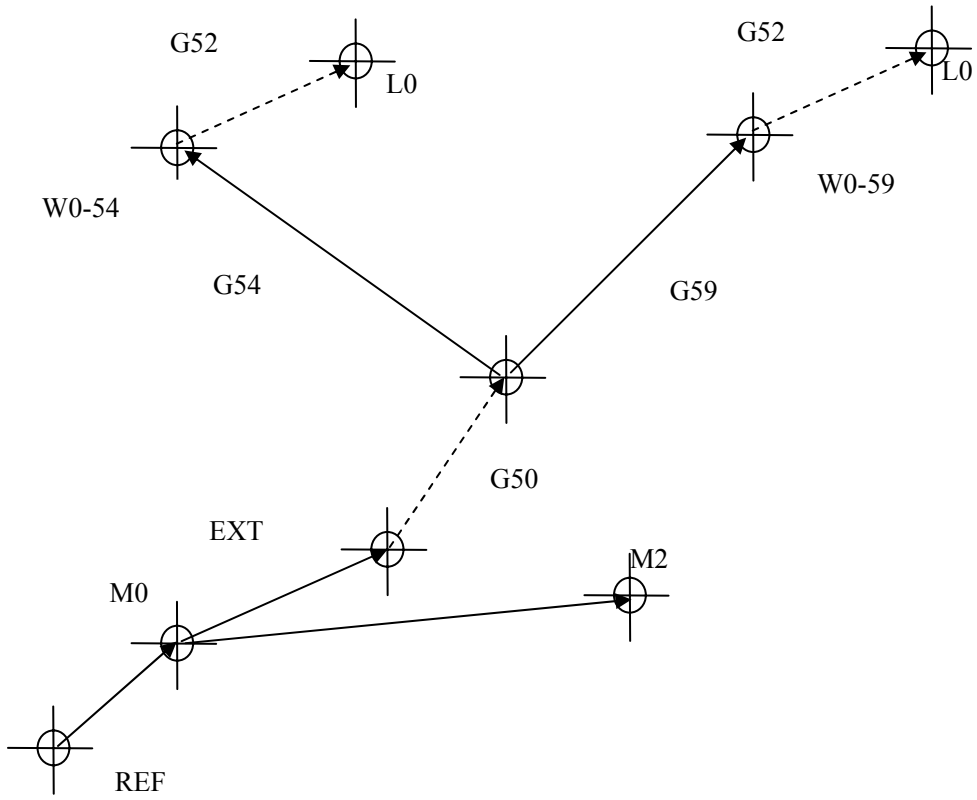


Fig.2-20

REF	Reference position.
M0	Origin of machine coordinate system is a fixed point on the machine, No. 1240 value confirms the relative position of the reference position and the machine origin.
M2	The 2nd reference position, No.1214 set the 2nd reference position in the machine coordinate system.
EXT	The outer origin offset can be set by No. 1220 or in the coordinate setting window.
G50	The offset set by G50 is 0 when the system is turned on.
G54, 59	The offset of the workpiece coordinate system is set by No. 1221, No. 1226, and is also set in the coordinate window.
W0-54, W0-59	Origin of the workpiece coordinate system.
G52	The offset of the local coordinate system is 0 when the system is switched on. All workpiece coordinate systems share, i.e. the local coordinate system offset set in one workpiece coordinate system can exist in other workpiece coordinate system.
L0	Origin of the local coordinate system.

Note: The system has created the above coordinate system after the reference point return is executed firstly. The coordinate system is created after the system is turned on with the absolute position encoder.

2.15.1 Selecting Machine Coordinate System Position G53

A particular on the machine as the machining reference is called the machine zero which is taken as the origin of the coordinate system is called as the machine coordinate system. After the system is turned on, executing the manual reference position return sets the machine coordinate system which

keeps till the system is turned off.

Command function: when the position of the machine coordinate system is executed, the tool traverses to the position at the rapid traverse speed.

Command format: G53 IP ;

Command explanation: G53 is non-modal;

IP_: coordinate values of each axis in the machine coordinate system must bespecified by the absolute value.

Execution process: As the following figure: the specified axis rapidly moves from A (10, 20) in the current workpiece coordinate system to point B (-8, -10) in the machine coordinate system.

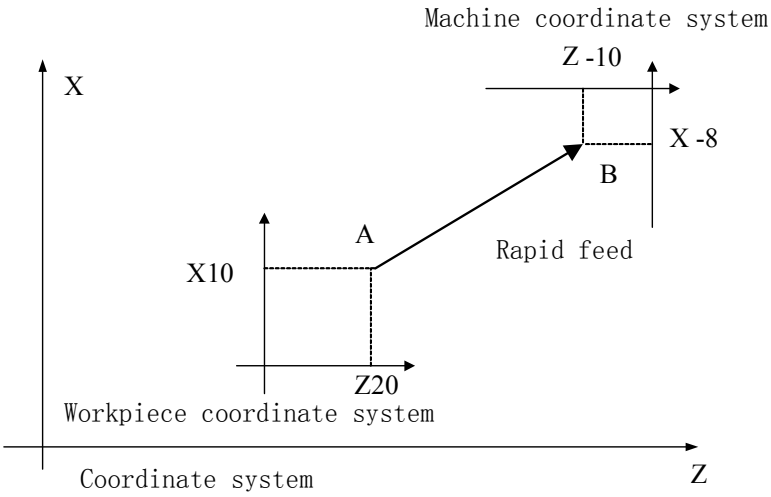


Fig. 2-21

Note 1: G53 is non-modal, and is valid in other blocks;

Note 2: G53 specifies the absolute position value in the machine coordinate system. The axis code is ignored when some axis uses the incremental value code; omitting one of axis means the axis does not move. When G53 is specified but other axes' position is not specified, other axes do not move;

Note 3: When G53 is executed, the system cancels the tool nose radius compensation;

Note 4: After the system is turned on, the system performs the manual reference position return or G28 automatic reference position return, and automatically creates the origin position of the machine coordinate system according to the value set by No. 1240;

Note 5: The machine coordinate system must be set before the system codes G53. So, the system must execute the manual reference position return or G28 automatic reference position return after it is turned on; the operation is not operated when the system uses the absolute position encoder;

Note 6: The system executes G53 and G00, G01 in Group 01 in the same block, G00 or G01 only modifies G modal value in Group 01.

2.15.2 Workpiece Coordinate System Setting G50

A coordinate system used to machining a workpiece is called a workpiece coordinate system. A set workpiece can set again the position of workpiece coordinate system by changing its origin position. G50 is used to setting a workpiece coordinate system in A set of G code system and G92 is used to setting a workpiece coordinate system in B set of G code system.

Command function: absolute coordinates of the current position can be set by setting the

absolute coordinates of current position to create the workpiece coordinate system (called as the floating coordinate system). After the workpiece coordinate system is created, the absolute coordinate programming inputs the coordinate value in the coordinate system till the new workpiece coordinate system in G50 is created.

Command format: G50 IP__ ;

Command explanation: G50 is non-modal G;

IP_: when the system uses the absolute code, it specifies the new absolute coordinate position of the current point in the coordinate system; when the system uses the incremental code, after its executes G50, the absolute coordinate value of the current point is equal to the sum between the absolute coordinate value before execution and the coordinate incremental value.

Note 1: After G50 changes the workpiece coordinate system, other workpiece coordinate systems also perform the same offset;

Note 2: In G50, the system can omit one or all code addresses for each axis, the current coordinate value is not input when the code value for each axis is not input. When the axis code address is omitted, the coordinate axis which is not input keeps its pervious coordinate value;

Note 3: When G50 and G codes (G00, G01) in Group are in the same block, the system only modifies the modal value of Group 1, and the coordinate value in the block is specified by G50;

Note 4: When the system does not set G50 offset value, it can set No. 1202 Bit(G50) to forbid G50;

Note 5: After G50 sets the coordinate system, the system must be turned off and then on, the coordinate values set by G50 remain unchanged before power off.

Note 6: In NC program, when LGT is set the coordinate offset mode to execute the tool offset, and the system executes T function does not execute the absolute value code, the coordinate system is set by G50, the absolute coordinate value displayed by G50 is the one that the coordinate value set by G50 adding the tool compensation value which is not executed. The difference between the relative coordinates and the machine coordinates is （-80，10）when the system executes N4, the difference value is caused because X100Z10 setting G50X20Z20 to create the workpiece coordinate system offset, i.e. the user does not think over the tool offset influence when G50 is set in NC program.

Program	Absolute coordinates	Relative coordinates	Machine coordinates
N1 T0100 G00 X100 Z10	X：100 Z：10	X：100 Z：10	X：100 Z：10
N2 T0101（No.01 tool offset value X12 Z23）	X：88 Z：-13	X：100 Z：10	X：100 Z：10
N3 G50 X20 Z20	X：8 Z：-3	X：20 Z：20	X：100 Z：10
N4 G00 X10 Z10	X：10 Z：10	X：22 Z：33	X：102 Z：23

2.15.3 Workpiece Coordinate System Selection G54～G59

Command function: One of G54～G59 is specified, one of workpiece coordinate system 1～6 can be selected. After the workpiece coordinate system is specified, the specified point in the block is in the specified workpiece till a new workpiece coordinate system is created as Fig. 2-21. The tool positions X60.0, Z20.0

in the workpiece coordinate system 3.

G56 G00 X60.0 Z20.0

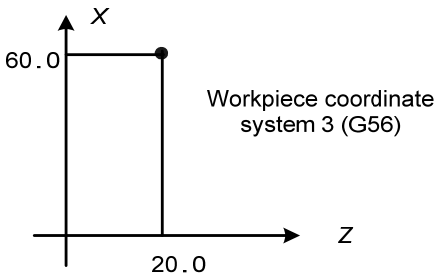


Fig. 2-22

Command format: G54 workpiece coordinate system 1;
G55 workpiece coordinate system 2;
G56 workpiece coordinate system 3;
G57 workpiece coordinate system 4;
G58 workpiece coordinate system 5;
G59 workpiece coordinate system 6;

Command explanation: G54～G59 are modal.

Note 1: A workpiece coordinate system is valid after a reference point is created.
When the system is turned on, No.1201 Bit 5 (EWZ) sets to memery the workpiece coordinate system or not. When EWZ is set to not to memory, the workpiece coordinate system defaults to G54 after power-on.
When reset, No.1201 Bit7 (WZR) sets the system returns to G54 workpiece coordinate system or not. But when No.3402 Bit 6(CLR) is set to 1, the mode in the group also returns to G54.

Note 2: G54-G59 describing the 6 workpiece coordinate systems can change their positions by the external workpiece zero offset value or workpiece zero offset value, and their relationship is as Fig. 2-23;

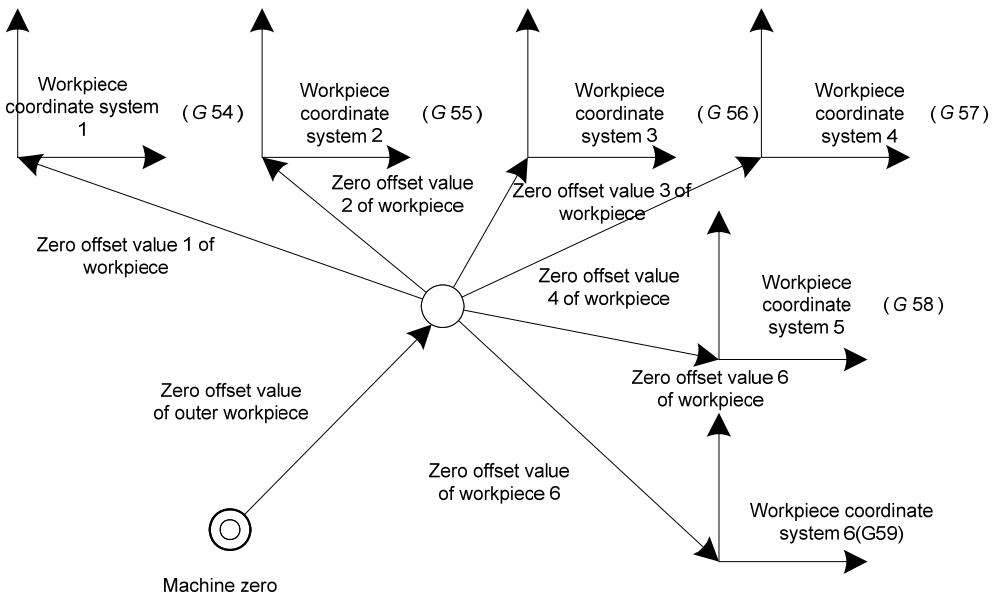


Fig. 2-23

Note 3: Use the following method to change:

1）MDI input changes the workpiece coordinate system zero;

2）Use G50 to move the workpiece coordinate system;

Specifying G50 IP_ makes the workpiece coordinate system（G54～G59）to set a new workpiece coordinate system where the current tool position is consistent with the specified coordinates. When G50 specifies the relative value, the value adding the previous tool position coordinate value creates a new coordinate system, but the tool position does not change but the coordinate system executes the offset as Fig. 2-24:

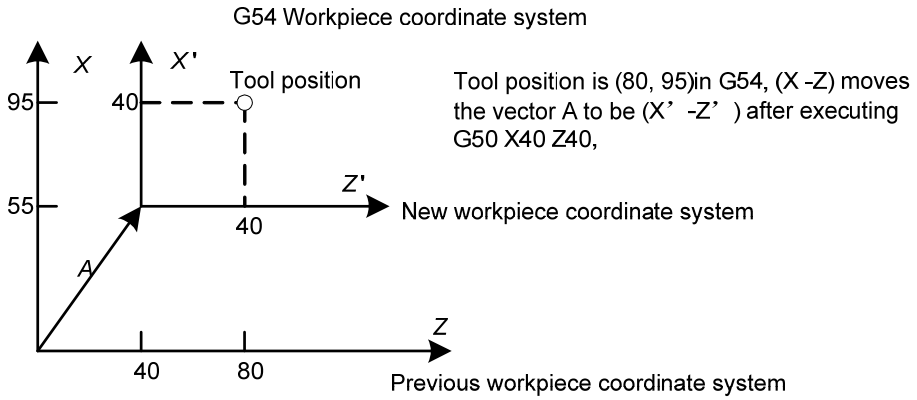
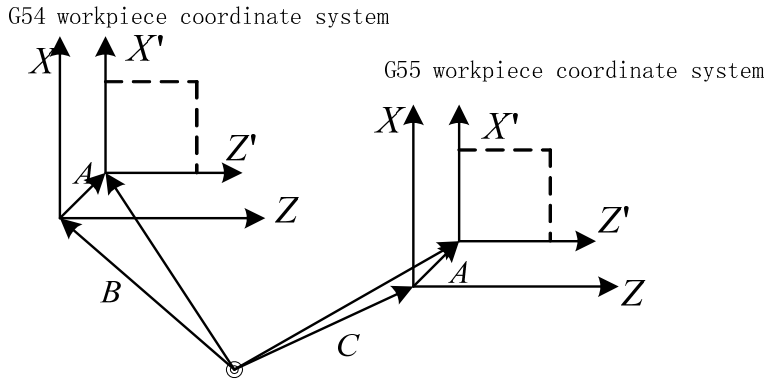


Fig.2-24

Note 4: The coordinate offset value created by G50 adds to the one of all workpiece zero to make ensure that all workpiece coordinate systems offset are the same value as Fig. 2-25:



$X'-Z'$, New workpiece coordinate system

$X-Z$, Previous workpiece coordinate system

A: $G50$ Offset value

B: Zero offset value of workpiece in G54

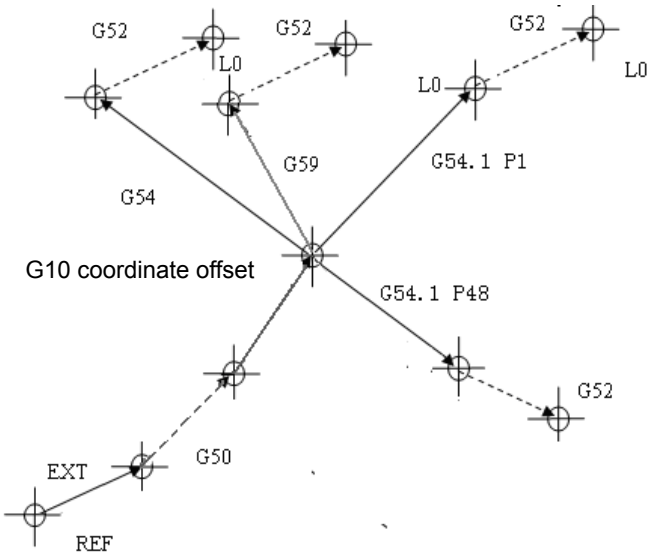
C: Zero offset value of workpiece in G55

Fig. 2-25

Note 5: The workpiece zero offset value of G54～G59 workpiece coordinate system can be set in the parameters and input in the coordinate setting window;

2.15.4 Additional Workpiece Coordinate System G54.1

Command function: The system uses 6 standard workpiece coordinate systems (G54~G59) and can use 48 additional workpiece coordinate systems.



Command format: G54.1 Pn ;
G54 Pn ;

Command explanation: Pn：specified the codes for additional workpiece coordinate systems
n : 1～48

When P and G54.1 (G54) are specified together, the system selects an additional workpiece coordinate system 1～48 according to P code. Once the workpiece coordinate system is selected, it is still valid till another workpiece coordinate system is selected. When the system is turned on, the system selects the standard workpiece coordinate system 1(using G54 to select it).

G54.1 P1 —— select additional workpiece coordinate system 1
G54.1 P2 —— select additional workpiece coordinate system 2
……
G54.1 P48 —— select additional workpiece coordinate system 48

The workpiece origin's offset amount of the additional workpiece coordinate system is the same that the standard workpiece coordinate system, which can execute the following operations:

① execute the display and setting by the setting page of workpiece origin offset amount.
② read and write a value by a user macro program's system variable.
③ input a workpiece origin's offset amount by the external data input.

Note 1：P after G54.1 (G54) is specified. When P after G54.1 in the same block is not specified, the system selects an additional workpiece coordinate system 1 (G54.1 P1).
Note 2: An alarm (PS0003) occurs when the specified P value exceeds its range.
Note 3: When G54.1(G54) and another code (such as G04, M98) with P code are in the same block, the two codes are simultaneously valid and P value does not exceed the offset number.
Example 1） G54.1 G04 P1000; an alarm occurs (PS0003).
Example 2） G54.1 M98 P48; is normally executed.

2.15.5 Local Coordinate System Setting G52

To be convenient to programming, the sub-coordinate system to set the workpiece coordinate system is called the local coordinate system.

Command function: executing G52 IP_； in the program can set the local coordinate system in

the workpiece coordinate system G54~G59. The origin of the local coordinate system can set in the position specified by IP_ in the workpiece coordinate system. The corresponding relationship is as Fig. 2-26.

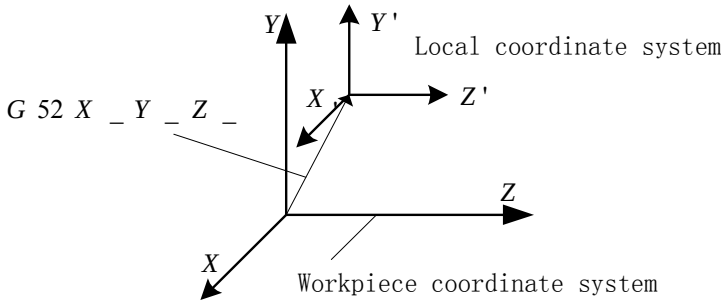


Fig. 2-26

Command format: G52 IP__; set the local coordinate system

……

G52 IP0; cancel the local coordinate system

Command explanation: G52 is non-modal;

IP_: when IP_ is absolute code, the system specifies the absolute coordinate value of origin of local coordinate system in the workpiece coordinate system; when IP_ is the incremental code, the execution result is the same that of the absolute code;

Once the local coordinate system is created, its coordinates are used to the axis motion code. Using G52 to code the zero of the new local coordinate system in the workpiece coordinate system can change the position of the local coordinate system. making the zero of the local coordinate system coincide with the one of the workpiece coordinate system can cancel the local coordinate system and returns to the workpiece coordinate system, i.e. code G52 X0 Z0 or G52 U0 W0.

Note 1: The local coordinate system setting does not change the workpiece coordinate system and the machine coordinate system;

Note 2: Executing G52 can temporarily cancel the offset in the tool nose radius compensation;

Note 3: In local coordinate system, when G50 sets the workpiece coordinate system and the system has not specified the coordinate values to all axes in the local coordinate system, the axis which is not specified in G50 in the local coordinate system still keeps, the local coordinate system corresponding to G50 axis is cancelled; For example:

G52 X50 Z50;

……

G50 X100; at the moment, Z coordinate value is not change, the local coordinate system corresponding to X is cancelled

Note 4: When the system selects the workpiece coordinate system code （G54～G59）to change the workpiece coordinate system in the local coordinate system, the local coordinate system also moves to the new workpiece coordinate system.

Note 5: Whether the local coordinate system in reset is cancelled is determined by No.1202 Bit 3(RLC) , the local coordinate system is cancelled in reset when the parameter is set to 1.

Note 6: Whether the local coordinate system in manual reference position return is cancelled is determined by No.1201Bit 2 (ZCL), the local coordinate system is cancelled in manual reference position return when the parameter is set to 1. When G28 and G30 execute the reference point return, the system does not cancel the local coordinate system.

2.16 Plane Selection Code G17～G19

Command function: G code selects to execute the circular interpolation and the tool nose radius compensation plane.

Command format: G17 selects XpYp plane;
G18 selects ZpXp plane;
G19 selects YpZp plane;

Command explanation: G17, G18, G19 are modal G codes.
Xp: X or its parallel axis
Yp: Y or its parallel axis
Zp: Z or its parallel axis

Note 1: Xp, Yp, Zp are determined by the axis addresses of G17, G18, G19 in the block; when the axis addresses are omitted, the system defaults the omitted are the addresses of the basic axis; the plane keeps when the system does not code G17, G18, G19 blocks.
Note 2: The parameter (No. 1022) sets each axis to have three basic axes (X, Y, Z) or the parallel axis.
Note 3: The plane remains unchanged in the G17, G18, G19 not be specified.
Note 4: When the system is turned on, its initialization is defaulted to G18 state, i.e. ZX plane;
Note 5: When the system repetitively specifies G17～G19 in the same block, and No.3403 Bit 6(AD2) is 0, the last G17～G19 word is valid, the system alarms when the parameter is set to 1;
Note 6: The multi-compound cycle code（G70～G76） and the fixed cycle code（G90, G92, G94）are used to ZX basic axis plane; when their functions are specified in other planes, an alarm occurs;
Note 7: The motion code is not related to the plane selection, besides the arc interpolation and tool nose radius compensation code, when the system codes the axis beyond the planes, no alarm exists and the axis can move; when the system selects the axis motion beyond the plane in the arc interpolation code, the system defaults it executes the spiral interpolation.

For example:

```
G17 X_ Y_ ；select XY plane
G17 A_ Y_ ；select AY plane
G18 X_ Z_ ；select ZX plane
G17；       select XY plane
G17 A_      select AY plane
G18 Y_      select ZX plane, Y motion is not relative the plane
```

2.17 Exact Stop Mode G61/Cutting Mode G64

G61 function: the programmed axis of a block must exactly stop at the end point of the block, and the system continuously executes a next block.

G64 function: the system executes a next block while the programmed axis of each block after G64 starts to decelerate (the axis does not reach the programmed end point). The programmed contour in G64 mode is different from the actual contour, and the different degrees is determined by F value and the angle between two paths, bigger their difference is, F value is bigger.

Command format: G61; （exact stop mode）
G64; （cutting mode, defaulted to default value）

Command explanations:

1. A block including G61 exactly stops the end point of the program before the system executes the next block, which is used to process sharp edges and corners. G61 is modal and valid before G64 is executed. The programmed contour is the same that of the actual.
2. G64 is modal, valid and default before G61 is executed. G64 path is different from that of G61 as Fig. 2-27;
3. G61, G64 belong to Group 15, and their relations with other G groups are referred to Group 5.
4. When G0 is executed, it is in the exact stop in cutting mode because it is non cutting code.
5. When G61 /G64 is specified, it is value in the next executed block.

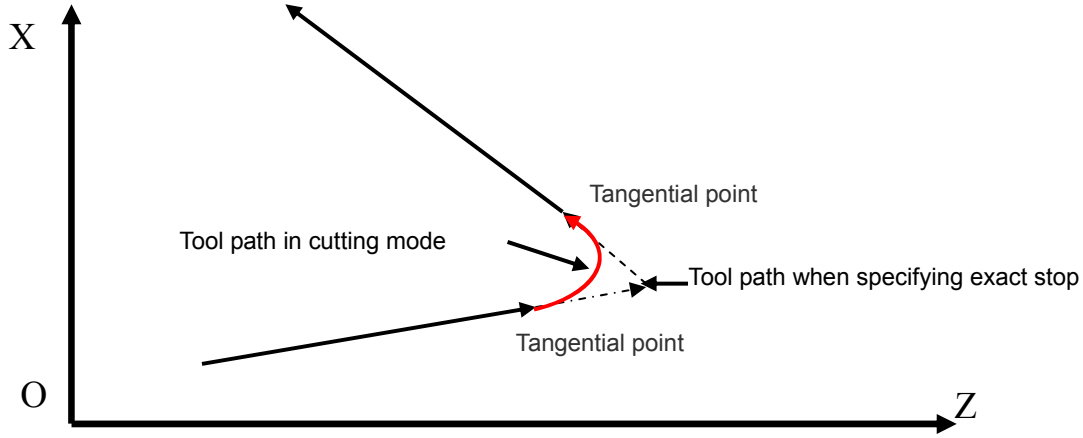


Fig. 2-27

2.18 Fixed Cycle Code

To simplify programming, GSK988TA/988TA1/988TB defines G code of single machining cycle with one block to complete the rapid traverse to position, linear/thread cutting and rapid traverse to return to the start point:

G90: axial cutting cycle; G92: thread cutting cycle; G94: radial cutting cycle;

Note: G92 thread cutting fixed cycle code is described in Section Thread Function.

2.18.1 Axial Cutting Cycle G90

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

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

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

Code specifications: G90 is modal;

X_,Z_	Coordinates of longitudinal cutting (C point in the figure below)
U_,W_	Movement to end point (C point in the figure below) of longitudinal cutting
F_	Cutting feedrate
R_	Taper（radius value, with direction, range referred to the table below

Address	Incremental system	metric（mm）input	Inch (inch) input
R	ISB system	-99999.999mm～99999.999mm	-9999.9999 inch～9999.9999inch
	ISC system	-9999.9999mm～9999.9999mm	-999.99999 inch～999.99999inch

Cycle process:

① X rapidly traverses from start point A to cutting start point B;

② Execute the linear interpolation from the cutting start point B to cutting end point C;

③ X executes the tool retraction at feedrate, and return to the position which the absolute coordinates and the start point D are the same;

④ Z rapidly traverses to return to the start point A and the cycle is completed.

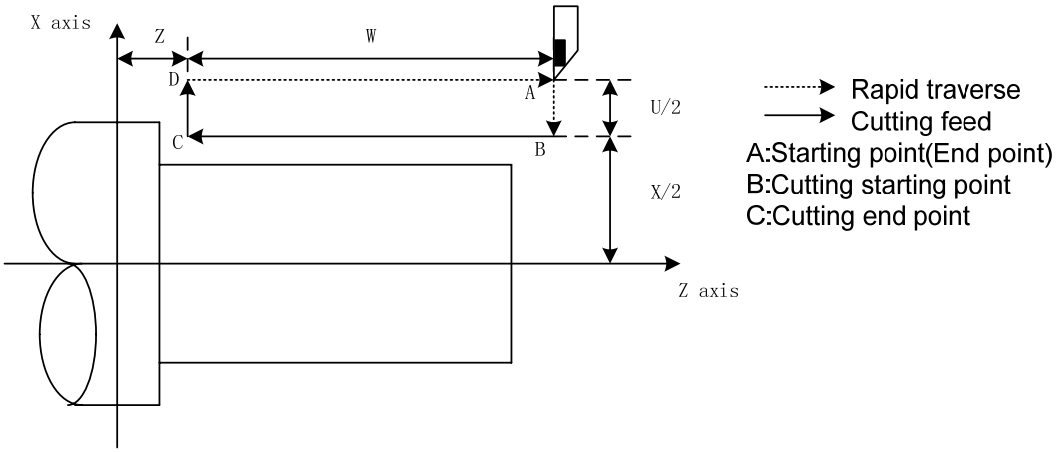


Fig.2-28

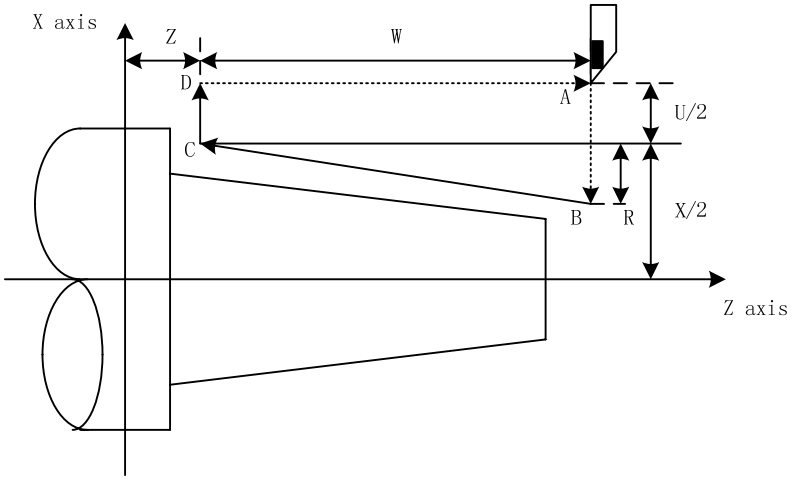


Fig. 2-29

Cutting path: Relative position between cutting end point and start point with U, W, R, and tool path of U, W, R with different sign symbols are as Fig. 2-30:

1） U>0，W<0，R>0　　2） U<0，W<0，R<0

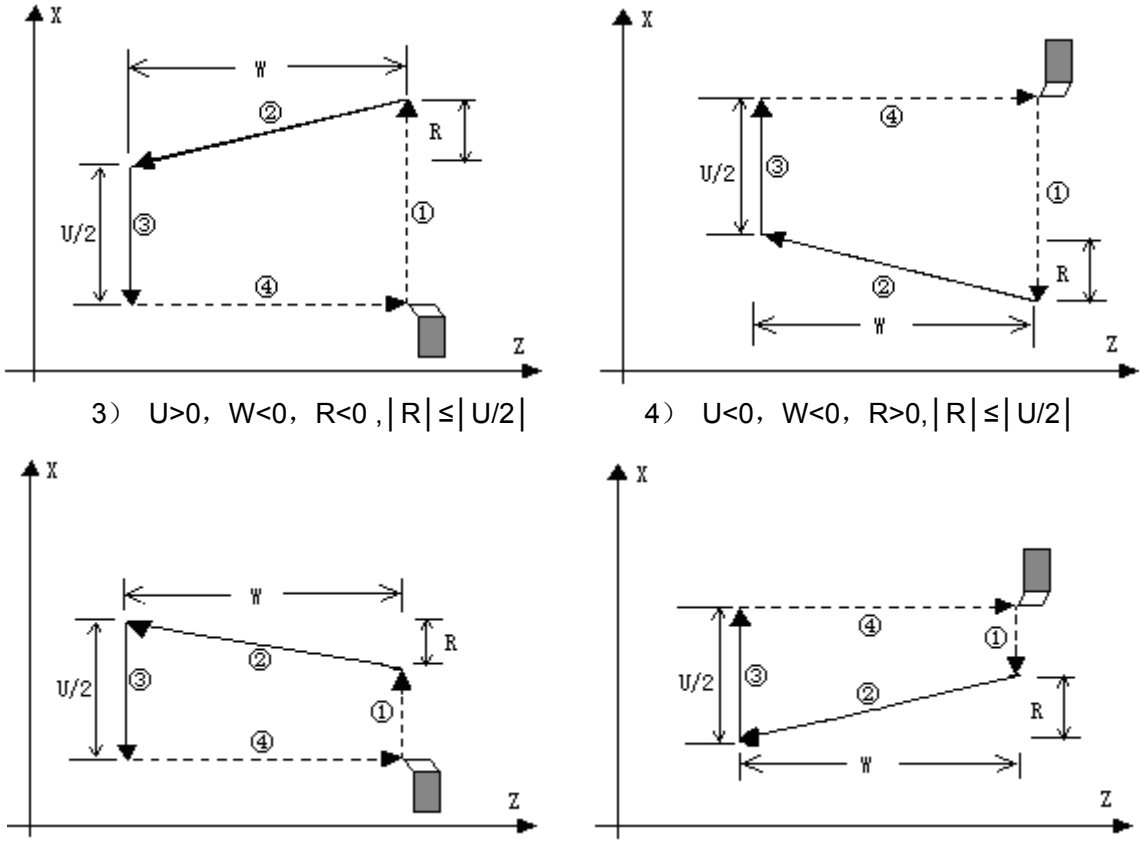


Fig.2-30

Example: Fig. 2-31, workblank Φ125×110

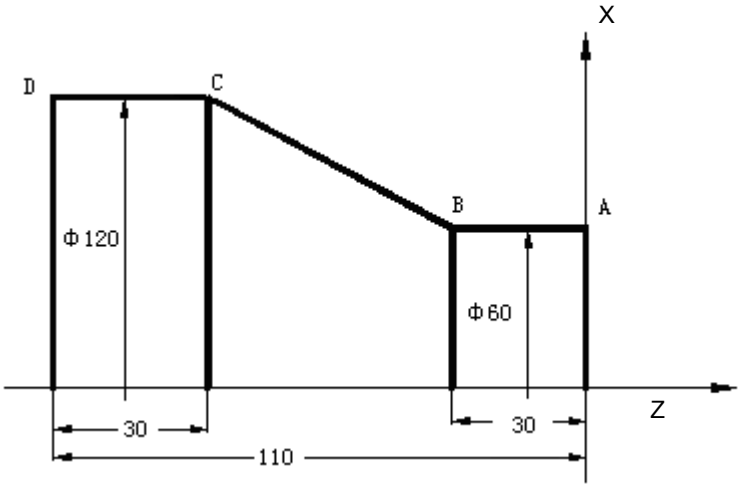


Fig.2-31

Program：

M03 S300 G0 X130 Z3；

G90 X120 Z-110 F200； （A→D，Φ120 cut）

```
X110 Z-30;
X100;
X90;        (A→B, 6 times cutting cycle Φ60, increment of 10mm)
X80;
X70;
X60;
G0 X120 Z-30;
G90 X120 Z-44 R7.5 F150;
Z-56 R-15;  (B→C, taper cutting, B→C, 4 times tool infeed cutting)
Z-68 R-22.5;
Z-80 R-30;
M30;
```

2.18.2 Radial Cutting Cycle G94

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

Command format(A set) : G94 X(U) __ Z(W) __ F__; (face cutting)
G94 X(U) __ Z(W) __ R__ F__; (taper face cutting)

Command format(B set) : G79 X__ Z__ F__; (face cutting)
G79 X__ Z__ R__ F__; (taper face cutting)

Code specifications: G94/G79 is modal;

X_,Z_	Coordinate of cutting end point(C point in the figure below)in the direction of the bottom side
U_,W_	Movement to cutting end point (C point in the figure below)in the direction of bottom side
F_	Cutting feedrate
R_	Taper（radius value, with direction, range referred to the table below

Incremental system	Metric（mm）input	Inch (inch) input	Incremental system
R	ISB system	-99999.999 mm～99999.999mm	-9999.9999 inch～9999.9999inch
	ISC system	-9999.9999 mm～9999.9999mm	-999.99999 inch～999.99999inch

Cycle process:

① Z rapidly traverses from start point A to cutting start point B;

② Execute linear interpolation from the cutting start point B to cutting end point C;

③ Z executes the tool retraction at the cutting feedrate (opposite direction to the above-mentioned ①), and returns to the position which the absolute coordinates and the start point D are the same;

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

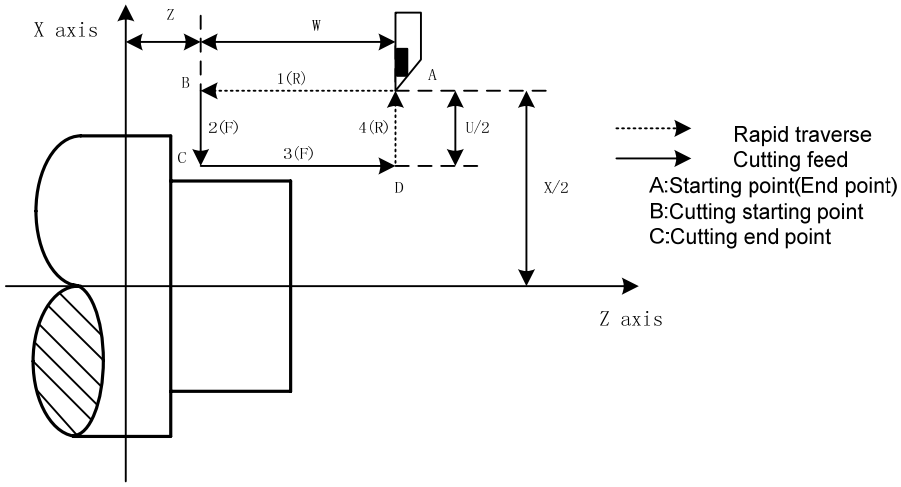


Fig. 2-32

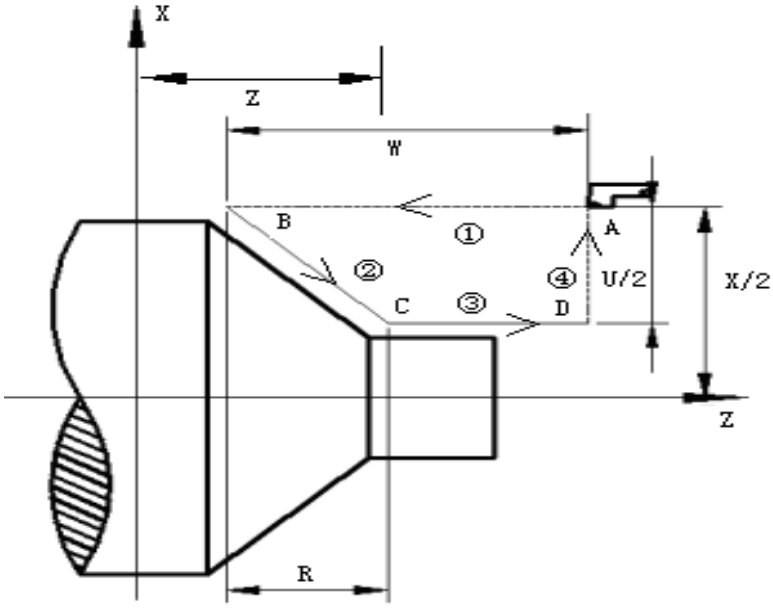
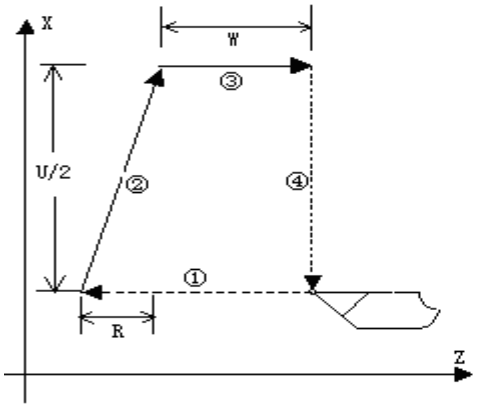


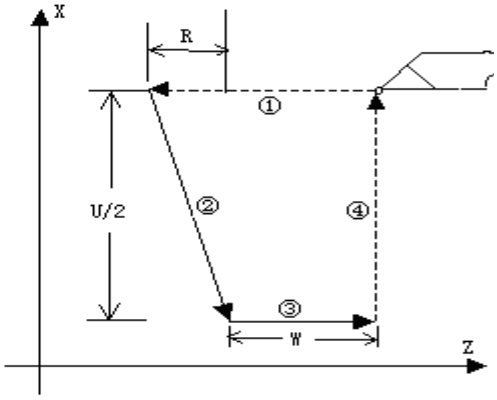
Fig.2-33

Cutting path: Relative position between cutting end point and start point with U, W is as Fig.2-34:

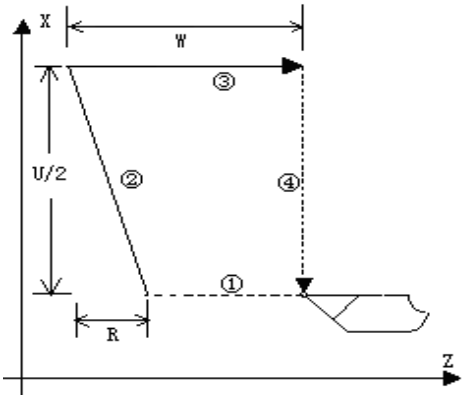
1） U>0 W<0 R<0



2） U<0 W<0 R<0



3）U>0 W<0 R>0 （｜R｜≤｜W｜）



4）U<0 W<0 R>0 （｜R｜≤｜W｜）

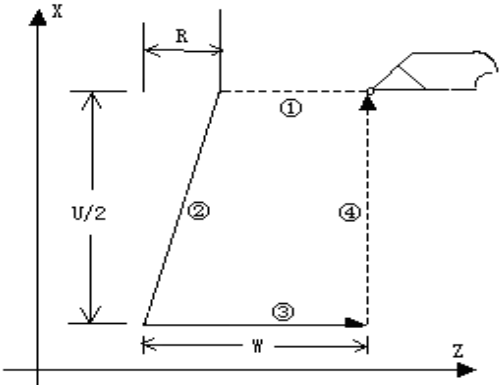


Fig.2-34

Example: Fig. 2-35，workblank Φ125×112

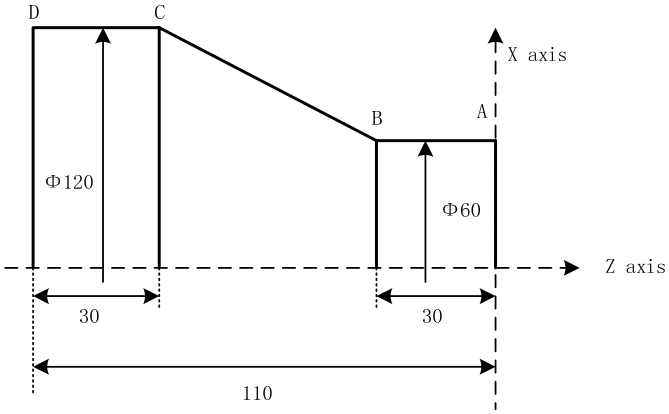


Fig.2-35

Program:

```
G00 X130 Z5 M3 S500;
G94 X0 Z0 F200;
X120 Z-110 F300;        End face cutting（cut outer Φ120）
G00 X120 Z0;
G94 X108 Z-30 R-10;
X96 R-20;
X84 R-30;               （C→B→A，Φ60 cut）
X72 R-40;
X60 R-50;
M30;
```

Note 1: The fixed cycle code is used to only ZX plane. An alarm occurs when other axis' motion in the fixed cycle code's blocks is executed;

Note 2: After X(U) , Z(W) , R are executed in the canned cycle code, their code values are value if X(U) , Z(W) ,R are not redefined by executing a new canned cycle codes. The code values of X(U) , Z(W) , R are cleared if non-modal G code (Group 00) or other G codes in Group 01 except for G04 is executed;

Note 3: In MDI mode, a fixed cycle which is the same with the previous canned cycle can be executed without inputting the code again after the run is completed;

Note 4: The previous cycle operation of the fixed cycle is executed repetitively when the next block immediately follows EOB (;) or there is a null line in G90～G94, but when the following block immediately follows M, S, T code, the system does not repetitively execute the previous cycle operation.

Example: …

N010 G90 X20.0 Z10.0 F400；

N011 ； （here, execute G90 one time again）

…

Note 5: The single block is executed in G90, G94, the single block stops after the whole cycle of the current block is completed.

2.19 Multiple Cycle Codes

GSK988TA/TB multiple cycle codes include 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 codes, 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 start point and the end point of code are the same one.

Note: G76 multiple thread cutting cycle code is described in *Thread Function*.

2.19.1 Axial Roughing Cycle G71

G71 roughing cycle type is divided into two: type Ⅰ and type Ⅱ. X, Z in type Ⅰ must monotonely increase or decrease; in type Ⅱ, X can be non-monotone and Z must monotonely increase or

decrease.

Command function: G71 is divided into three parts:

(1): blocks for tool infeed value and tool retraction value, the cutting feedrate, the spindle speed and the tool function when roughing;

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

(3): 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, and the roughing is completed by multiple executing the cutting cycle tool infeed→ cutting→tool retraction. The start point and the end point are the same one. The code is applied to the formed roughing of non-formed rod.

Command format：

Type 1：

```
G71 U (△d) R (e) F   S   T   ;        (1)
G71 P (ns) Q (nf) U(△u)W(△w);         (2)
N(ns) G0/G1 X(U);
……;
……F;                                  (3)
……S;
 N(nf) ……;
```

Type Ⅱ：

```
G71 U (△d) R (e) F   S   T   ;        (1)
G71 P (ns) Q (nf) U(△u)W(△w);         (2)
  N(ns) G0/G1 X(U) Z(W);
  ……;
  ……F;                                (3)
  ……S;
  N(nf) ……;
```

Code specifications:

1. ns～nf blocks in programming must be followed G71 blocks. If they do not follow closely G71 blocks, after the system executes roughing, it executes from the next block of G71;
2. ns block belongs to G00, G01 in group 01. When ns block does not contain Z(W), it is type 1; when it contains Z(W), it is type 2;
3. In ns～nf blocks, for type 1, X, Z dimension must monotonously change (always increase or decrease); for type 2, Z dimension must monotonously change;
4. ns～nf blocks are used to count the roughing path and the blocks are not executed when G71 is executed. F, S, T codes of ns～nf blocks are invalid when G71 is executed, at the moment, F, S, T codes of G71 blocks are valid. F, S, T of ns～nf blocks are valid when executing ns～nf to code G70 finishing cycle;
5. In ns～nf blocks(without ns block), there are only G codes: G01, G02, G03, G04, G96, G97, G98, G99, G40, G41,G42 and the system cannot call subprograms(M98/M99);
6. G96, G97, G98, G99, G40, G41, G42 are invalid in G71 and valid in G70, G96, G97, G98;
7. When G71 is executed, the system can stop the automatic run and executes the manual move, but returns to the position before manual traversing when G71 is executed again, otherwise, the following path will be wrong;
8. When the system is executing the feed hold or single block, the program pauses after the

system has executed end point of current path;

9. △d，△u are specified by the same U and different with or without being specified P,Q codes;
10. G71 cannot be executed in MDI, otherwise, an alarm occurs.

Relevant definitions:

Finishing path	As Fig. 2-36, Part (3) (ns～nf block)defines the finishing path, and the start point of finishing path (start point of ns block)is the same these of start point and end point of G71, called A point; the first block of finishing path(ns block)is used to 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 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（radius value）of X tool infeed in roughing without sign symbols, and the direction of tool infeed is defined by move direction of ns block. The code value Δd is reserved after executing U（Δd）and the value of NO.5132 is rewritten. The value of system parameter NO.5132 is regarded as the travel of tool infeed when U（Δd）is not input
e	It is travel（radius value）of X tool retraction in roughing（radius value）withoutsign symbols, and the direction of tool retraction is opposite to that of tool infeed, the code value e is reserved and the value of system parameter NO.5133 is rewritten after R（e）is executed. The value of system parameter NO.5133 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	X finishing allowance range is as the following table (diameter) with sign symbols. X coordinate offset of roughing path compared to finishing path, i.e. the different value of X absolute coordinates between A' and A. The system defaults Δu=0 when U（Δu）is not input, i.e. there is no X finishing allowance for roughing cycle
Δw	Z finishing allowance range is as the following table (diameter) with sign symbols. X coordinate offset of roughing path compared to finishing path, i.e. the different value of X absolute coordinates between A' and A. The system defaults Δw=0 when U（Δw）is not input, i.e. there is no Z finishing allowance for roughing cycle
M, S T, F	F：Cutting feedrate; S: Spindle speed; T: Tool number, tool offset number 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 only G70 finishing blocks.

Address	Incremental system	metric（mm）input	inch(inch) input
U（Δd）	ISB system	0.001~99999.999	0.0001~9999.9999
	ISC system	0.0001~9999.9999	0.00001~999.99999
R（e）	ISB system	0～99999.999	0～9999.9999
	ISC system	0～9999.9999	0～999.99999
U（Δu）	ISB system	-99999.999～99999.999	-9999.9999～9999.9999
	ISC system	-9999.9999～9999.9999	-999.99999～999.99999
W（Δw）	ISB system	-99999.999～99999.999	-9999.9999～9999.9999
	ISC system	-9999.9999～9999.9999	-999.99999～999.99999
P（ns）	ISB system	1～99999	1～99999
	ISC system	1～99999	1～99999
Q（nf）	ISB system	1～99999	1～99999
	ISC system	1～99999	1～99999

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. 2-36: B→C for finishing path, B'→C' for roughing path and A is the tool start point.

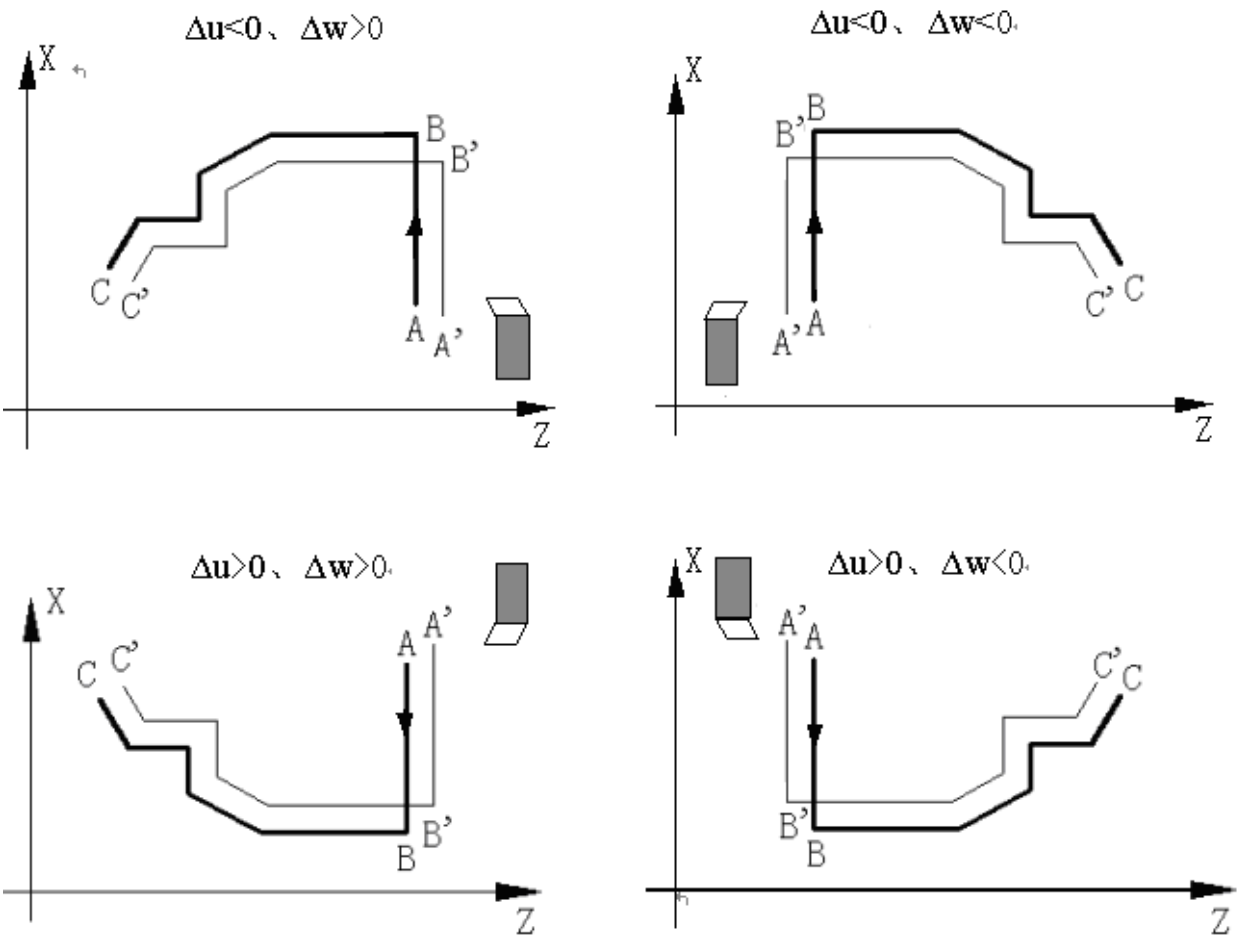


Fig.2-36

Execution process: as Fig. 2-37. G71 execution process in type 1：Fig. 2-37.

1. Rapidly traverses to A' from A point, X movement is Δu, and Z movement is Δw;
2. X moves 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. Z executes the cutting feeds to the roughing path, and its direction is the same that of Z coordinate A→B point;
4. X, Z execute the tool retraction e (45°straight line) at feedrate, the directions of tool retraction is opposite to that of too infeed;
5. Z rapidly retracts at rapid traverse speed to the position which is the same that of Z coordinate;
6. After executing X tool infeed (Δd+e)again, 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 X executes the tool infeed to B' point, and then the next step is executed;
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.

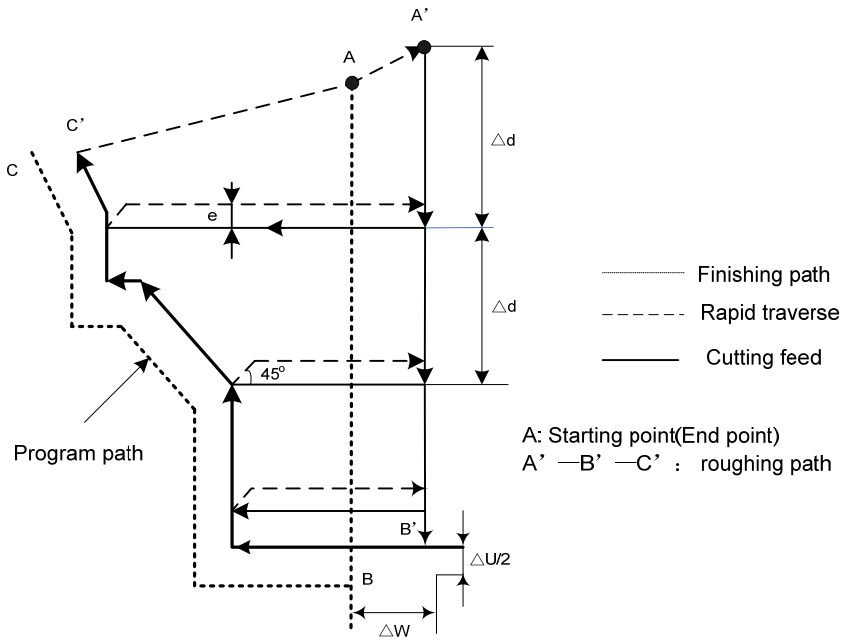


Fig. 2-37 G71 cycle path

Example:

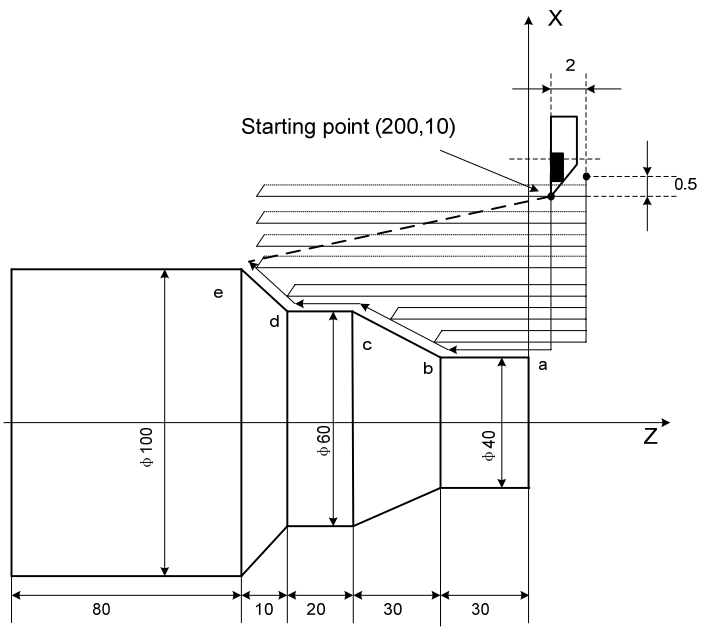


Fig.2-38

Program：O0004;

G00 X200 Z10 M03 S800; (Spindle clockwise with 800 rev/min)

G71 U2 R1 F200; (Cutting depth each time 4mm，tool retraction 2mm [in diameter])

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

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)

G71: type 2

Direction of the shape in the 2nd axis of the plane (X axis in ZX plane) is not necessary to monotonous rise or fall, and there may be up to 10 groovings, which is shown below:

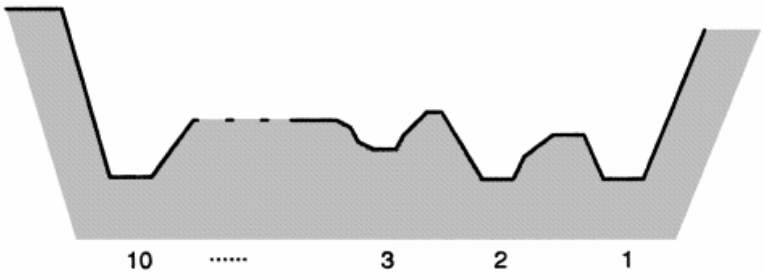


Fig. 2-39

But, external contour along Z must mononously rise or fall, and the following contour cannot be machined:

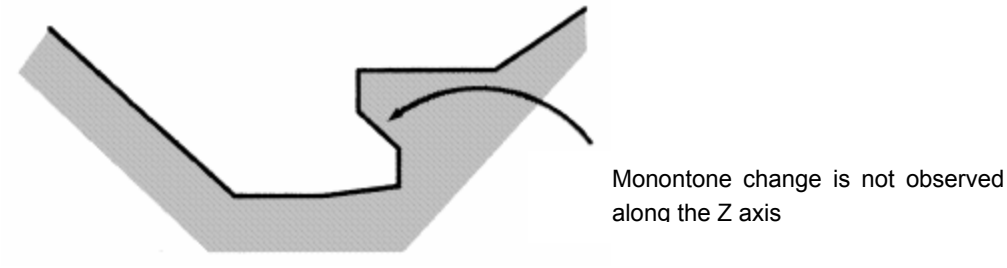


Fig. 2-40

The first tool must be vertical: the machining can be executed when the shape along Z changes mononously, which is shown below:

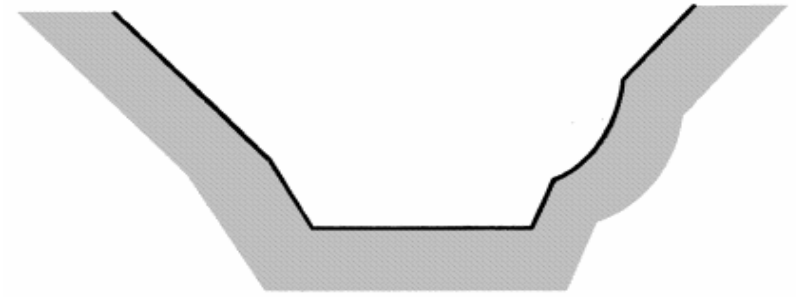


Fig. 2-41

The tool retraction should be executed after turning, and the retraction amount is specified by R (e) or No 5133, which is shown below:

e（set by a parameter）

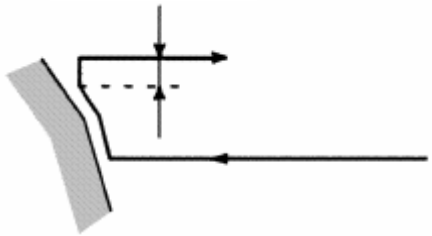


Fig. 2-42

Type 2 code execution process:

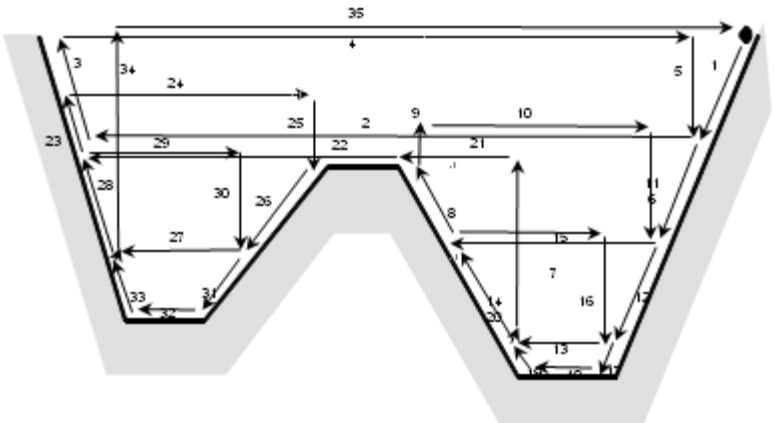


Fig. 2-43

Note 1: In ns block, X (U), Z(W) must be specified, and W0 is done when Z does not move.

Note 2: The finishing allowance is specified to X direction, is invalid for Z direction.

Note 3: When the current grooving is completed, the tool retraction amount is left to make the tool approach the workpiece (Label 25, 26) with G1 speed after the current grooving is done to execute the next grooving. When the retraction amount is 0 or the left distance is less than retraction amount, the tool approaches the workpiece with G1 speed.

Note 4: The finishing path (ns～nf block), Z dimension must mononously change (always increase or decrease)

Note 5: When there is an arc in finishing path (ns～nf), # 3410 parameter (the arc radius permits error) cannot be non-zero, i.e., the permitting function of arc radius error cannot be activated.

Note 6: Radius error is irrelevant to cutting allowance, and radius error is permitted and checks whether the alarm occurs.

2.19.2 Radial Roughing Cycle G72

Command function: G72 is divided into three parts:

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

(2) defining the block interval, finishing allowance;

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

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, and the roughing is completed by multiple executing the cutting cycle tool infeed→cutting feed→tool retraction. The start point and the end point of G72 are the same one. The code is applied to the formed roughing of non-formed rod.

Command format:

```
G72  W(△d)  R(e)  F     S    T          (1)
G72  P(ns)  Q(nf)  U(△u)  W(△w)          (2)
 N  (ns)……;  ⎫
……;         ⎪
……F;        ⎪
……S;        ⎬                          (3)
……;         ⎪
N  (nf) ……;  ⎭
```

Code specifications:

1. ns～nf blocks in programming must be followed G72 blocks. If they are in the front of G72 blocks, and after the system executes roughing cycle, and then executes the next program following G72;
2. ns～nf blocks are used for counting the roughing path and the blocks are not executed when G72 is executed. F, S, T codes of ns～nf blocks are invalid when G72 is executed, at the moment, F, S, T codes of G72 blocks are valid. F, S, T of ns～nf blocks are valid when executing ns～nf to code G70 finishing cycle;
3. There are G00,G01 without the word X(U) in ns block, otherwise the system alarms;
4. X,Z dimensions in finishing path(ns～nf blocks) must be changed monotonously (always

increasing or reducing) for the finishing path;

5. In ns～nf blocks, there are only G codes: G01, G02, G03, G04, G96, G97, G98, G99, G40, G41,G42 and the system cannot call subprograms(M98/M99);
6. G96, G97, G98, G99, G40, G41, G42 are invalid in G72 and valid in G70;
7. 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;
8. When the system is executing the feed hold or single block, the program pauses after the system has executed end point of current path;
9. △d，△u are specified by the same U and different with or without being specified P,Q codes;
10. G72 cannot be executed in MDI, otherwise, the system alarms.

Relevant definitions:

Finishing path	the above-mentioned Part③ of G71(ns～nf block)defines the finishing path, and the start point of finishing path (i.e. start point of ns block)is the same these of start 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 of Z tool infeed in roughing without sign symbols, and the direction of tool infeed is defined by move direction of ns block. Δd is reserved after the system executes W（Δd）and NO.5132 value is modified. The value of system parameter NO.05132 is regarded as the travel of tool infeed when W（Δd）is not input
e	It is each travel of Z tool infeed in roughing without sign symbols, and the direction of tool retraction is opposite to that of tool infeed; after R(e) is executed, e value e is reserved and the system modifies No.5133 value. The value of system parameter NO.5133 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	X finishing allowance in roughing, (X coordinate offset of roughing path compared to finishing path, i.e. the different value of X absolute coordinate between A'and A, diameter value with sign symbols)
Δw	Z finishing allowance in roughing, its value: -9999.999~9999.999 (Z coordinate offset of roughing path compared to finishing path, i.e. the different value of X absolute coordinates between A' and A, with sign symbols)

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 only G70 finishing blocks

Address	Incremental system	Metric（mm）input	Inch (inch) input
W（Δd）	ISB system	0.001~99999.999	0.0001~9999.9999
	ISC system	0.0001~9999.9999	0.00001~999.99999
R（e）	ISB system	0～99999.999	0～9999.9999
	ISC system	0～9999.9999	0～999.99999
U（Δu）	ISB system	-99999.999～99999.999	-9999.9999～9999.9999
	ISC system	-9999.9999～9999.9999	-999.99999～999.99999
W（Δw）	ISB system	-99999.999～99999.999	-9999.9999～9999.9999
	ISC system	-9999.9999～9999.9999	-999.99999～999.99999
P（ns）	ISB system	1～99999	1～99999
	ISC system	1～99999	1～99999
Q（nf）	ISB system	1～99999	1～99999
	ISC system	1～99999	1～99999

Execution process: Fig. 2-44.

① X rapidly traverses to A' from A point, X travel is Δu, and Z travel is Δw;

② X moves from m A'is Δd(tool infeed), ns block is for tool infeed at rapid traverse speed with G0, is for tool infeed at G72 feedrate F in G1, and its direction of tool infeed is that of A→B point;

③ X executes the cutting feeds to the roughing path, and its direction is the same that of X coordinate B→C point;

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

⑤ X rapidly retracts at rapid traverse speed to the position which is the same that of Z coordinate;

⑥ After Z tool infeed (Δd+e)again is executed, 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 Z executes the tool infeed (Δd+e)again, ③ is executed；after the tool infeed (Δd+e) is executed again, the end point of tool traversing reaches B' point or exceeds the straight line between A'→B' point and Z executes the tool infeed to B' point, and then the next step is executed;

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.

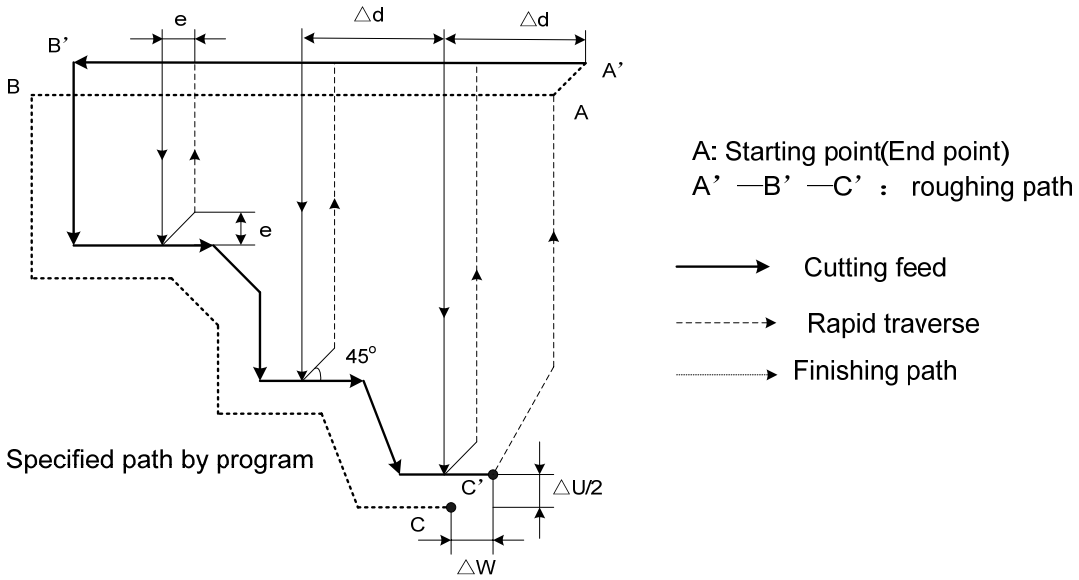


Fig. 2-44

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. 2-45: B→C for finishing path, B'→C' for roughing path and A is the start-up tool point.

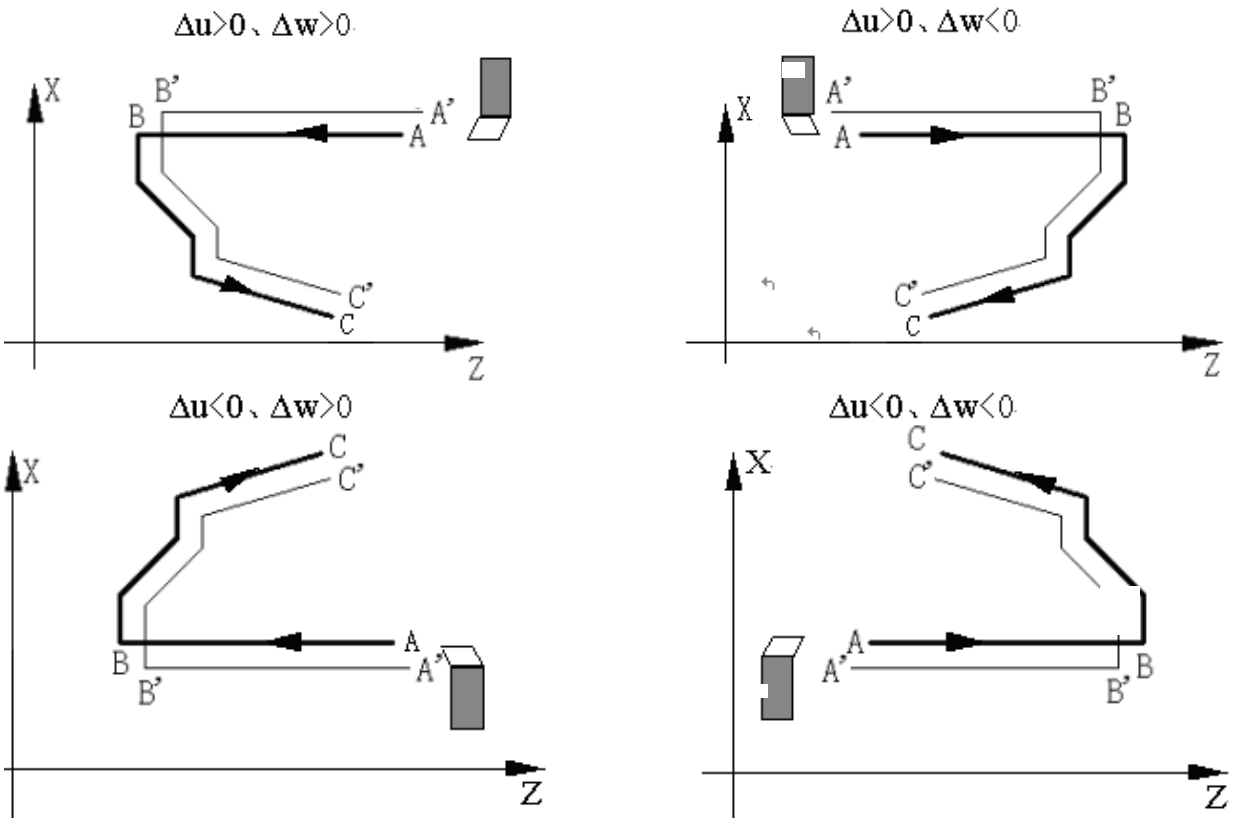


Fig.2-45

Example: Fig.2-46

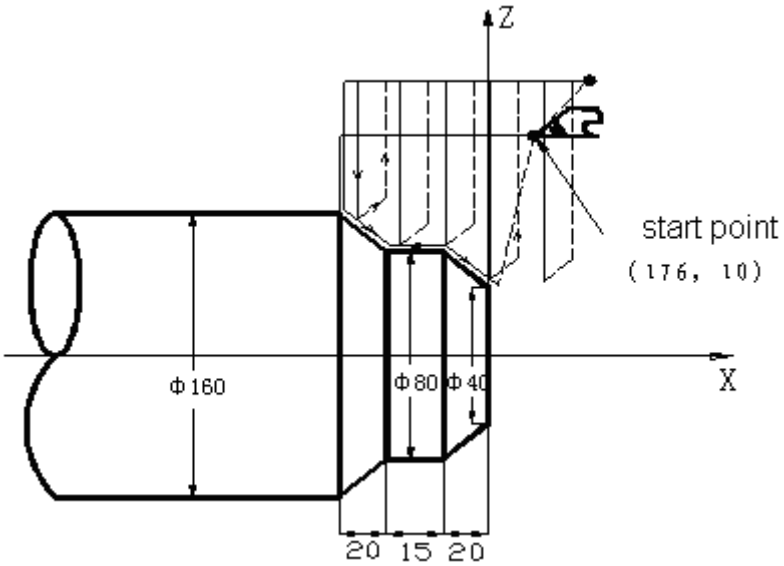


Fig.2-46

```
Program: O0005;
G00 X176 Z10 M3 S500 T0202; (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 0.5mm)
G72 P10 Q20 U0.2 W0.1;      (Roughing a--d，X roughing allowance 0.2mm and Z
                             0.1mm)
N10 G00 Z-55 S800;   (Rapid traverse)
G01 X160 F120;       (Infeed to a point)
X80 W20;             (Machining a—b)      } Blocks for finishing path
W15;                 (Machining b→c)
N20 X40 W20;         (Machining c→d)
G70 P10 Q20;         (Finishing a→d)
M30；
```

2.19.3 Closed Cutting Cycle G73

Command 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 start 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.2-47.

Command format: G73 U(△i) W(△k) R（d） F S T (1)

G73 P(ns) Q(nf) U(△u) W(△w) (2)

N (ns)……;
……;
……F;
……S;
……;
N (nf) ……; (3)

Code specifications:

1. ns～nf blocks in programming must be followed G73 blocks. If they are in the front of G72 blocks, and after the system executes roughing cycle, and then executes the next program following G73;
2. ns～nf blocks are used for counting the roughing path and the blocks are not executed when G73 is executed. F, S, T codes of ns～nf blocks are invalid when G71 is executed, at the moment, F, S, T codes of G73 blocks are valid. F, S, T of ns～nf blocks are valid when executing ns～nf to code G70 finishing cycle.
3. There are only G00, G01 in ns block.
4. In ns～nf blocks, there are only G codes:G00, G01, G02, G03, G04, G96, G97, G98, G99, G40, G41,G42 and the system cannot call subprograms(M98/M99);
5. G96, G97, G98, G99, G40, G41, G42 are invalid in G73 and valid in G70;
6. 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;
7. When the system is executing the feed hold or single block, the program pauses after the system has executed end point of current path;
8. △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 codes;
9. G73 cannot be executed in MDI, otherwise, the system alarms;
10. Z must be the monotonous in the cycle body specified by P and Q. Z tool retraction and finishing allowance are set to 0 when the system executes X non-monotonous workpiece. When No. 5102 Bit0 (MRI) is set to 1, the system does not alarm;
11. When the programming is executed, and the initial positioning point retreats one tool infeed value in the direction of cutting but the result is in the contour range, the dry run is executed to observe whether its own path of the system has overcutting because the tool retraction direction is the same that of tool infeed in programming state.

Relevant definitions:

Finishing path	The above-mentioned Part 3 of G73(ns～nf block)defines the finishing path, and the start point of finishing path (start point of ns block)is the same these of start 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（Δi×2/d-1，Δk/d-1）
Δi	Travel of X tool retraction in roughing is the following table（radius value with sign symbols）, Δi is equal to X coordinate offset value（radius value）of A_1 point compared to A_d point. The X total cutting travel(radius value) is equal to |Δi| in roughing, and X cutting direction is opposite to the sign symbol of Δi: Δi＞0, cut in X negative direction in roughing. It is reserved after Δi code value is executed and the system rewrites No.5135 value. NO.5135 value is regarded as the travel of X tool retraction of roughing when U（Δi）is not input
Δk	Travel of Z tool retraction in roughing is the following table（radius value with sign symbols）, Δk is equal to X coordinate offset value（radius value）of A1point compared to A_d point. The Z total cutting travel(radius value) is equal to |Δk| in roughing, and Z cutting direction is opposite to the sign symbol of Δk: Δk＞0, cut in Z negative direction in roughing. It is reserved after Δk code value is executed and the system rewrites No.5136 value. NO.5136 value is regarded as the travel of X tool retraction of roughing when W（Δk）is not input
d	It is the cutting times and its range is referred to the following table. R5 means the closed cutting cycle is completed by 5 times cutting. R（d）is reserved after it is executed and the system rewrites NO.5137. The value of system parameter NO.5137 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 X finishing allowance as the following table (diameter value with sign symbols) and is the X coordinate offset of roughing contour compared to finishing path, i.e. the different value of X absolute coordinates of A_1 compared to A. Δu＞0，it is the offset of the last X positive roughing path compared to finishing path. The system defaults Δu=0 when U（Δu）is not input, i.e. there is no X finishing allowance for roughing cycle
Δw	It is Z finishing allowance as the following table -99.999~99.999 (unit: mm) and is the Z coordinate offset of roughing contour compared to finishing path, i.e. the different value of Z absolute coordinate of A_1 compared to A. 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 W（Δw）is not input, i.e. there is no Z finishing allowance for roughing cycle
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

Address	Incremental system	Metric（mm）input	Inch (inch) input
U（Δi）	ISB system	-99999.999～99999.999	-9999.9999～9999.9999
	ISC system	-9999.9999～9999.9999	-999.99999～999.99999
W（Δk）	ISB system	-99999.999～99999.999	-9999.9999～9999.9999
	ISC system	-9999.9999～9999.9999	-999.99999～999.99999
R（d）	ISB, ISC	1 ～ 999 （ times ）（ ignore decimal part）	1～999（times）（ignore decimal part）
U (Δu）	ISB system	-99999.999～99999.999	-9999.9999～9999.9999
	ISC system	-9999.9999～9999.9999	-999.99999～999.99999
W（Δw）	ISB system	-99999.999～99999.999	-9999.9999～9999.9999
	ISC system	-9999.9999～9999.9999	-999.99999～999.99999
P（ns）	ISB system	1～99999	1～99999
	ISC system	1～99999	1～99999
Q（nf）	ISB system	1～99999	1～99999
	ISC system	1～99999	1～99999

Execution process:(Fig. 2-47)

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 feed.

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

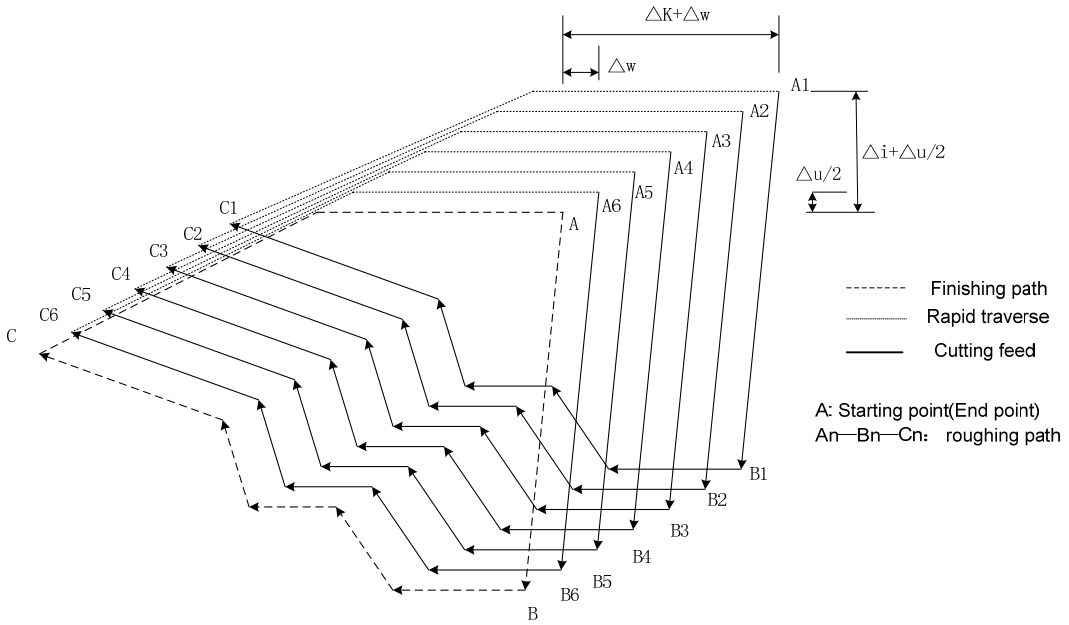
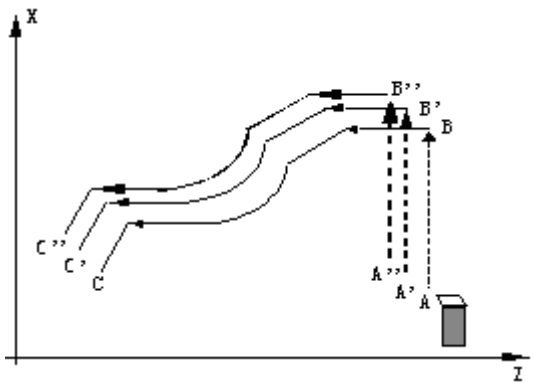


Fig. 2-47 G73 run path

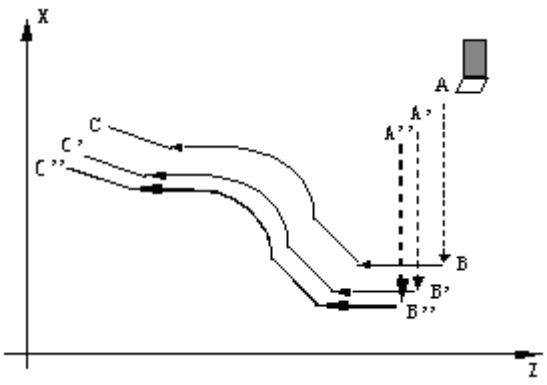
Coordinate offset direction with finishing allowance:

Δi, Δk define the coordinates offset and its direction of roughing, Δu, Δw define the coordinates offset and cut-in direction in finishing; Δi, Δk, Δu, Δw can consist of many groups. Generally, the sign symbols of Δi and Δu are consistent, the sign symbols of Δk and Δw are consistent, there are four kinds of combination as Fig. 2-48,A for start-up tool point, B→C for workpiece contour, B'→C'for roughing contour and B"→C" for finishing path.

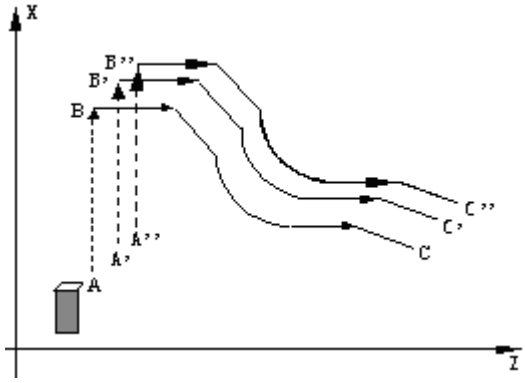
1) Δi<0 Δk>0, △u<0 Δw>0;



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；

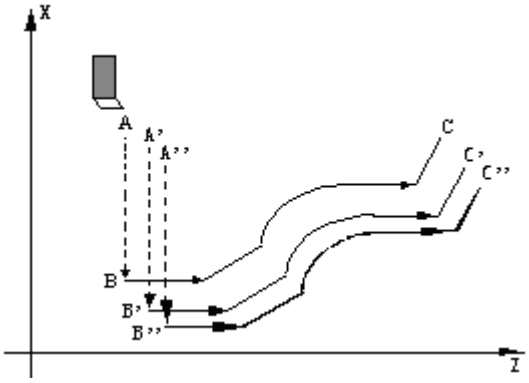


Fig.2-48

Example: Fig. 2-49

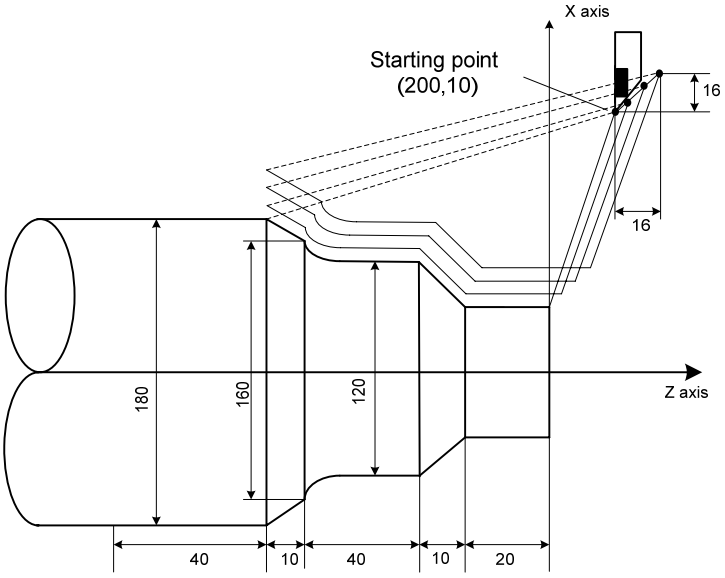


Fig.2-49

Program：O0006；

G99 G00 X200 Z10 M03 S500;	（Specify feedrate per rev and position start point and start spindle）
G73 U1.0 W1.0 R3 ;	（X tool retraction with 15mm, Z 15mm）
G73 P14 Q19 U0.5 W0.3 F0.3 ;	（X roughing with 2 allowance and Z 1mm）
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）

2.19.4 Finishing Cycle G70

Command function: The tool executes the finishing of workpiece from start 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 start point and execute the next block following G70 block after G70 cycle is completed.

Command format: G70 P（ns） Q（nf）;

Code specifications:

1. ns：Block number of the first block of finishing path, range: 1～99999;
   nf：Block number of the last block of finishing path, range: 1～99999;
   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;                    Blocks for finishing path
……S;
N   (nf) ……;
G70   P（ns）   Q（nf）;
……
```

2. G70 is compiled following ns～nf blocks;
3. F, S, T in ns～nf blocks are valid when executing ns～nf to code G70 finishing cycle;
4. G96, G97, G98, G99, G40, G41, G42 are valid in G70;
5. 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;
6. When the system is executing the single block, the program pauses after the system has executed end point of current path;
7. G70 cannot be executed in MDI mode, otherwise, an alarm occurs.

Note: When the tool cuts to the end point of finishing shape in the finishing cycle, the two axes simultaneously returns to the cycle start point, so the user should pay attention to avoid overcut.

2.19.5 Axial Grooving Multiple Cycle G74

Command function: Axial (X) tool infeed cycle compounds radial discontinuous cutting cycle: Tool infeeds from start point in radial direction(Z), retracts, infeeds again, and again and again, and last tool retracts in axial direction, and retracts to the Z position 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 start point (start 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 start point of cutting. The code is used to machine radial loop groove or column surface by radial discontinuously cutting, breaking stock and stock removal.

Command format: G74 R（e）;

G74 X（U） Z（W） P（Δi） Q（Δk） R（Δd） F ;

Code specifications:

1. 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;
2. 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;
3. 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 again, otherwise the following path will be wrong.
4. When the single block is running, programs pauses after each axial cutting cycle is completed.
5. 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.

Relevant definitions:

Start point of axial cutting cycle	Starting position of axial tool infeed for each axial cutting cycle, defining with A_n(n=1,2,3……), Z coordinate of A_n is the same that of start point A, the different value of X coordinate between A_n and A_{n-1} is Δi. The start point A_1 of the first axial cutting cycle is the same as the start point A, and the X coordinate of start point (A_f) of the last axial cutting cycle 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……), Z coordinate of B_n is the same that of cutting end point, X coordinate of B_n 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……), Z coordinate of C_n is the same that of cutting end point, and the different value of X coordinate between C_n and A_n 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……), Z coordinate of D_n is the same that of start point, X coordinate of D_n is the same that of C_n (the different value of X coordinate between it and A_n is Δd)

Cutting end point	It is defined by X（U）__ Z（W）___ ,and is the end point B_f of last axial tool infeed
R（e）	It is the travel of tool retraction after each axial (Z) tool infeed without sign symbols as the following table. The code value is reserved after executing R（e）and the value of NO.5139 is rewritten. The value of NO.5139 is regarded as the travel of tool retraction when R（e）is not input
X	absolute coordinate value of the 2nd axis (X in ZX plane) in path diagram B point
U	Relative movement amount of the 2nd (U in ZX plane) in path diagram A→B plane.(for G code system A. X_,Z_ is executed for other conditions)
Z	absolute coordinate value of the 1st axis (Z in ZX plane) in path diagram C point
W	Relative movement amount of the 2nd (W in ZX plane) in path diagram A→C plane. (for G code system A. X_,Z_ is executed for other conditions)
P（Δi）	Travel of radial(X) cutting for each axial cutting cycle without sign symbols, and the value range is referred to the following table
Q（Δk）	Travel of Z discontinuous tool infeed without sign symbols in axial(Z) cutting, and the value range is referred to the following table
R（Δd）	Travel (radius value) of radial (X) tool retraction after cutting to end point of axial cutting. The value range is referred to the following table. The radial (X) tool retraction is 0 when R（Δd）is omitted and the system defaults the axial cutting end point. The radial (X) tool retraction is 0 when P（Δi） is omitted

Address	Incremental system	metric（mm） input	Inch (inch) input
P(Δi) Q(Δk)	ISB system	1~99999999(unit ： 0.001mm)	1~99999999(unit ： 0.0001inch)
	ISC system	1~99999999(unit ： 0.0001mm)	1~99999999(unit ：0.00001inch)
R(e) R(Δd)	ISB system	0～99999.999mm	0～9999.9999 inch
	ISC system	0～9999.9999 mm	0～999.99999 inch

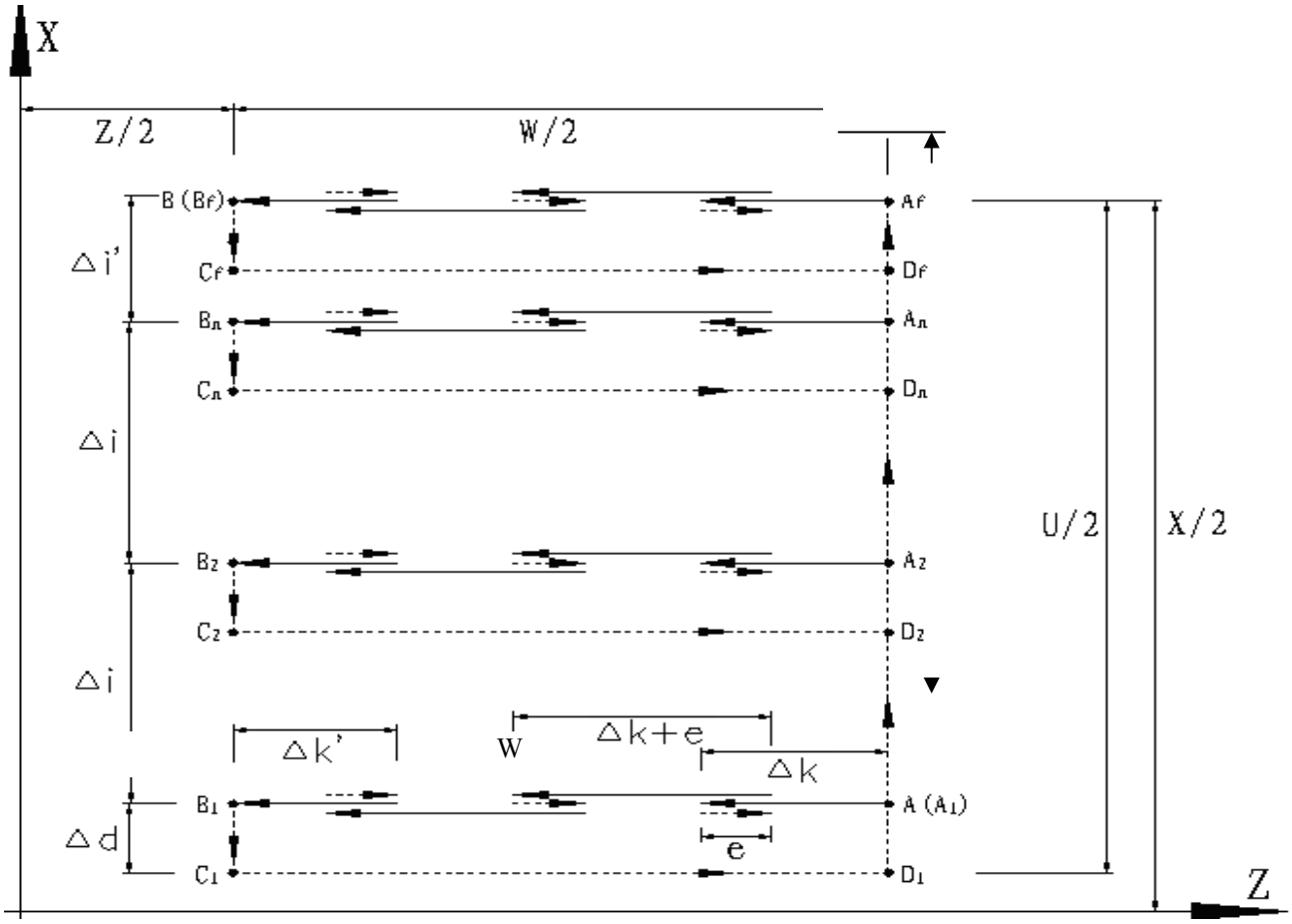


Fig.2-50 G74 path

Code execution process: Fig. 2-50.

① Z executes the axial cutting feed △k from the start point of axial cutting cycle An. When Z coordinate of cutting end point is less than that of the start point, Z negatively feeds, otherwise, Z positively feed;

② Z executes the axial tool retraction e rapidly, and its tool retraction direction is opposite to the feed direction of Step ① ;

③ When Z executes feed cutting (Δk+e) again, and the feed end point is still between the axial cutting cycle starting point An and the axial tool infeed end point Bn. It executes the cutting feed (Δk+e) again, and executes the Step ②; after Z executes the cutting feed (Δk+e)，its feed end point reaches Bn or is not between An and Bn，Z executes the cutting feed to Bn，and execute the Step ④；

④ X executes the radial tool retraction △d（radius value）rapidly to Cn. When X coordinate of cutting end point Bf is less than that of the start point A, X positively executes tool retraction, otherwise, Z executes negatively tool retraction;

⑤ Z executes the axial tool retraction to Dn rapidly, and the n times cutting cycle ends. When the current is not the last axial cutting cycle, the system executes the Step ⑥; when the current is the last axial cutting cycle, the system executes the Step ⑦;

⑥ X executes the rapid tool infeed, and its infeed direction is opposite to the Step ④. After X executes the tool infeed (△d+△i)（radius value），the tool infeed end point is still between A and Af (it is the start point of the last axial cutting cycle), X executes the rapid tool infeed

(△d+△i)（radius value），i.e.，：Dn→An+1，then executes the Step ①（start the next axial cutting cycle）; after X executes (△d+△i) （radius value），the tool infeed end point reaches Af or is not between Dn and Af，X performs the rapid traverse to Af，then, executes the Step ①，and starts the last axial cutting cycle；

⑦ X rapidly returns to the start point A, and G74 execution processing ends.

Example: Fig. 2-51

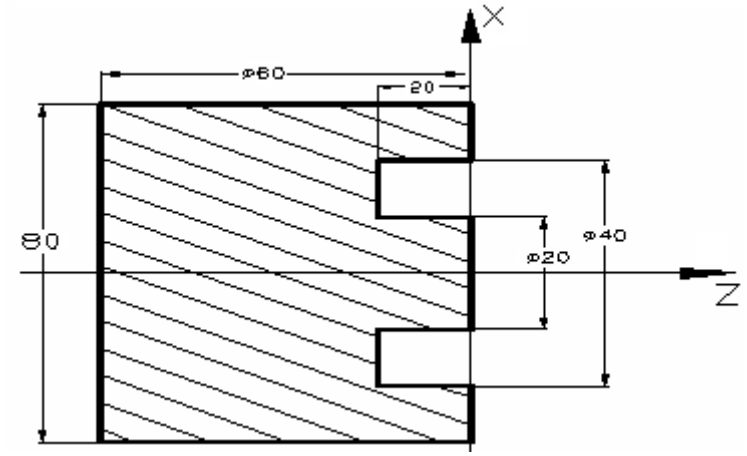


Fig.2-51

Program:

O0007；

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

G74 R0.5 ；　　（Machining cycle）

G74 X20 Z-20 P3000 Q5000 F50；　(Z tool infeed 5mm and tool retraction 0.5mm each time; rapid return to start point(Z5) after cutting feed to end point(Z-20), X tool infeed 3mm and cycle the above-mentioned steps)

M30；　　（End of program）

2.19.6 Radial Grooving Multiple Cycle G75

Command function: Axial (Z) tool infeed cycle compounds radial discontinuous cutting cycle: Tool infeeds from start point in radial direction, retracts, infeeds again, and again and again, and last tool retracts in axial direction, and retracts to position 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 start point (start 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 start point of cutting. G75 is used to machine the radial loop groove or column surface by radial discontinuously cutting, breaking stock and stock removal.

Command format: G75 R（e）;

G75 X（U） Z（W） P（Δi） Q（Δk） R（Δd） F ；

Command explanations:

1. The cycle movement is executed by X（W）and P（Δi）blocks of G75, G75 is not executed when there is no X(U) in G75 block. When only “G75 R（e）；” block is executed and only No.5139

value is modified, the cycle operation cannot be executed;

2. Δd and e are specified by the same address R and whether there are X（U）or not in blocks can distinguish them;
3. 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;
4. When the system is executing the single block, the program pauses after the system has executed end point of current path;
5. 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.

Relevant definitions:

Start point of radial cutting cycle	Starting position of axial tool infeed for each radial cutting cycle, defined by A_n(n=1,2,3……),X coordinate of A_n is the same that of start point A, the different value of X coordinate between A_n and A_{n-1} is Δk. The start point A_1 of the first radial cutting cycle is the same as the start point A and Z start point (A_f) of the last axial cutting cycle 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……), X coordinates of B_n is the same that of cutting end point, Z coordinates of B_n 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……), X coordinate of C_n is the same that of cutting end point, and the different value of Z coordinate between C_n and A_n 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……), X coordinate of D_n is the same that of start point, Z coordinates of D_n is the same that of C_n (the different value of Z coordinate 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 of tool retraction after each radial(X) tool infeed without sign symbols and its value range is referred to the following table. The code value is reserved and the value of system parameter NO.5139 is rewritten after R（e）is executed. The value of NO.5139 is regarded as the travel of tool retraction when R（e）is not input
X	absolute coordinate value of the 2nd axis (X in ZX plane) in path diagram B point
U	Relative movement amount of the 2nd (U in ZX plane) in path diagram A→B plane. (for G code system A. X_,Z_ is executed for other conditions)
Z	absolute coordinate value of the 1st axis (Z in ZX plane) in path diagram C point
W	Relative movement amount of the 2nd (W in ZX plane) in path diagram A→C plane. (for G code system A. X_,Z_ is executed for other conditions)
P（Δi）	Travel (radius value) of X discontinuous tool infeed without sign symbols in radial

	(X) tool infeed, and the value range is referred to the following table
Q（Δk）	Travel of radial(Z) cutting for each axial cutting cycle without sign symbols, and the value range is referred to the following table
R（Δd）	Travel of axial (Z) tool retraction after cutting to radial end point. The value range is referred to the following table. The system defaults travel of the axial (Z) tool retraction is 0 when R（Δd）and Q（Δk）are omitted The system defaults the negative tool retraction is done when Z(W) is omitted. The radial (X) tool retraction is 0 when P（Δi） is omitted

Address	Incremental system	Metric（mm）input	Inch (inch) input
P（Δi） Q（Δk）	ISB system	1~99999999（0.001mm）	1~99999999（0.0001inch）
	ISC system	1~99999999(0.0001mm）	1~99999999（0.00001inch）
R（e） R（Δd）	ISB system	0～99999.999mm	0～9999.9999 inch
	ISC system	0～9999.9999 mm	0～999.99999 inch

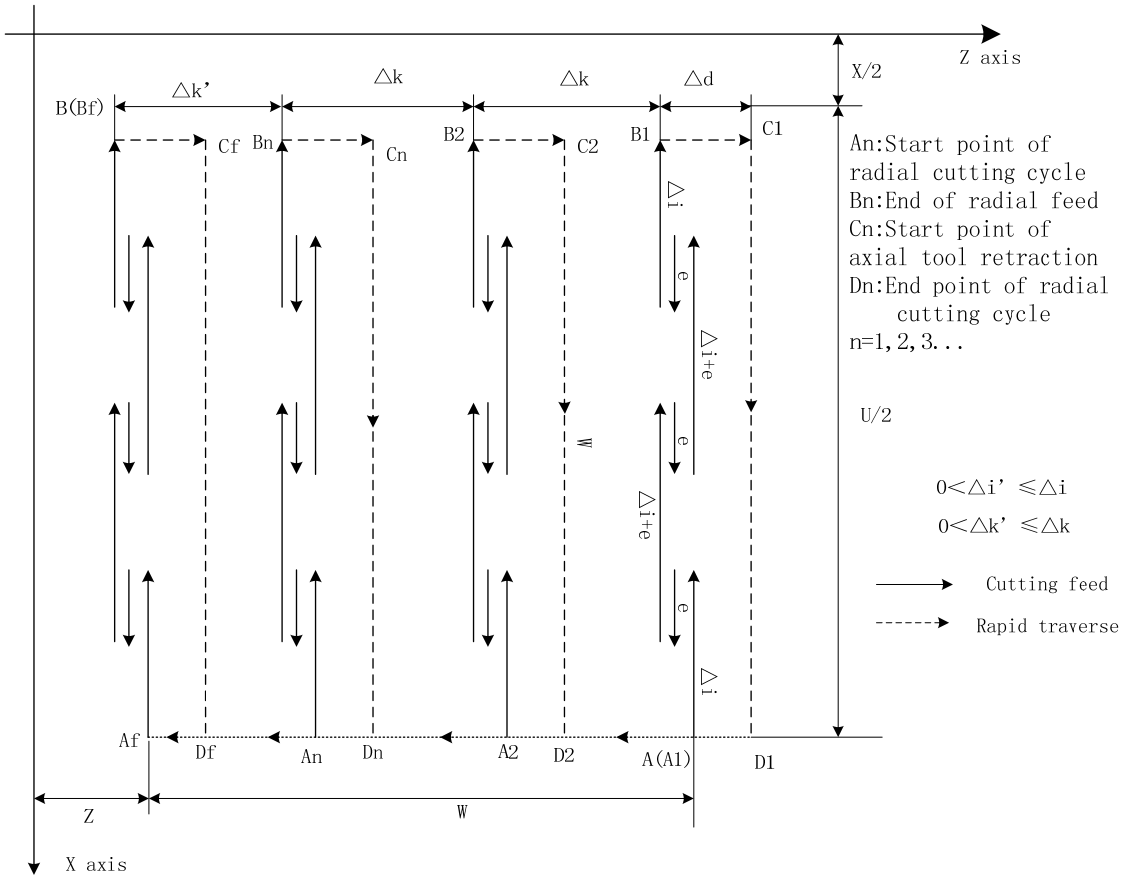


Fig. 2-52 G75 path

Execution process: Fig. 2-52

①X executes the radial cutting feed △k from the start point of radial cutting cycle An. When X coordinate of cutting end point is less than that of the start point, X negatively feeds, otherwise, X positively feed;

② X executes the radial tool retraction e rapidly, and its tool retraction direction is opposite to the feed direction of Step ① ;

③When X executes feed cutting (Δi+e) again, and the feed end point is still between the radial cutting cycle starting point An and the axial tool infeed end point Bn. It executes the cutting feed (Δi+e) again, and executes the Step ②; after X executes the cutting feed (Δi+e)，its feed end point reaches Bn or is not between An and Bn，X executes the cutting feed to Bn，and execute the Step ④;

④Z executes the axial tool retraction △d（radius value）rapidly to Cn. When Z coordinate of cutting end point Bf is less than that of the start point A, Z positively executes tool retraction, otherwise, Z executes negatively tool retraction;

⑤X executes the radial tool retraction to Dn rapidly, and the n times cutting cycle ends. When the current is not the last radial cutting cycle, the system executes the Step ⑥; when the current is the last radial cutting cycle, the system executes the Step ⑦;

⑥Z executes the axial tool infeed rapidly, and its infeed direction is opposite to the Step ④. After Z executes the tool infeed △d+△k)（radius value)，the tool infeed end point is still between A and Af (it is the start point of the last radial cutting cycle), Z executes the rapid tool infeed (△d+△k)，i.e.，：Dn→An+1，then executes the Step ①（start the next radial cutting cycle）; after Z executes (△d+△k)，the tool infeed end point reaches Af or is not between Dn and Af，Z performs the rapid traverse to Af，then, executes the Step ①，and starts the last radial cutting cycle；

⑦ X rapidly returns to the start point A, and G75 execution processing ends.。

Example：Fig. 2-53

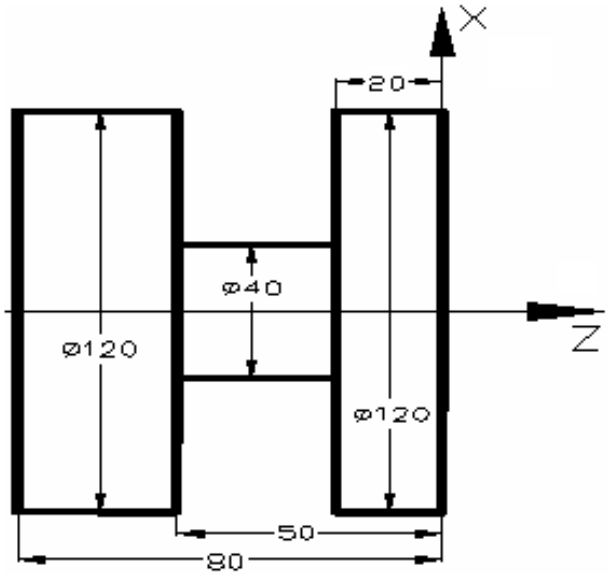


Fig. 2-53 G75 cutting path

Program:

O0008；

G00 X150 Z50 M3 S500； （Start spindle with 500 rev/min）

G0 X125 Z-20； （Position to start point of machining）

G75 R0.5 F150； （Machining cycle）

G75 X40 Z-50 P6000 Q3000；（X tool infeed 6mm every time, tool retraction 0.5mm, rapid returning to start point (X125) after infeeding to end point

(X40), Z tool infeed 3mm and cycle the above-mentioned steps to continuously run programs）

G0 X150 Z50； （Return to start point of machining）

M30； （End of program）

2.19.7 Notes for Multi Cycle Machining

Note 1. When the multi cycle blocks are executed, they should be the specified address P, Q, X, Z, U, W, R of each block;

Note 2. The block specified by P in G71，G72, G73 should be G00 or G01. When there is no code, the system alarms;

Note 3. In MDI and DNC mode, G70，G71，G72 or G73 can not be specified, otherwise, an alarm occurs. But in MDI and DNC mode, G74，G75 or G76 can be specified;

Note 4: The block quantity of G70, G71, G72 or G73 in the sequence numbers specified by P and Q cannot exceed 100;

Note 5: The blocks in the serial numbers specified by P and Q in G71，G72 or G73 cannot specify the following code:

（1）non-modal G code except for G04 in group 00;

（2）all G codes except for G00, G01, G02, G03 in group 01;

（3）G20 and G21;

（4）M98 and M99;

Note 6: The skip function should not be executed in the blocks of their serial number specified by P and Q. when the skip function is used in the blocks of their serial numbers specified by P and Q.

Note 7: The tool nose radius compensation is invalid in G71～G76.

Note 8: No.5104 Bit2 (FCK）sets whether G71, G72 or G73 executes the outer check. When it is set to1, the check is executed. The system alarms when the positioning point is in the cutting range.

Note 9: No.5102 Bit1（MRC）set whether the system alarm when the finishing cycle in G71，G72 is in non-monotonous, and an alarm occurs when Bit1 is set to 1.

2.20 Threading Cutting

GSK988TA/AB 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, Z thread cutting G33, Thread cutting cycle G92, Multiple thread cutting cycle G76.

The machine used to thread cutting must be installed with spindle encoder, the transmission ratio between spindle and encoder is set by the parameter. There are two kind of communication connection method. The encoder data is transferred to the CNC by the servo spindle in bus communication mode or the spindle encoder is connected with the CNC by the encoder wires. X or Z 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.

GSK988TA/TB CNC 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 X and Z thread cutting, and so there is length of thread lead-in and distance of tool retraction at the actual starting and ending of thread cutting.

X or Z traverse speed 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 X or Z acceleration/deceleration, 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.

Note: When the modal function is used, M30 (M30 modal in MDI mode cannot be cancelled) in Auto mode or G codes in Group 01 can cancel the mode state.

2.20.1 Thread Cutting with Constant Lead G32

Command function: Executing G32 can machine the metric or inch straight, taper, end face thread and continuous multi-section thread:

Command format: G32 IP_ F_ J_ K_ Q_

Code specifications: G32 is modal;

IP_	End point coordinate value. It can be specified by the absolute code value or incremental code value. The system specifies the different IP_ value to execute the straight thread cutting, end face thread cutting and taper thread cutting.
F	Metric pitch is moving distance of long axis when the spindle rotates one-turn and its value range is referred to the following table. After F is executed, it is valid until F with specified pitch is executed again. The pitch F value precision is the last two-digit of the decimal. The value is specified by radius value.
J	Travel in the short axis in thread run-out with positive/negative sign symbols and the value range is referred to the following table; the value is specified by the radius value.
K	Length in the long axis in thread run-out. The value range is referred to the following table without direction; the value is specified by radius value.
Q	Initial angle between spindle rotation one-turn and start point of thread cutting. The value range without the decimal is referred to the following table. Q is non-modal parameter, must be defined every time, it is 0^0 .when it is not specified, the system specifies Q different value can cut multi-thread.

Q rules:

1. Its initial angle is 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. In ISB mode, Q unit is 0.001°. ISC mode, Q unit is 0.0001°. Example, in ISB mode, Q180000 is input in program if it offsets 180° with spindle one rev; if Q180 or Q180.0, it is 0.18°. When the system specifies the value is more than 360000, it counts based on 360000(360°);
4. It is suggested that the system should use G97 instead of the constant surface cutting speed control in thread cutting.

Address	Incremental system	Metric（mm）input	Inch (inch) input
F	ISB	0.01 mm～500 mm	0.0001 inch～9.99inch

	ISC	0.01 mm～500 mm	0.0001 inch～9.99inch
J	ISB	-99999.999 mm～99999.999mm	-9999.9999 inch～9999.9999 inch
	ISC	-9999.9999 mm～9999.9999 mm	-999.99999 inch～999.99999 inch
K	ISB	0～99999.999mm	0～9999.9999 inch
	ISC	0～9999.9999mm	0～999.99999 inch
Q	ISB	0~99999999 (unit：0.001°)	0~99999999（unit：0.001°）
	ISC	0~99999999 (unit：0.0001°)	0~99999999（unit：0.0001°）

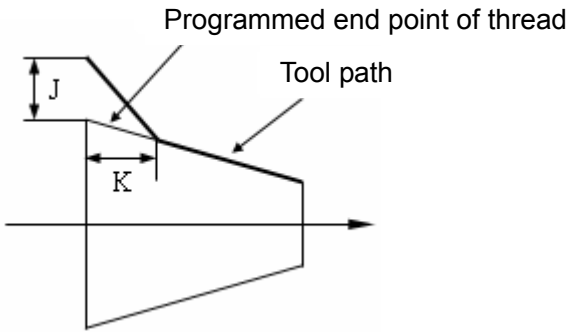


Fig. 2-54 thread run-out

Command path:

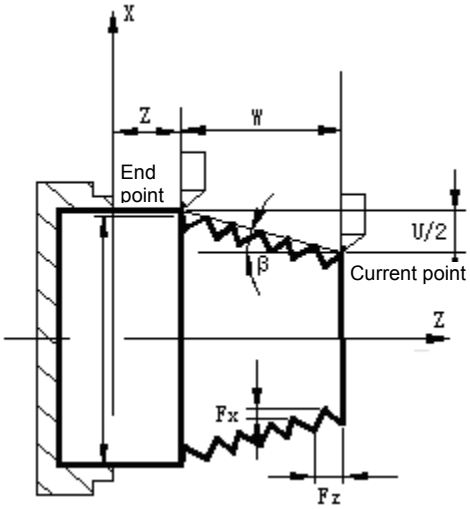


Fig.2-55 G32 path

Difference between long axis and short axis:

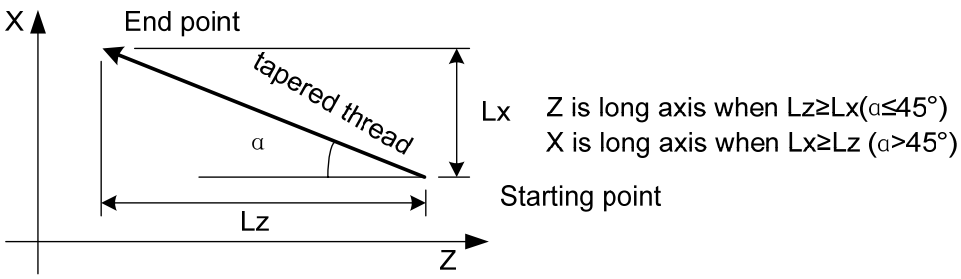


Fig.2-56 long axis, short axis

Note 1: When the thread run-out, the short axis executes the thread run-out at the speed of No. 1466 value, and the long does at the current thread cutting speed.

Note 2: J, K are modal. J, K mode is cancelled when the system executes the non thread cutting code; it cannot code J, K value in the 1st block and the middle block when the system continuously executes the thread cutting, but it can specify J0 K0, otherwise, it considers the non continuous thread machining is done. The thread run-out is done when J, K value is executed in the last thread cutting;

Note 3: There is no thread run-out when J, or J, K is omitted;

Note 4: When only K is omitted, the long axis does not execute thread run-out, but the short axis executes thread run-out with J value;

Note 5: There is no thread run-out when J=0;

Note 6:J≠0, the long axis without thread run-out and the short axis with J value thread run-out: in the course of thread run, after the tool cuts to the thread's end point at the thread feedrate, it retracts vertically along the short axis (the thread run-out speed of short axis is performed when No.466 is set to 0) ; to avoid of appearing a groove, No.1466 thread run-out speed should be increased properly, and No.1628's acceleration/deceleration time constant of thread run-out should be decreased properly;

Note 7: If the current block is for thread and the next block is the same, the system does not test the spindle encoder signal per rotation at starting the next block to execute the direct thread cutting, which function can realize continuous thread machining;

Note 8: The feed hold operation is executed during the thread cutting, and the system displays "Run" and the thread cutting does not stop till the current block is executed. When the thread cutting block is executed in continuous thread machining, the program run pauses after the thread cutting blocks are executed completely;

Note 9: In Single block, the program stops run after the current block is executed. The program stops run after all blocks for continuous thread cutting are executed;

Note 10: The thread cutting decelerates to stop when the system resets, emergently stop or its driver alarms;

Note 11: An alarm occurs when the thread run-out length is more than the thread machined length of the long axis.

Note 12: In G32, the basic axis code cannot be in the same block with its parallel axis code, otherwise, the system alarms.

Note 13: When machining the thread in the metric tool machine in the unit of tooth/inch, using the expression calculated value programs F code. For example, when the thread with 10 teeth/inch is machined, using F[25.4/10] programs.

Note 14: The system automatically checks the spindle speed before machining the thread, the system alarms when the spindle speed is not executed. The spindle speed cannot be checked in the course of the machining.

Example: Pitch: 2mm. δ1 = 3mm,δ2 = 2mm, total cutting depth 2mm with two times cut-in.

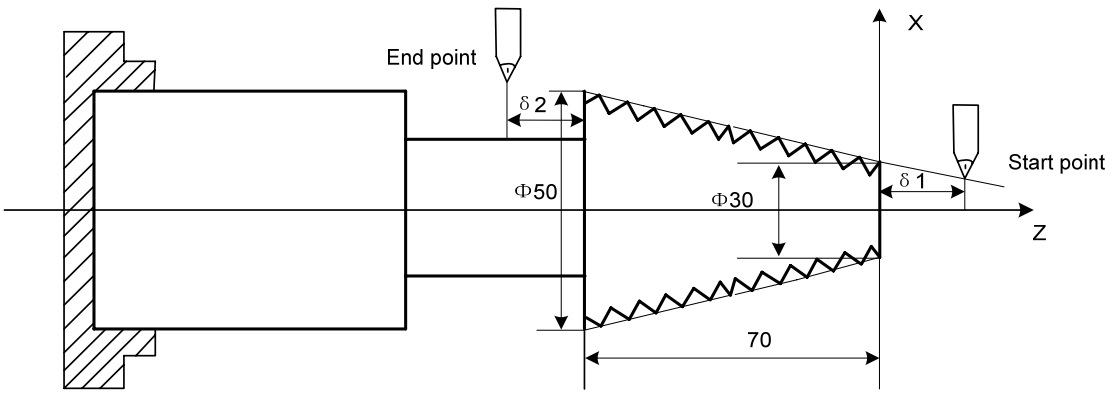


Fig. 2-57

Program:

```
O0009;
G00 X30.14 Z3;            (the first cut-in 1mm)
G32 X51.57 W-75 F2.0;     (the first taper cutting)
G00 X55;                  (First taper cutting)
W75;                      (Z returns to the start point)
X29.14;                   (the second tool infeed 0.5mm)
G32 X50.57 W-75 F2.0;     (the second taper thread cutting)
G00 X55;                  (Tool retraction)
W75 ;                     (Z returns to the start point)
M30;
```

2.20.2 Thread Cutting with Variable Lead G34

Command function: G34 can machine the metric, inch straight, tater thread with variable pitch and end thread.

Command format: G34 IP__ F__ J__ K__ R__ Q__;

Code specifications: G34 is modal;

IP_, J_, K_, Q_	Meaning and value range are the same those of G32
F	It is the first thread pitch from start point, and its range is the same that of G32
R	Incremental value or decremental value of spindle per pitch, R=F2-F1, R is with a direction; F1>F2, the pitch decreases when R is negative; F1<F2, the pitch increases when R is positive; R range: ±0.01 inch/pitch~ ±499.99 mm/pitch(metric thread); ±0.0001 inch/pitch~ ±9.9899 inch/pitch（inch thread）; An alarm occurs when R value exceeds the above range and the pitch exceeds the permissive range because of R increment/decrement or the pitch is negative.

Command path:

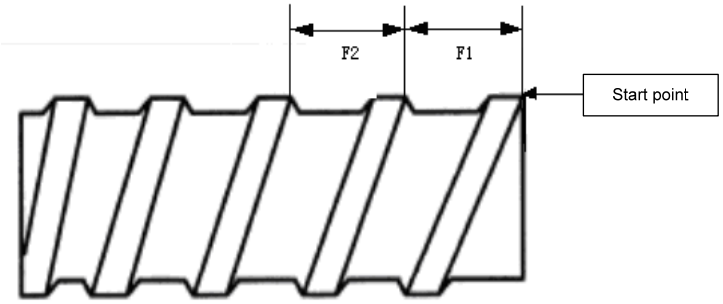


Fig. 2-58

Note: They are the same as those of G32.

Example: First pitch of start point: 4mm, increment 0.2mm per rotation of spindle

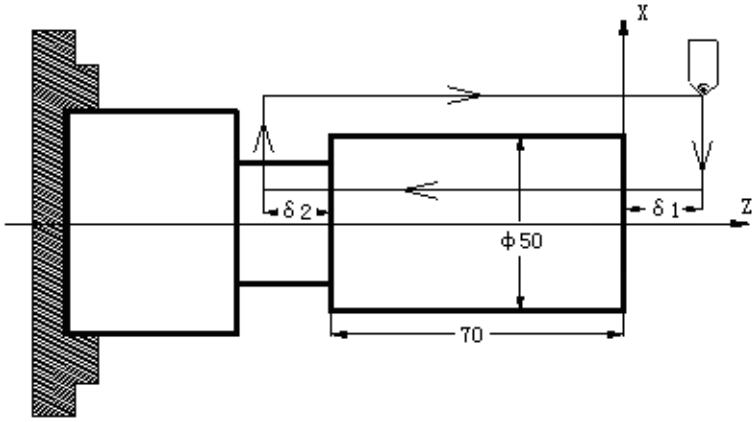


Fig.2-59 Variable pitch thread machining

Value：δ1 = 4mm，δ2 = 4mm，total cutting depth 1mm，total cutting cycle 2 times；the 1st tool infeed 0.7mm.

Program: O0010；

G00 X60 Z4 M03 S500；	
G00 U-10；	Tool infeed Φ50
G00 U-0.7；	Tool infeed
G34 W-78 F4 J5 K2 R0.2；	Variable pitch thread cutting
G00 U10；	Tool retraction
Z4；	Z returns to initial point
G00 X50；	Tool infeed again Φ50
G00 U-1.0；	Tool infeed
G34 W-78 F4 J5 K2 R0.2；	Variable pitch thread cutting
G00 U10；	Tool retraction
Z4；	Z returns to initial point
M30；	

2.20.3 Thread Cutting Cycle G92

Command function: Tool infeeds in radial(X) direction and cuts in axial(Z or X, Z) direction from start 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, Z executes thread interpolation and X retracts with exponential or linear acceleration, and X retracts at rapidly traverse speed after Z reaches to end point of cutting as Fig. 2-60 and Fig.2-61.

Command format: G92 X（U）_ Z（W）_ F_ J_ K_ L Q ；（straight thread cutting cycle）
G92 X（U）_ Z（W）_ R_ F_ J_ K_ L Q ；（taper thread cutting cycle）

Code specifications: G92 is modal;

X	X absolute coordinate of end point of cutting
U	Different value of X absolute coordinate from end point to start point of cutting
Z	Z absolute coordinate of end point of cutting
W	Different value of Z absolute coordinate from end point to start point of cutting
R	Different value(R value) of X absolute coordinate from end point to start point of cutting. When the sign of R is not consistent with U, R|≤|U/2|
F	Metric thread pitch is the same that of G32. After F value is executed, it is reserved and can be omitted
J	Travel in the short axis in thread run-out is same that of G32 and cannot be less than 0 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	Travel in the long axis in thread run-out is same that of G32 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)
Q	Shift angle of initial angle at the beginning of thread cutting. Its range is referred to G32

The system can machine one thread with many tool infeed 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(X in radius) when the spindle rotates one rotation.

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

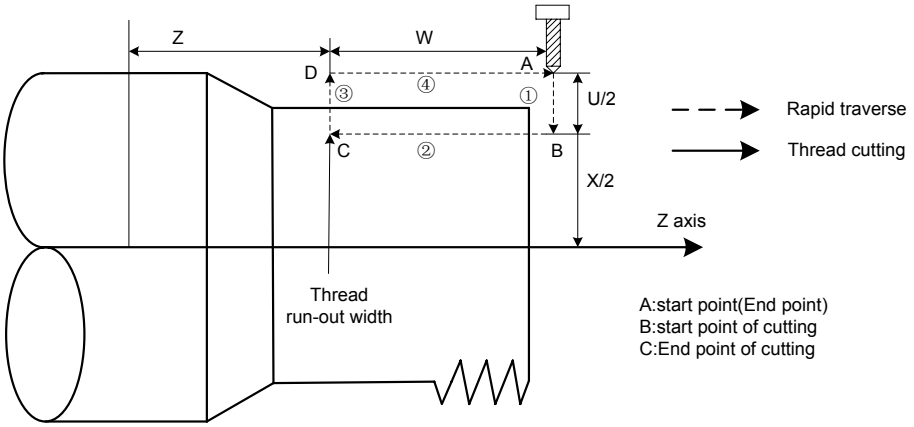


Fig. 2-60 Straight thread

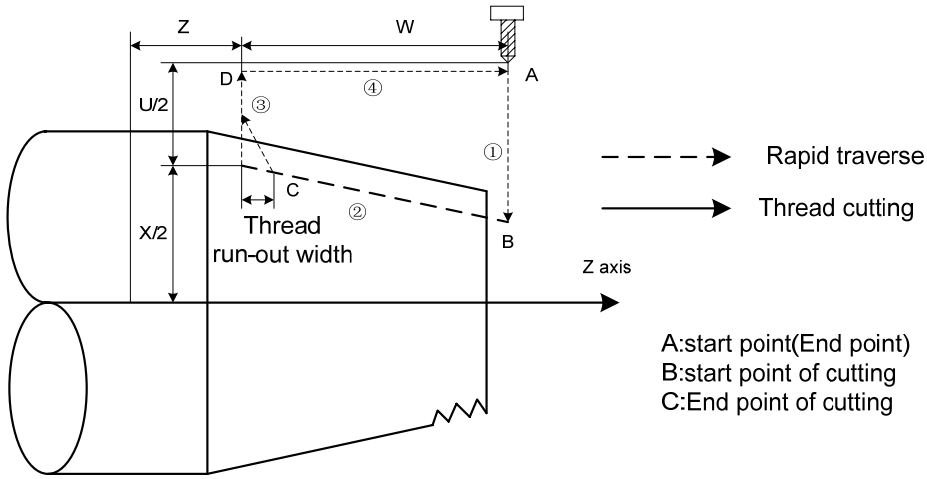


Fig.2-61 Taper thread

Cycle process: straight thread as Fig.2-60 and taper thread as Fig.2-61.

① X traverses from start point to cutting start point;

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

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

④ Z rapidly traverses to return to the start point and the cycle is completed.

Note 1: When J, K is omitted, the thread run-out is confirmed by No. 5130 (chamfering value) and No. 1531 (run-out angle) , the run-out value of the long axis =No. 5130 setting value ×0.1×F, and F is the thread pitch. When No. 5131 (the run-out angle) is set to 0, the long axis and the short axis execute 45°run-out; when the setting value is positive integer, the run-out is done based on the run-out value and angle of the long axis(the system automatically counts the run-out value of the short axis):

Note 2: Length of thread run-out is K in the long direction and is specified by No.5130 when J is omitted;

Note 3: Length of thread run-out is J=K when K is omitted;

Note 4: There is no thread run-out when J=0 or J=0, K=0;

Note 5: Length of thread run-out is J=K when J≠0,K=0;

Note 6: There is no thread run-out when J=0,K≠0;

Note 7: After executing the feed hold in thread cutting, the system still executes the thread cutting. After returning the start point (one thread cutting cycle is completed) , the system displays "Stop", and the program run pauses;

Note 8: After executing single block in thread cutting, the program run stops after the system returns to start point(one thread cutting cycle is completed);

Note 9: Thread cutting decelerates to stop when the system resets, emergently stops or its driver alarms;

Note 10: The system alarms when the thread run-out length of the long axis is more than the thread machining length of the long axis;

Note 11: The system alarms when the thread run-out length of the short axis is more than the thread machining length of the short axis;

Note 12: The system automatically checks the spindle speed, and an alarm occurs when there is no speed signal which is feedback to the system or the speed is too slow or there is no speed arrival signal (No.3708.1=1: check the signal) spindle speed is not specified. The spindle speed cannot be checked during the machining.

Example:

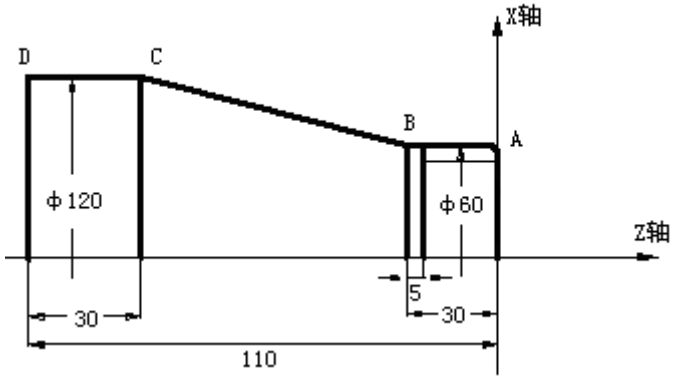


Fig. 2-62

Program:

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

2.20.4 Multiple Thread Cutting Cycle G76

Command 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.2-63.

Command format: G76 P_(m)(r)(a) Q_(△dmin) R_(d);

G76 X(U)__ Z(W)__ R_(i) P_(k) Q_(△d) F_ J_ K_;

Command explanations:

Start point (end point)	Position before block runs and behind blocks run, defined by A point.
End point of thread	End point D of thread cutting defined by X(U) __ Z(W) __.The length axis direction of cutting end point is the one of thread cutting, and position after thread run-out of short axis direction

Start point of thread	Its absolute coordinates is the same that of A point and the different value of X absolute coordinates between C and D 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 position of thread cutting depth	Its absolute coordinates is the same that of A point and the different value of X absolute coordinate between B and C is k(thread taper with radius value).The cutting depth of thread at B point is 0 which is the reference position 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 X absolute coordinate 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, $\triangle d$ 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) tool retraction after the thread cutting in each thread roughing, finishing cycle is completed, is defined by E point
Thread cut-in point	Actual start thread cutting point in each thread roughing cycle and finishing cycle. It is defined by (n is the cutting cycle times), B_n is the first thread roughing cut-in point, $B_1$is the last thread roughing cut-in point, B_e is the thread finishing cut-in point. B_n is the X, Z displacement formula for B point: $$tg\frac{a}{2}=\frac{\lvert Z\ \text{replacement}\rvert}{\lvert X\ \text{replacement}\rvert}$$ a：thread angle
X	X absolute coordinate of thread end point.
U	Difference value of X absolute coordinate between thread end point and start point.
Z	Z absolute coordinate of thread end point.
W	Different value of Z absolute coordinate between thread end point and start point.
P(m)	Times of thread finishing: 01～99 (unit: times) with 2-digit digital. It is valid after m code value is executed, and the value of system parameter No.5142 is rewritten to m. The value of system parameter No.5142 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 code value is executed and the value of system parameter No.5130 is rewritten to r. The value of system parameter No.5130 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.5130 is valid for G92, G76

P(a)	Angle at taper of neighboring two tooth is 0~99, unit: degree(°) ,with 2-digit digital. It is valid after a code value is executed and the value of system parameter No.5143 is rewritten to a. The value of system parameter №05143 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 (radius value without sign symbols). When ($\sqrt{n}-\sqrt{n-1}$) ×△d<△dmin, △dmin is regarded as the cutting travel of current roughing, i.e. depth of current thread cutting is ($\sqrt{n-1}$ ×△d+△dmin) . △dmin is applied because the cutting travel of roughing is undersize and the times of roughing are excessive, which is caused the cutting travel of thread roughing gradually decreases. After Q(△dmin) is executed, the code value △dmin is value and the value of system parameter No.5140 is rewritten to minimum cutting travel; when Q（△dmin） is not input, the system takes No.5140 value as the least cutting value
R(d)	It is the cutting travel of thread finishing, and is the different value(radius value without sign symbols) of X absolute coordinates between cut-in point Be of thread finishing and Bf of thread roughing. After R(d) is executed, the code value d is value and the value of system parameter No.5141 is rewritten to d×1000(unit: 0.001 mm) . The value of system parameter No.5141 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 X absolute coordinate between thread start point and end point (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(radius value without sign symbols), and the system alarms when P(k) is not input
Q(△d)	It is the first depth of thread cutting (radius value without sign symbols).The system alarms when △d is not input
F	Pitch is defined to moving distance (radius value in X direction) of long axis when the spindle rotates one rev. Z 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
J	When the thread run-out is executed, the movement range in the short axis direction is the same that of G32, must not be less than 0 without direction (the system automatically confirms the run-out direction according to the initial point of the program), is modal and its value is specified by radius
K	When the thread run-out is executed, the range in the long axis direction is the same that of G32, is modal without direction, and the value is specified by radius

Address	Incremental system	Metric （mm）input	Inch (inch) input
Q(△dmin	ISB system	0~99999999（unit：0.001mm）	0~99999999（unit：0.0001inch）

)	ISC system	0~99999999（unit：0.0001mm）	0~99999999（unit：0.00001inch）
R（d）	ISB system	0.001～99999.999（mm）	0.0001～9999.9999 (inch)
	ISC system	0.0001～9999.9999（mm）	0.00001～999.99999 (inch)
R（i）	ISB system	-99999.999～99999.999（mm）	-9999.9999～9999.9999 (inch)
	ISC system	-9999.9999～9999.9999（mm）	-999.99999～999.99999 (inch)
P(k)	ISB system	1~99999999（unit：0.001mm）	1~99999999（unit：0.0001inch）
	ISC system	1~99999999（unit：0.0001mm）	1~99999999（unit：0.00001inch）
Q(△d)	ISB system	1~99999999（unit：0.001mm）	1~99999999（unit：0.0001inch）
	ISC system	1~99999999（unit：0.0001mm）	1~99999999（unit：0.00001inch）
F	ISB, ISC	0.01～500 （mm）	0.01～9.99（inch）

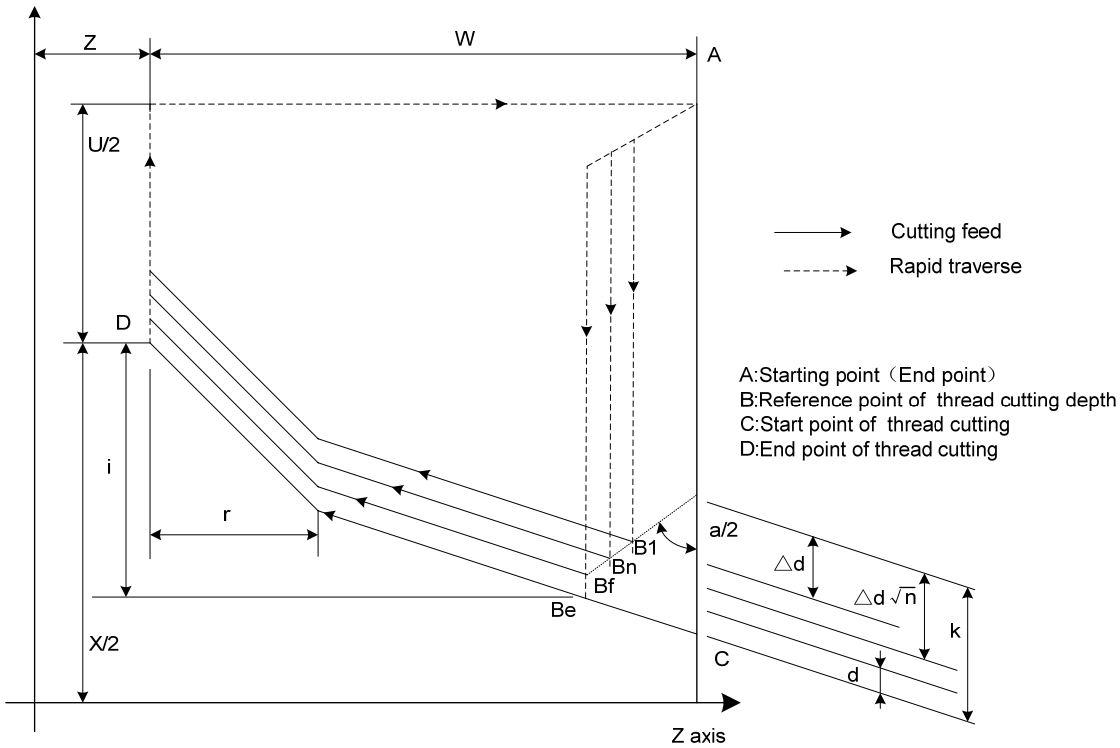


Fig. 2-63

Cut-in method is shown in Fig. 2-64:

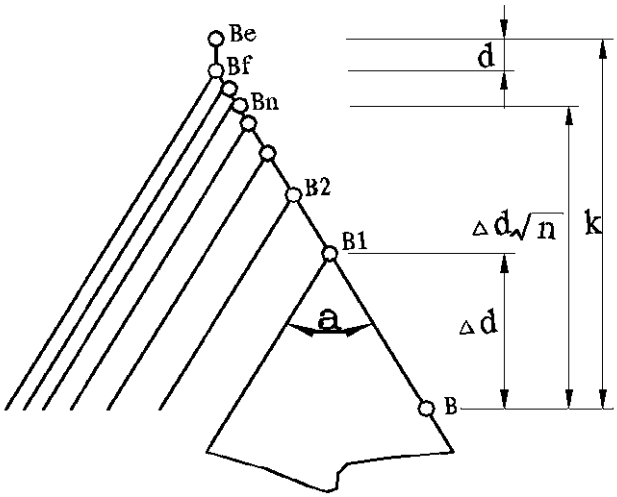


Fig. 2-64

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+△d_{min}) , 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) X axis rapidly traverses to E point;

(8) Z axis traverses to A point and the thread roughing cycle is completed to execute the finishing;

(9) After the tool rapidly traverses to Be (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 time 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.

Note 1: When G76 is executed, after【FEED HOLD】key is pressed and the system executes this thread cutting cycle, the system enters pauses state and "Stop" appears in the status column;

Note 2: The single block is executed during the course of thread cutting, the run stops after the system returns to start point (one thread cutting cycle is completed);

Note 3: The thread cutting speed stops when the system resets, emergently stops or the drive until alarms;

Note 4: All or some addresses of G76 P（m）（r）（a） Q（△dmin） R（d） are omitted, and omitted addresses runs according to the setting value;

Note 5: m, r, a uses the same address P to be input one time. When m, r, a are all omitted, the system runs at the setting value of No.5142, No.5130 or No.5143; when P is with non regular value, the system takes the last two digits of P value as a value, and the last third and fourth digits as r value, and the left as m value;

Note 6: Signs of U, W determines direction of A→C→D→E, R (i) determines that of C→D. Four kind of combination of U, W correspond to 4 kind of machining path;

Note 7: When the set first thread cutting depth is more than the total cutting depth, one roughing is executed, and its cutting depth is equal to the total cutting depth of roughing;

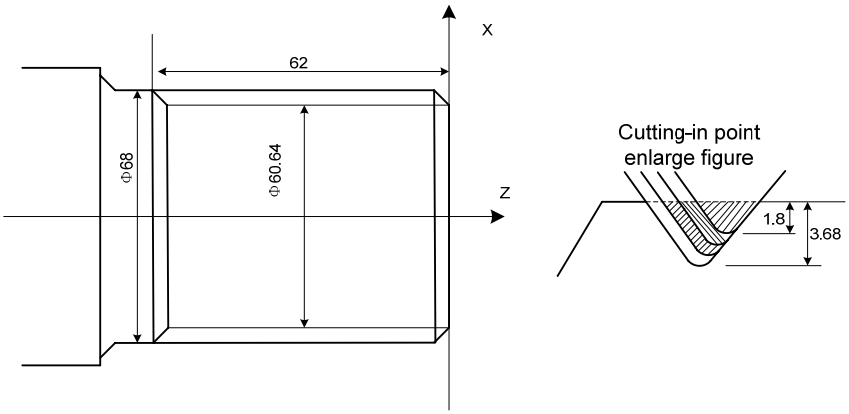
Note 8: When the least cutting amount or finishing allowance is more than thread tooth height in the course of thread roughing, an alarm occurs;

Note 9: When the run-out length is more than the machining thread length of long axis, an alarm occurs;

Note 10: The system automatically checks the spindle speed before machining, an alarm occurs when the spindle speed is not executed. The spindle speed cannot be checked in the course of machining;

Note 11：The run-out format is the same that of G32 when there is J, K.

Example: Fig.2-65，thread M68×6.



Program: O0013；

```
O0013;
G50 X100 Z50 M3 S300;          (Set workpiece coordinate system, start spindle
                                and specify spindle speed)
G00 X80 Z10;                   (Rapid traverse to start point of machining)
G76 P020560 Q150 R0.1;          (Finishing 2 times, chamfering width 3mm, 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 depth 1.8)
G00 X100 Z50 ;                 (Return to start point of program)
M30;                            (End of program)
```

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

G96 command function: The constant surface speed control is valid, the cutting surface speed is defined (m/min) and the constant rotational speed control is cancelled.

Command format: G96 Sxxxx;

Command explanation: G96 is modal G code. If the current modal is G96, G96 can not be input;

it is the cutting surface speed in Sxxxxx constant surface control.

G97 Command function: the constant surface speed control is cancelled, the constant rotational speed control is valid and the spindle speed is defined (r/min).

Command format: G97 Sxxxx;

Command explanation: G97 is modal G code. If the current modal is G97, G97 cannot be input;

It is the spindle speed in Sxxxxx constant speed control(r/min).

Relative code: G50

Command function: define max. spindle speed limit (r/min) in the constant surface speed control(r/min).

Command format: G50 Sxxxx;

Command explanation: After the system is turned on, and the max. spindle speed is not specified, the system does not limit the spindle speed state. Max. spindle speed limit is valid for G96, and is invalid for G97;

S value set by G50 is modal and is value before the new max. speed is set;

Note: when G50 S0 is executed, the spindle speed is limited in 0 r/min (the spindle does not rotate) in the constant surface control;

Address	Incremental system	Metric（mm）input	Inch (inch)input
S（G96）	ISB, ISC	0～20000 m/min	0～2000 feet/min
S（G97）	ISB, ISC	0～20000 r/min	0～20000 r/min

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 speed (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 coordinate of programming path in the constant speed control. If the absolute value of X absolute coordinate increases, the spindle speed reduces, and vice versa, which make the cutting surface speed as S code 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: r/min

|X|: absolute value of X absolute coordinate value (diameter value) π≈3.14

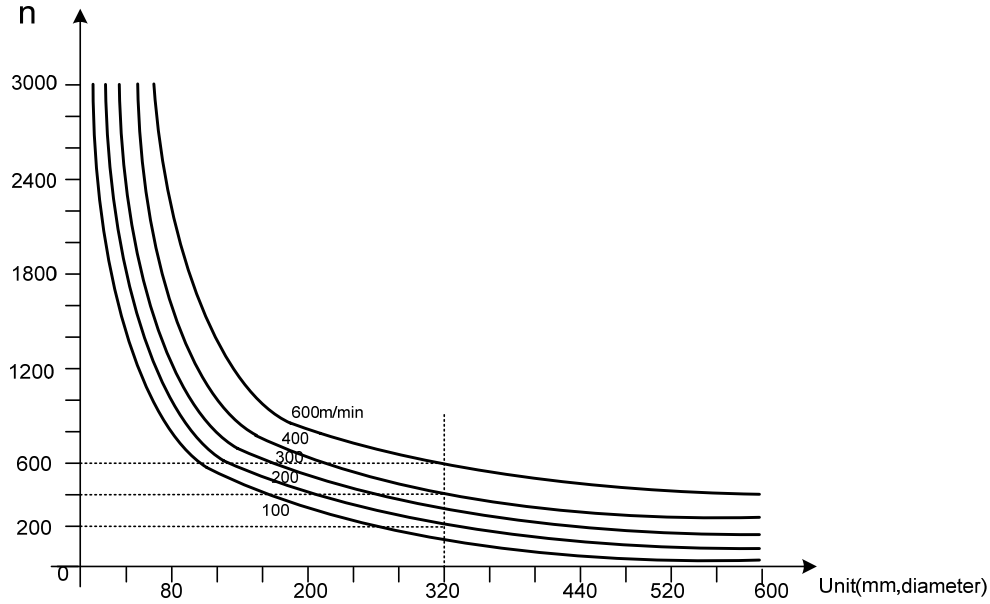


Fig. 2-66

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.

When the constant surface speed is valid, G50 Sxxxxx can limit max. spindle speed (r/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 Sxxxxx. 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 Sxxxxx is reserved before it is defined again and its function is valid in G96. Max. spindle speed defined by G50 Sxxxxx is invalid in G97 but its limit value is reserved.

Note 1: G96, G97 are modal in the same group, and one of them is valid in the same time. G97 is initial word and is valid after the system is turned on.

Note 2: In G96, S value executed is reserved in G97. there is no new S is executed and the S value in the last G96 state is recovered to the current valid surface speed after the system returns to G96 state, the system outputs the least surface speed in G96 when there is no saved value.

Note 3: From G96 to G97, if none of S code (r/min) is executed in the program block in G97, the last spindle speed in G96 is taken as S code in G97, namely, the spindle speed is not changed at this time;

Note 4: The constant surface speed control function is still valid when the machine is locked(X, Z do not move when the system executes X, Z motion codes);

Note 5: In G96, when the spindle speed counted by the cutting surface speed is more than max. speed of current spindle gear, at this time, the spindle speed is limited to max. one of current spindle gear;

Note 6: In thread cutting, 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 7: No.3031 sets the numerical digit permitted by S.

Note 8: X=0: the theory speed is infinite but the actual speed corresponds to 10V voltage because the maximum voltage of sent analog is 10V.

Example:

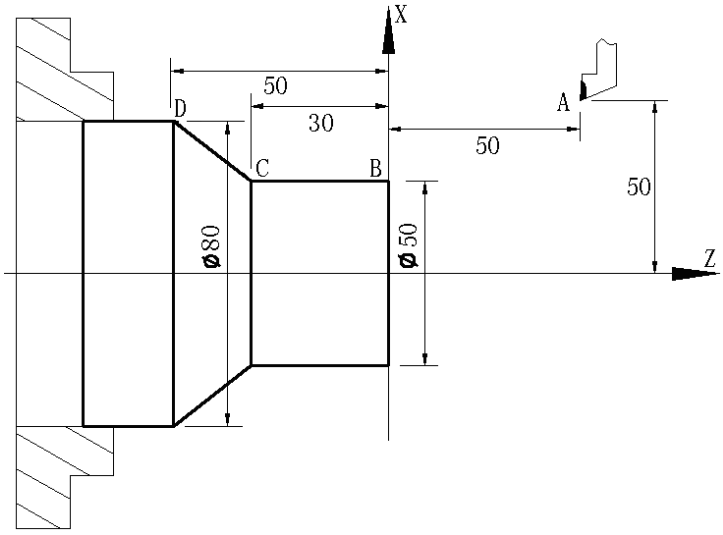


Fig.2-67

Program:

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 r/min)
G0 X50 Z0;	(Rapid traverse to B point with spindle speed 1910 r/min)
G1 W-30 F200;	(Cut from B to C with spindle speed 1910 r/min)
X80 W-20 F150;	(Cut from C to D with spindle speed 1910 r/min and surface speed 1194 r/min)
G0 X100 Z100;	(Rapid retract to A point with spindle speed 955 r/min)
M30;	(End of program, spindle stop and cooling OFF)

2.22 Feedrate per Minute G98/G94, Feedrate per Rev G99/G95

G98 command function: G98 specifies the cutting feedrate is feedrate per minute, G98 is modal. When the current is G98 modal, G98 cannot be input.

Command format: G98 Fxxxx;

G99 command function: G99 specifies the cutting feedrate is feedrate per rotation, G99is modal. When the current is G99 modal, G99 cannot be input.

Command format: G99 Fxxxx;

Command explanation:

When G99 Fxxxx (B set of G code is G95) is executed, the actual cutting feedrate is gotten by multiplying the F code value (mm/r) to the current spindle speed(r/min). If the spindle speed varies, the actual feedrate changes too. If the spindle cutting feed amount per rev is specified by G99 FXXXX (B set of G code is G95), the even cutting texture on the surface of workpiece will be gotten. In G99 (B set of G code is G95)

state, a spindle encoder should be fixed on the machine tool to machine the workpiece.

F range in G98, G99 （B set of G code is G94, G95）is shown below:

Address	Incremental system	Metric（mm）input	Inch (inch)input
F（G98）	ISB system	0．001～60000 (mm/min)	0.00001～2400 (inch/min)
	ISC system	0.001～24000 (mm/min)	0.01～960 (inch/min)
F（G99）	ISB system	0.001～500 (mm/r)	0.0001～9.99 (inch/r)
	ISC system	0.00001～960 (mm/r)	0.0001～9.99 (inch/r)
F（G98）	ISB system	1～60000 (mm/min)	0.01～2400 (inch/min)
	ISC system	1～24000 (mm/min)	0.01～960 (inch/min)
F（G99）	ISB system	0.01～500 (mm/r)	0.01～9.99 (inch/r)
	ISC system	0.01～500 (mm/r)	0.01～9.99 (inch/r)

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) .

F value is reserved after the system executes F code.

Note 1: G98, G99 are the modal G codes in the same group and only one is valid. G98 is the initial state G code and the system defaults the modal can be set by No.3402 Bit4 (FPM) when the system turns on;

Note 2: In G99 mode, there is the uneven cutting feed rate when the spindle speed is lower than 1 r/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 converter;

Note 3: No.1422 set the upper of the cutting feedrate. When the actual cutting feedrate (the value is multiplied by the override) exceeds the specified upper limit, it is clamped to the upper limit value;

Note 4: No. 1403 Bit0(MIF)can set the cutting speed unit per minute and the detailed is referred to Ⅱ Operation;

Note 5: When G99 instead of F code in G98 mode is executed, F is the previous modal value in G99. In a similar way, when G98 instead of F code in G99 mode is executed, F is the previous modal value in G98;

Note 6: When the initial mode is G98/99, and G99/G98 is alone executed after power on, the system runs at the speed set by No. 1411.

2.23 Drilling/Boring Fixed Cycle Code

Many blocks completes one machining in the course of drilling. To simplify programming, GSK988TA/TB uses one drilling cycle G codes to complete a series of drilling machining. (C tool compensation vector in the course of drilling/boring will temporarily cancel, automatically recovers

after the code is completed.)

- Execution process:
  The drilling fixed cycle is composed of the following 6 operations.
  Operation 1: X(Z) and C axis (requirement in some occasion) positions to the hole position of initial plane;
  Operation 2: rapidly traverse to point R;
  Operation 3: drilling (cutting feed or interval feed);
  Operation 4: pause at the hole bottom;
  Operation 5: retract tool to the plane where point R is;
  Operation 6: rapidly traverse to initial plane

Operation sequence

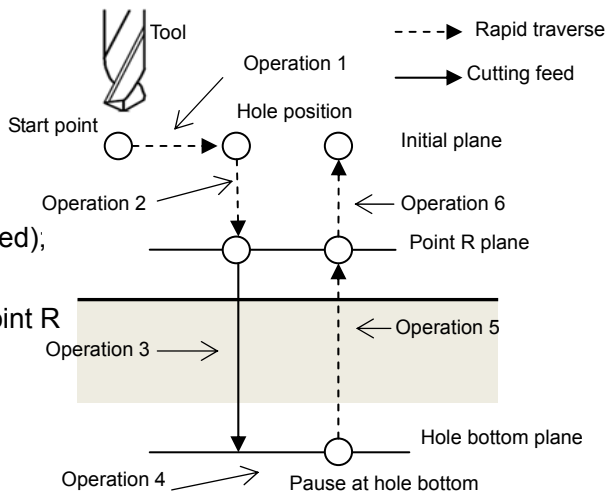


Fig.2-68

- Drilling fixed cycle G codes（included in Group 10）

G code	Drilling axis	Drilling operation	Operation at bottom	Tool retraction	Function
G83	Z	Interval feed/ cutting feed	Pause	Rapid traverse	End drilling cycle
G87	X	Interval feed / cutting feed	Pause	Rapid traverse	Side drilling cycle
G85	Z	Cutting feed	Pause	Cutting feed	End boring cycle
G89	X	Cutting feed	Pause	Cutting feed	Side boring cycle
G80	/	/	/	/	Cancel drilling fixed cycle

- Positioning axis and drilling axis
  G confirms the drilling axis and the positioning axis is the others except for the drilling axis.

G code	Drilling axis	Positioning axis
G83, G85	Z	X and C
G87, G89	X	Z and C

Note: C axis can be omitted.

- Fixed cycle is cancelled
  G80 or G codes included in Group 01 can cancel the fixed cycle.

2.23.1 End drilling cycle G83 /side drilling cycle G87

Command format: G83 X(U)_ C(H)_ Z(W)_ R_ P_ Q_ F_ K_ M_;
G87 Z(W)_ C(H)_ X(U)_ R_ P_ Q_ F_ K_ M_;

Code definition:

X_ C_ or Z_ C_	It is hole position data, and valid in the specified block
Z(W)_ or X(U)_	The absolute value specifies the coordinates of hole bottom or the incremental value specifies the distance from Point R plane to the hole bottom, which is value in the specified block.
R_	It is the distance from the initial plane to point R, is specified by radius value with direction. Its unit and range are shown in the following table.
P_	It is pause time at the bottom. ISB system unit is 1ms，ISC system unit is 0.1ms.
Q_	It is cutting amount every time and specified by radius value. Cutting amount, radius value every time, unit and range are shown in the following table.
F_	Cutting feedrate.
K_	Program execution times (if necessary).
M_	M code for clamping C axis (if necessary)

	Incremental system	Metric input （mm）	Inch input (inch)
Q	ISB system	0~99999999（unit ：0.001mm）	0~99999999（unit：0.0001inch）
	ISC system	0~99999999（unit ：0.0001mm）	0~99999999（unit ：0.00001inch）
R	ISB system	-99999.999～99999.999mm	-9999.9999～9999.9999 inch
	ISC system	-9999.9999～9999.9999 mm	-999.99999～999.99999 inch
K	ISB system	1～99 times	1～99 times
	ISC system	1～99 times	1～99 times

In G83/87, high speed deep hole drilling cycle, deep hole drilling cycle and standard drilling cycle can be selected by Q value (cutting amount every time) and RTR (NO.5101#2).

High speed deep hole drilling cycle	Q value is specified（Q value is not zero） and the parameter RTR（NO.5101#2）=“0”
Deep hole drilling cycle	Q value is specified（Q value is not zero） and the parameter RTR（NO.5101#2）=“1”
Standard drilling cycle	Q value is not specified or Q value is zero.

G83, G87 are modal, remain valid once are specified until the fixed cycle is cancelled.

● **High speed deep hole drilling cycle**（Q value is specified (it is not zero) and RTR（NO.5101#2）=“0”）

The system executes the intermittent cutting and chip removal with the specified tool retraction amount before entering the hole bottom, which is executed repetitively until the tool infeeds to the bottom, and then the tool retraction is performed, so the machining is completed. Command format and definition are referred to the previous description.

Execution process:

(1)The tool rapidly positions to the hole from starting point（the hole is determined by the hole position data at the initial plane）;

(2)Rapidly position to point R;

(3)Cutting feed executes the cutting amount q specified ;

(4)Rapid tool retraction executes retraction amount d specified by No. 5114;

(5)Repeat the above Step ③④ until the tool reaches the plane where the hole bottom is;

(6)Pause is executed in the time specified by P;

(7)Return rapidly to the plane where point R is, execute M(a+1) and pause the time specified by P2;

(8)Return rapidly to the initial plane;

(9)Drilling cycle ends.

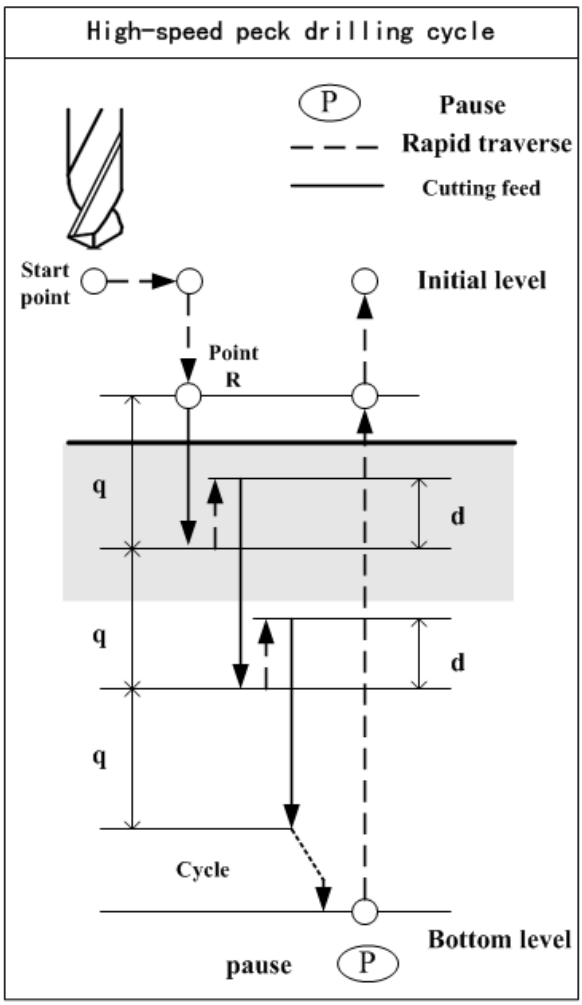


Fig.2-69

● **Deep hole drilling cycle**（specify Q value and RTR（NO.5101#2）="1"）

the command format and definition are referred to the previous description.

Execution process:

(1)The tool rapidly positions to the hole from starting point （the hole is determined by the hole position data at the initial plane）, and execute Ma;

(2)Rapidly position to point R;

(3)Cutting feed executes the cutting amount q specified by Q;

(4)Rapidly retract the tool to the plane where R point is;

(5)Rapidly retract the tool to the distance d from the previous machine plane (idle travel d of deep hole drilling cycle specified by No.5115);

(6)Cutting feed the distance q+d ;

(7)Repeat the above steps ④⑤⑥ till the tool reaches the plane where the hole bottom is;

(8)Pause is executed in the time specified by P;

(9)Return rapidly to the plane where point R is execute M(a+1) and pause the time specified by P2;

(10)return rapidly to the initial plane;

(11)Drilling cycle ends.

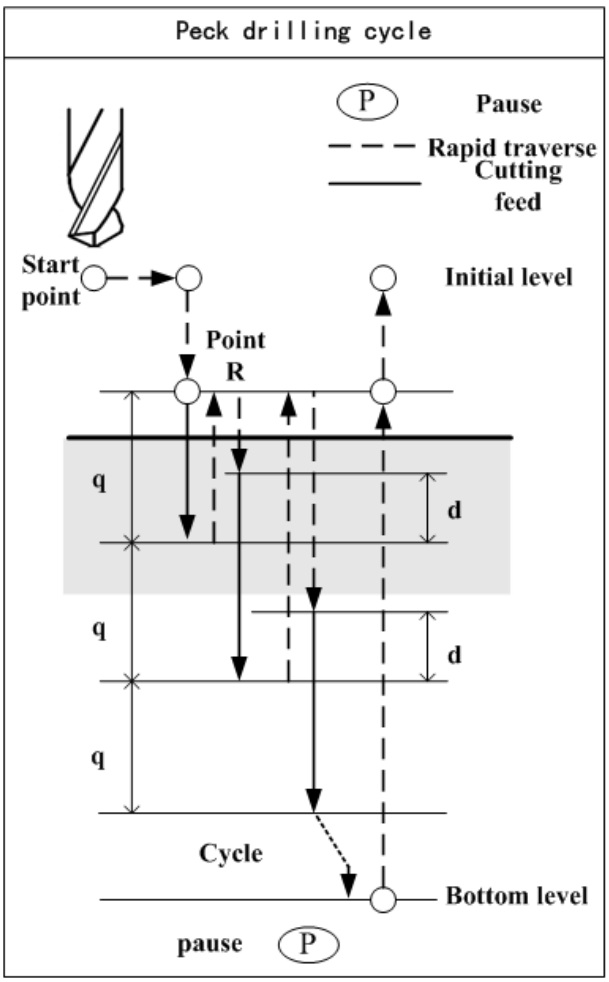


Fig.2-70

● **Standard drilling cycle**（Q value is not specified）

Command format: G83 X(U)_ C(H)_ Z(W)_ R_ P_ F_ K_ M_; or
G87 Z(W)_ C(H)_ X(U)_ R_ P_ F_ K_ M_;

Command explanation: the code definition is referred to the previous description.

Execution process:

(1)The tool rapidly positions to the hole from starting point（the hole is determined by the hole position data at the initial plane）, and execute Ma;

(2)Rapidly position to point R;

(3)Cutting feed to the plane where the hole bottom is;

(4)Pause is executed in the time specified by P;

(5)Return rapidly to the plane where point R is, execute M(a+1) and pause the time specified by P2;

(6)Return rapidly to the initial plane;

(7)Drilling cycle ends.

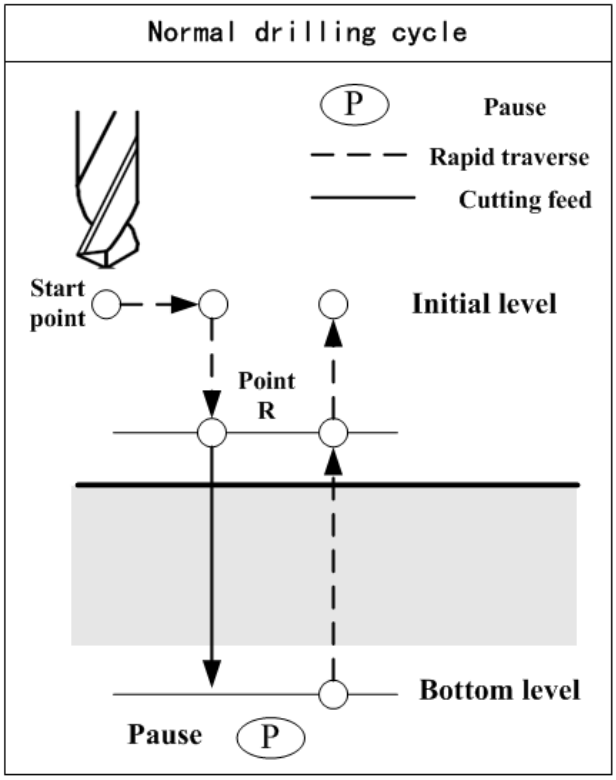


Fig. 2-71

Program example:

Suppose the current system is ISB, the minimum input unit is 0.001mm and RTR is set to 1.

G98;	feed mode per minute
M14;	activate C indexing（suppose M14 is for activating C indexing）
M3 S1500;	the tool starts rotation

G0 X50 C0 Z0；	X, Z and C axis position to the starting point
G83 X100 Z-50 R-4 Q5000 P3000 F200；	starting point is X50 C0，hole position is X100 C0, point R is X100 Z-4，hole position is X100 Z-50, the cutting amount every time is 5mm, pause time is 3s. the block is for deep hole drilling according to Q value and RTR
C120；	position to C120 to drilling the 2nd point
C240；	position to C240 to drilling the 3rd point
G80 M05；	the fixed cycle is cancelled, the tool stops rotation
M15；	C axis indexing closes（suppose M15 is for closing C axis indexing）
M30；	end of program

2.23.2 End Boring CycleG85 / Side Boring Cycle G89

Command format: G85 X(U)_ C(H)_ Z(W)_ R_ P_ F_ K_ M_；
G89 Z(W)_ C(H)_ X(U)_ R_ P_ F_ K_ M_；

Code definition:

X_ C_ or Z_ C_	It is the hole position data and is valid only in the specified block.
Z(W)_or X(U)_	It specifies the coordinate value of hole bottom by using absolute coordinate , or specifies the distance from point R plane to the hole bottom by using incremental value, and it is valid in the specified block.
R_	It is the distance from the initial plane to point R and is specified by radius value with direction. Its unit and range is shown below.
P_	Hole bottom pause time. Unit of ISB system is 1ms and ISC is 0.1ms.
F_	Cutting feedrate.
K_	Execution times of program（if neccessary）.
M_	M code for clamping C axis（if neccessary）.

Relevant command explanations are referred to those of G83/87.

Execution process:

(1)The tool rapidly positions to the hole from starting point（the hole is determined by the hole position data at the initial plane）, and execute Ma;

(2)Rapidly position to point R;

(3)Cutting feed to the plane where the hole bottom is at the speed specified by F;

(4)Pause is executed in the time specified by P;

(5)Return rapidly to the plane where point R is, execute M(a+1) and pause the time specified by P2 (No.5149 is used to set override of boring retraction operation. When it is set to 0, the system defaults the tool retracts at the speed of F value's twice) ;

(6)Return rapidly to the initial plane;

(7)Drilling cycle ends.

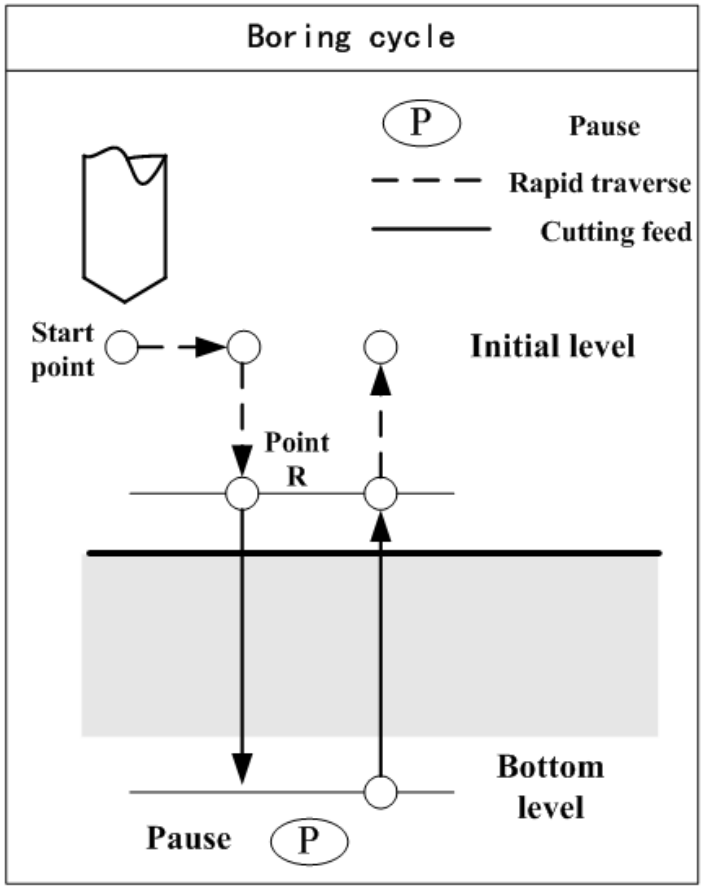


Fig.2-72

2.23.3 Cancelling Drilling/Boring G80

The code is used for cancel the drilling fixed cycle.

Command format: G80;

After G80 is executed, the hole position data, R and other drilling data are cancelled, and the mode of drilling cycle is also done.

2.23.4 Notes for Drilling/Boring Cycle

Note 1: When the reset or emergency stop is executed, the mode of drilling cycle remains. The user must pay more attention to it when the program is started again.

Note 2: The single block stops at end point of operation 1, operation 2 or operation 6.

Note 3: When drilling/boring cycle is executed, creating or cancelling tool compensation code is executed, the code is valid after the cycle ends.

2.24 Tapping Cycle Code

GSK988TA/TB CNC Turning System uses end tapping cycle (G84) and side tapping cycle (G88) to complete the tapping function. Tapping is divided into common tapping (flexible) and rigid tapping mode. In the common tapping mode, the spindle rotation and feed amount of tapping axis are controlled separately, their synchronous relationship is not controlled well. In the rigid tapping mode, the control of spindle motor is the same that of servo motor, the spindle rotating one circle corresponds to some axial feed amount of the spindle even if the spindle accelerates/decelerates. In the rigid tapping, the spindle can rapidly and exactly tap without using the floating chuck or variable screw tap (use it in the common tapping mode).

M29 (it can set other M code according to parameter or directly use G code to specify rigid mode without M code)specifies the rigid tapping cycle when programming.

When the rigid tapping is executed, the machine must have the corresponding conditions, i.e. the spindle uses the position control and is applied to Cs axis, otherwise, the system does not support the function. The function is applied to the machine with high configuration.

End tapping cycle (G84), side tapping cycle (G88), drilling fixed cycle G83/G87 and boring cycle G85/G89 are in the same Group 10. G80 or one code included in Group 01 can cancel the tapping fixed cycle. The system executes the normal operation after the drilling fixed cycle is cancelled. Clear point R and hole bottom (point X or Z) data and other tapping data (P, K, F) is also cleared.

Vector of C tool compensation during the course of tapping is temporarily cancelled, but automatically recovers after the code is executed.

2.24.1 Tapping Mode

Tapping cycle is divided into common mode and rigid tapping mode, and the follow method can specify the rigid tapping mode; when N0.5200#0=0 and M29 is not specified, the system executes the common tapping mode

Specify M29 S**** before G84 (G88) blocks;

```
M29 S    ;
G84 X    C    R    P    F    K    (M    );
X    C    ;
G80;
```

It is specified in the same block in G84 (G88) tapping blocks; M code for clamping C axis cannot

be specified in G84/G88 blocks in the mode.

G84 X_ C_ Z_ R _ P_ F_ K_ M29 S_;
X_ C_;
G80;

G84/G88 is used for rigid tapping(Bit0 of No.5200 is set to 1); in the mode, G84/G88 is used for only the rigid tapping mode instead of the common tapping mode.

G84 X_ C_ Z_ R _ P_ F_ K_ M _;
X_ C_;
G80;

M29 (the parameter sets other M code to specify it) is for rigid tapping, the system alarms when S is specified between M29 and G84/G88 blocks or the axis movement code is specified; the system alarms when M29 is specified repetitively in tapping cycle (M29 cannot be specified repetitively).

M29 Sxxxx codes rigid tapping mode. The corresponding switch is done after PLC receives M29 and the spindle stops rotation. The spindle output is equivalent to S0 output in M29.

2.24.2 End Rigid Tapping Cycle (G84) / Side Rigid Tapping Cycle (G88)

Command function: When the spindle is controlled in rigid mode (it is taken as the servo motor), the rigid tapping cycle is executed.

Command format: G84 X (U)_ C (H)_ Z (W)_ R_ Q_ P_ F_ K_ M_ ; or
G88 Z (W)_ C (H)_ X (U)_ R_ Q_ P_ F_ K_ M_ ;

X_ C_ or Z_ C_	It is the hole position data and is valid only in the specified block; the hole position data can specify other valid axes except for X, Z, C.
Z(W)_ or X(U)_	It specifies the coordinate value of hole bottom by using absolute coordinate, or specifies the distance from point R plane to the hole bottom by using incremental value, and it is valid in the specified block.
R_	It is the distance from the initial plane to point R and is specified by radius value with direction. Its unit and range is shown below.
P_	Hole bottom pause time. Unit of ISB system is 1ms and ISC is 0.1ms.
Q_	Cutting amount every time is specified by radius value. Its unit and range are shown below. When Q value is specified, G84/G88 selects the high speed deep hole rigid tapping cycle or deep hole rigid tapping cycle by PCP (No. 5200#5). Q value is not specified or Q value is 0, the standard rigid tapping cycle is selected.
F_	Cutting feedrate.
K_	Execution times of program（if neccessary）.
M_	M code for clamping C axis（if neccessary）.

The tapping feed axis specifies X or Z by G84/G88. The tapping axis specified by G84 is Z an the one specified by G88 is X. the relevant G signal confirms to select a spindle (it is relevant with the PLC).

	Incremental system	Metric input（mm）	Inch input(inch)
Q	ISB system	0~99999999（unit：0.001mm）	0~99999999（unit：0.0001inch）
	ISC system	0~99999999（unit：0.0001mm）	0~99999999（unit：0.00001inch）
R	ISB system	-99999.999～99999.999mm	-9999.9999～9999.9999 inch
	ISC system	-9999.9999 ～9999.9999 mm	-999.99999 ～999.99999 inch

The thread lead is determined by the cutting feedrate F(i.e., the tapping axis' feedrate) and the spindle speed S.

In per minute mode, thread lead =code cutting feedrate F /spindle speed.

In per rotation mode, thread lead =code feedrate F.

In rigid tapping mode, the system can select three kind of machining mode by Q value (cutting amount every time) and PCP (NO.5200#): Standard rigid tapping cycle, high-speed deep hole rigid tapping cycle, and deep hold rigid tapping cycle.

Standard rigid tapping cycle	Q value is not specified or Q value is 0
High-speed deep hole rigid tapping cycle	Specify Q value（it is not zero） and PCP（NO.5200#5）="0"
Deep hold rigid tapping cycle	Specify Q value（it is not zero） and PCP（NO.5200#5）="1"

● Standard rigid tapping cycle（Q value is not specified or Q value is 0）.

Command format:

G84 X (U)_ C (H)_ Z (W)_ R_ P_ F_ K_ M_ ;
G88 Z (W)_ C (H)_ X (U)_ R_ P_ F_ K_ M_ ;

Execution process:

(1)The tool rapidly positions to the hole from starting point (the hole is determined by the hole position data at the initial plane）;

(2)Rapidly position to point R;

(3)The spindle starts rotation, and tapping axis is executed to the hole bottom plane at the speed specified by F, and the spindle stops when the axis reaches the hole bottom;

(4)Pause is executed in the time specified by P;

(5)The spindle starts rotation reversely and tapping axis retracts to the Point R plane at the speed specified by F, the spindle stops rotation at R point; execute M (a+1) and pause the time specified P2;

(6)return rapidly to the initial plane;

(7)The standard tapping cycle ends.

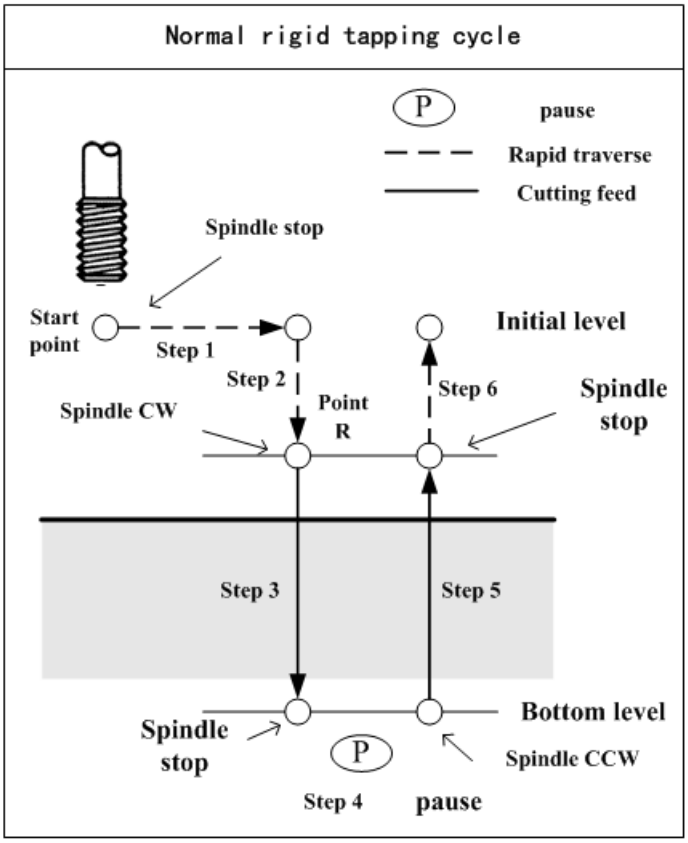


Fig. 2-73

- High speed deep hole rigid tapping cycle (Q value is specified（it is not zero）and PCP（NO.5200#5）="0")

  Before the tool enters the hole bottom, the intermittent tapping is executed and the chip removal is done with the specified tool retraction amount, which are done repetitive until the tool reaches the hole bottom, then the tool retracts and the machining ends.

  Command format: G84 X (U)_ C (H)_ Z (W)_ R_ Q_ P_ F_ K_ M_ ; or
  G88 Z (W)_ C (H)_ X (U)_ R_ Q_ P_ F_ K_ M_ ;

Execution processing:

(1)The tool rapidly traverses to the hole bottom position from the start point (i.e., the start point confirmed by the hole position data in the initial plane)；

(2)Rapidly position to point R;

(3)The spindle starts to rotate;

(4)The tapping axis feeds the cutting amount q specified by the cutting feedrate F; after the feed ends, the spindle stops rotation;

(5)The spindle rotates reversely, and the tapping axis executes the deep-hole rigid tapping's retraction amount d specified by NO.5213；the spindle stops rotation after the tool retraction ends;

(6)Repeat the above Step 3, 4, 5 till the tool reaches the hole bottom plane; the spindle stops rotation;

(7)The system dwells the time specified by P；

(8)The spindle rotates oppositely, and the tapping axis returns to point R plane at the special speed;

(9)Rapidly return to the initial plane；

(10)The high-speed deep-hole rigid tapping cycle ends.

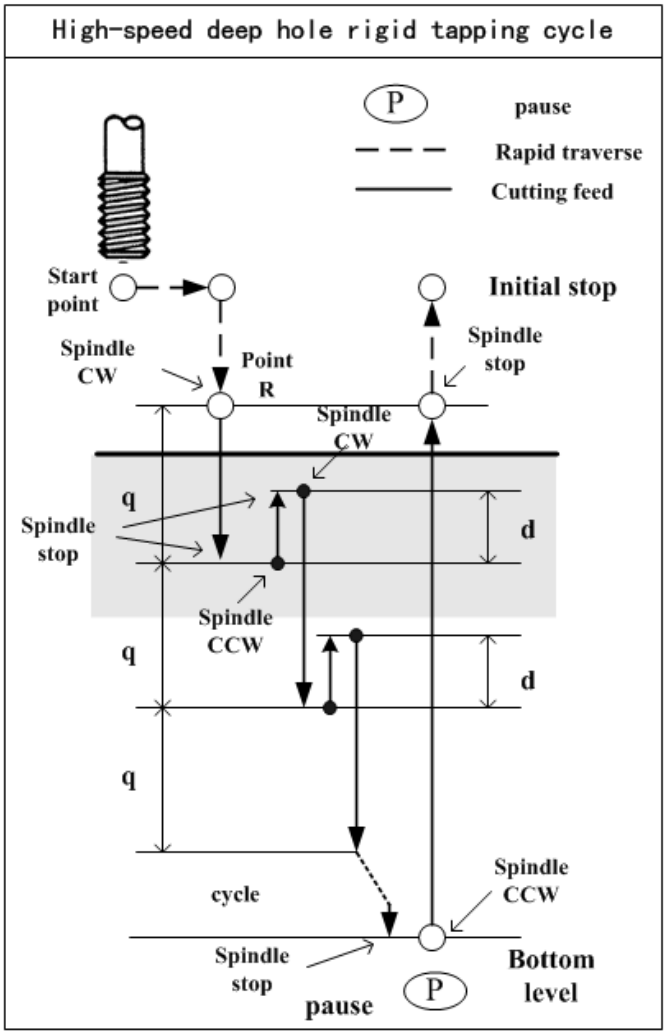


Fig. 2-74

- **Deep hole rigid tapping cycle**（Q value is specified（it is not zero）and RTR（NO.5200#5）="1"）
  The cycle executes the deep hole rigid tapping operation.

  Command format: G84 X (U)_ C (H)_ Z (W)_ R_ Q_ P_ F_ K_ M_ ; or
  G88 Z (W)_ C (H)_ X (U)_ R_ Q_ P_ F_ K_ M_ ;

Execution processing:

(1)The tool rapidly traverses to the hole
bottom position from the start point (i.e., the start point confirmed by the hole position data in the initial plane)；

(2)Rapidly position to point R;

(3)The spindle starts to rotate;

(4)The tapping axis feeds the cutting amount q specified by the cutting feedrate F; after the feed ends, the spindle stops rotation;

(5)The spindle rotates reversely, and after the tapping axis executes the tool retraction to point R plane, the spindle stops rotation

(6)The spindle starts rotation; the tapping axis executes the tool infeed to the distance d (the retraction amount d of the deep-hole rigid tapping specified by No. 5213) away from the previous machining plane;

(7)The tapping axis executes the cutting feed to the distance q+d；
(8)Repeat the above Step 5, 6, 7 till the tool reaches the hole bottom plane; the spindle stops rotation
(9)The system dwells the time specified by P；
(10)The spindle rotates oppositely, and the tapping axis returns to point R plane at the special speed;
(11)Rapidly return to the initial plane；
(12)The high-speed deep-hole rigid tapping cycle ends.

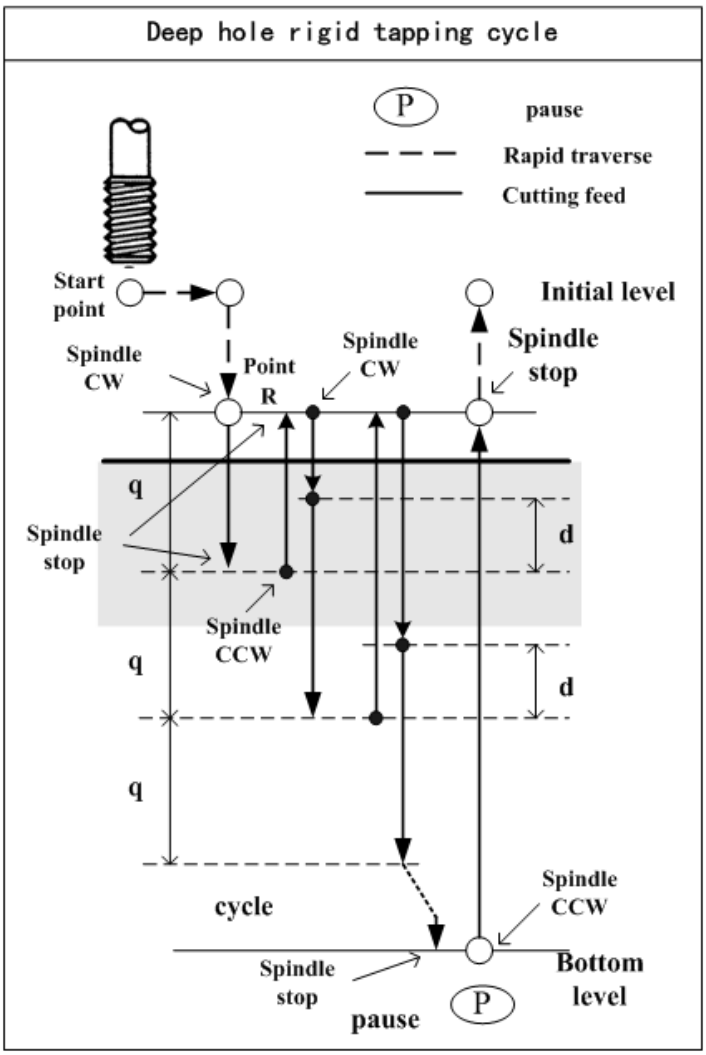


Fig. 2-75

Note 1：In the rigid tapping cycle, the tool retraction speed and the cut-in previous machining plane's speed are confirmed by the feedrate F (for the tapping axis, the federate is the specified F, used to differentiate G98 and G99；for the spindle, the feedrate is to specify the spindle speed) and the drawing override.

When DOV(NO.5200#4，the tool retraction is valid or not when the rigid tapping is performed) is set to 0, the drawing override is fixed to 100%.

When DOV(NO.5200#4) is set to 1，it is divided into two conditions:

（1）When OV3（NO.5201#4，the address J specifies the spindle speed is valid or not during the tool retraction, which can confirm the drawing override）. It is set to 0，the drawing override is set by NO.5211（override value during the rigid tapping drawing，thereinto，OVU(NO.5201#3)

It is used to set the setting unit of rigid tapping's drawing override parameter. i.e., NO.5211 unit is 1% or 10%。

（2）When OV3 is set to 1, J address specifies the spindle speed during the tool retraction.

Override value (%) = 100% × spindle speed (J) in drawing/ spindle speed in tapping tool infeed(S)

Besides, the override value exceeds the range 100%～2000%，it becomes 100%. When the spindle speed's address "J" during drawing is specified in rigid tapping mode, it is valid till the fixed tapping cycle is cancelled.

OVE(No.5202#6)="0":

Spindle speed code in drawing		DOV= "1"		DOV= "0"
		OV3= "1"	OV3= "0"	
Spindle speed code with "J" specifying drawing	Within 100～200%	Program code	(No.5211)	100%
	Beyond 100～200%	100%		
Spindle speed code without "J" specifying drawing		(No.5211)		

OVE(No.5202#6)="1":

Spindle speed code in drawing		DOV= "1"		DOV= "0"
		OV3= "1"	OV3= "0"	
Spindle speed code with "J" specifying drawing	Within 100～2000%	Program code	(No.5211)	100%
	Beyond 100～2000%	100%		
Spindle speed code without "J" specifying drawing		(No.5211)		

Note 2：P/Q is specified in the drilling blocks. When it is not specified in the drilling blocks, it is not taken as the modal data to store.

When Q0 is specified, the system does not execute the deep-hole rigid tapping operation.

Note 3：The retraction amount d (No.5213) in the deep-hole tapping cycle should be less than the cutting amount q.

Note 4：R code is the distance from the initial plane to R point, is expressed with a radius value, can be omitted. After it is omitted, the initial plane is the R plane.

Note 5：G84/G88 can be used to the dry run. The feedrate F is the feedrate in the dry run.

Note 6：For the feed pause, single block, when G84/G88 is at the operation 1, operation 2 and operation 6, "Feed pause" is pressed to decelerate to stop；when it is at the operation 3, 4 and 5（during tapping），the movement does not stop. When the tool returns to the point R plane, the feed stops. When G84/G88 is the single block mode or the single block mode is open in the cycle, the single block stop is at the operation 1， 2，6's end point（the operation3，4, 5 and 6 are combined into one single block）.

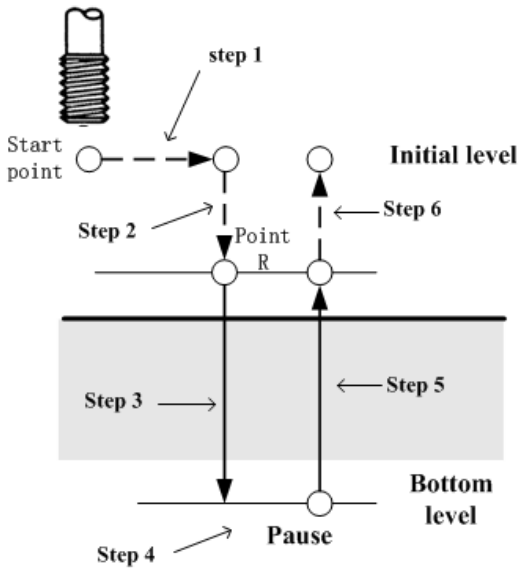


Fig. 2-76

Note 7: When the tool radius compensation in tapping cycle is temporarily cancelled, it is recovered when the fixed cycle is cancelled.

Note 8：When the fixed cycle in the rigid tapping is cancelled, the S value used in the rigid tapping is also cleared. (its state is the same with the specified S0). Namely, S used to specifiy the rigid tapping cannot be used in the program following the fixed cycle program cancelling the rigid tapping.

Note 9：After the rigid tapping's fixed cycle is cancelled, S is specified again according to requirements.

Note 10：N0.5209#0=0, i.e., "in the rigid tapping, the drilling axis is selected by the plane selection". In G84, when G17, G18, G19 is separately specified, the drilling axis is the basic axis X, Z, Y; in G88, when G17, G18, G19 is separately specified, the drilling axis is the basic axis Y, X, Z.

Note 11：The left-hand thread rigid tapping is realized. When GSK988TA/TB G84/G88 rigid tapping's tool infeed is performed with default, the spindle rotates CW, when the rigid tapping's tool retraction is done, the spindle rotates CCW. In some special applications, when the tapping tool infeed is needed, the spindle rotates CCW, but when the tapping's tool retraction is performed, the spindle rotates CW. When the left-hand thread tapping is executed, GSK988TA/TB uses the rigid tapping's spindle rotation direction selection signal（RGROD，i.e., G61.2 in PLC address）to realize the left-hand thread tapping. Before G84/G88 rigid tapping is executed, the CNC checks the state of rigid tapping's spindle rotation direction selection signal to confirm the tapping axis' rotation direction. When RGROD signal is 0 and G84/G88 rigid tapping's tool infeed is executed, the spindle rotation CW; when the rigid tapping's too retraction is performed, the spindle rotates CCW, which is the normal thread tapping; when RGROD signal is 1 and the tapping's tool infeed is performed, the spindle rotates CCW; when the tapping's tool retraction is done, the spindle rotates CW, which is the left-hand thread tapping. After the CNC is turned on, RGROD signal value is defaulted to 0.

During G84/G88 rigid tapping's execution process, RGROD state cannot be changed. After G80, RGROD state can be set again. Or it is set first before G84/G88 rigid tapping is executed. RGROD is added to the ladder, which can realize the left-hand thread rigid tapping.

Program example

Suppose the current system is ISB, the least input increment is 0.001mm

G98；	Feed per minute mode
M29 S1000；	Switch to the rigid tapping mode (very important), code the spindle speed 1000 rev/min. After the block is executed, the spindle does not start rotating.
G0 X50 Z0；	X and Z position to the start point
G84 Z-50 P3000 F2000；	The start point is X50 Z0，the hole position is the same with the start point, the pause time is 3 seconds, and executing F value and S value confirm the lead to 0. When Q value is not executed, it is a standard rigid tapping cycle.
G80；	The fixed cycle is cancelled and the driving tool stops rotation
M30；	End of program

2.24.3 End Common Tapping Cycle G84/Side Common Tapping Cycle G88

When G84/G88 is a common tapping mode, the system uses the miscellaneous function to control the spindle Start/Stop：M03(the spindle rotation CW), M04(the spindle rotation CCW) and M05(the spindle stop)；the CNC detects the spindle rotation by the spindle encoder and the tapping axis rotates along with the spindle. When the machine cannot use the rigid tapping function, the common tapping mode provides an economic tapping method.

When the system executes the common mode to tapping, the spindle must use a flexible chuck or the tool use a variable screw tap.

Command function: the spindle rotating one rotation makes Z move one pitch, which keeps consistent with the screw tap's pitch, forming a spiral grooving in the workpiece's inner hole to complete its thread machining one time. **Note: it is different from the spindle tapping**.

Command format：G84 X (U)_ C (H)_ Z (W)_ R_ P_ F_ K_ M_ ;
G88 Z (W)_ C (H)_ X (U)_ R_ P_ F_ K_ M_ ;

Command explanation:

X_ C_ or Z_ C_	It is the hole position data and is valid in the specified blocks; specify other axes which are not X, Z,C axis are valid at the hole position data.
Z(W)_ or X(U)_	Using an absolute value specifies the hole bottom's coordinate value or using an incremental value specifies the distance from point R plane to the hole bottom, and is valid in the specified blocks.
R_	Distance from the initial plane to point R, is presented with radius value with direction.
P_	Hole bottom pause time, ISB system time is 1ms，ISC system time is 0.1ms.
F_	Cutting feedrate，
K_	Program execution times（if necessary）.
M_	M code for C clamping（if necessary）.

	Incremental system	Metric input（mm）	Inch input (inch)
R	ISB system	-99999.999～99999.999mm	-9999.9999～9999.9999 inch
	ISC system	-9999.9999～9999.9999 mm	-999.99999～999.99999 inch

The tapping feed axis specifies X or Z according to G84/G88. The tapping axis specified by G84 is Z and the tapping axis specified by G88 is X. The relevant G signal selects one spindle (relevant with the PLC).

The cutting feedrate F (i.e. the tapping axis' feedrate) and spindle speed S confirm the thread's lead.

In feed per minute mode, thread lead formular=cutting feedrate F /spindle speed S；

In feed per rotation mode, thread lead formular=cutting feedrate F.

Note: Before the spindle speed S specifies the common tapping, the CNC memorizes the spindle speed S's modal value. The CNC counts the thread lead according to S's modal value and F value specified in the code. When a common tapping is executed, the spindle override is influenced by N0.3708#6.

When G84/G88 is the rigid tapping cycle, Q code and the parameter PCP determine three kind of rigid tapping mode: standard rigid tapping cycle, high-speed deep-hole rigid tapping cycle and deep hole rigid tapping cycle. When G84/G88 is the common tapping cycle, there is only one mode below.

Before G84/G88, the spindle rotation (the operator confirms the spindle rotation CW or CCW according to the used screw tap) can be specified firstly, and the CNC can confirm the M code for the spindl's reverse rotation according to G84/G88's previous spindle rotation direction; when it is not specified, the spindle defaults the spindle rotation CW M03 when the system executes G84/G88 common tapping cycle.

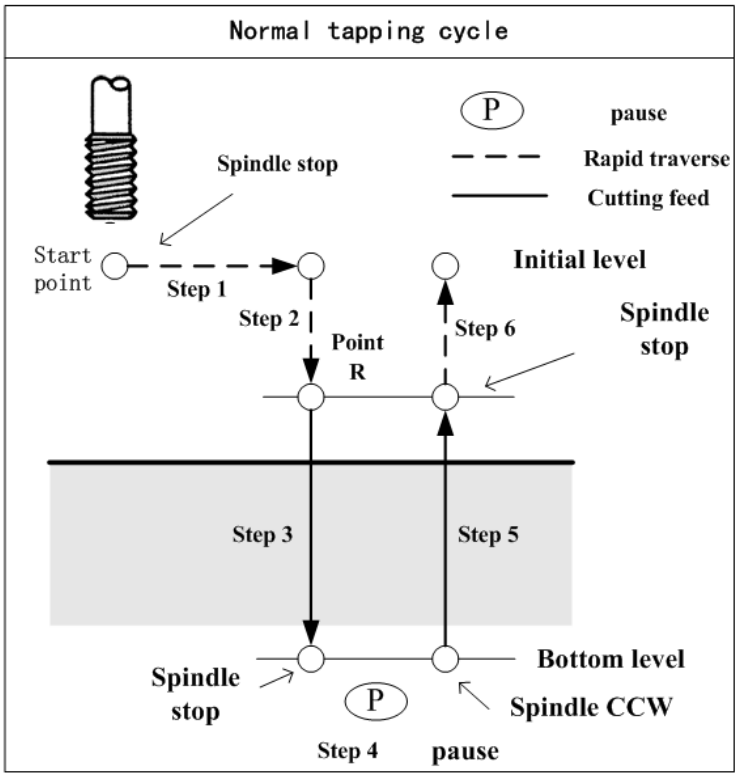


Fig. 2-77

Execution process:

① The tool rapidly positions to the hole position from the start point (i.e., the point confirmed by the hole position data in the initial plane);

② Rapidly position to point R;

③ The spindle rotation M code outputting makes the spindle rotate (the spindle rotation before the tapping cycle is executed, here M code does not output);

④ The tapping axis rotates along with the spindle rotation to cut to the hole bottom plane at the speed specified by F; (when it is about to reach the programmed hole bottom's coordinate position, the spindle stops M05 output, it starts to decelerate to stop, and the tapping axis remains feed till the spindle completely stops rotation);

⑤ Dwell is executed in the time specified by P;

⑥ The M code for spindle's reverse rotation outputs (the spindle rotation direction is opposite to the tool infeed);

⑦ The tapping axis executes the tool retraction to point R plane at the speed specified by F;

⑧ The spindle stops M05 output and stops rotation;

⑨ The tool rapidly return to the initial plane;

⑩ The common tapping cycle ends.

Note 1: The code is for the flexible tapping and the tapping axis' feed follows the spindle rotation. After the spindle stop signal M05 is valid at the hole bottom, the spindle decelerates to stop rotation, at the moment, Z still feeds along with the spindle rotation till the spindle exactly stops, so, the thread hole bottom position during the actual machining is higher or lower than the actual programmed position, and the concrete error length is determined by the spindle speed and the spindle brake equipment during tapping.

So, in order to safety, before executing G84/G88 tapping, the operator can move the slide to a safety position. G84/G88 is executed without cutting a workpiece (note: here is not the dry run

mode) . The program can be modified according to the actual position of the spindle stop at the hole bottom in G84/G88 machining away from G84/G88 start point's coordinate value. So, remain enough hole depth before G84/G88 machining to execute G84/G88 machining.

Note 2：Before tapping cycle, the operator can specify the spindle rotation direction (i.e., code the spindle rotation CW/CCW in advance) in advance according to the screw tap. When the tool reaches point R to start tapping, at the moment, the CNC does not output the spindle rotation M code. After the tool reaches the hole bottom, the CNC automatically judges the corresponding M code when the spindle rotates CCW. After the tapping ends, the spindle stops rotation. When G84/G88 is still used in the next block, and the tool reaches point R, the CNC outputs again the M code for the spindle rotation to make the spindle rotation, at the moment, the spindle rotation direction is consistent with the specified in advance before tapping cycle.

When the spindle rotation is not specified before tapping, the CNC defaults the spindle rotation CW M03 during tapping. After the fixed cycle is cancelled, the spindle stops rotation. The spindle is started again when the machining should be continuously executed.

Note 3: During the tapping, the tapping axis' move speed is determined by the spindle speed and pitch instead of the cutting feedrate override; the spindle override is influenced by N0.3708#6 during cutting.

Note 4: When the single block or feed hold is executed, the system displays "Pause", and the tapping cycle does not stop till the tapping is completed and the tool returns to the start point.

Note 5: When a reset, emergency stop or drive alarm occurs, the tapping cutting decelerates to stop. In the course, the spindle needs to decelerate to stop but Z exactly stops feed, so, the workpiece and screw tap maybe be damaged. So, G84/G88 should be not forcibly interrupted as possible during machining.

Note 6: N0.5209#0=0, i.e., "in rigid tapping, the drilling axis executes the selection by the plane". The drilling axis is separate X, Z, Y for the separately specified G17, G18, G19 in G84; the drilling axis is separate Y, X, Z for the separately specified G17, G18, G19 in G88.

Note 7: When R plane exceeds the initial plane and the hole bottom plane in the tapping blocks, an alarm occurs.

Program example: in the following figure, thread M10×2

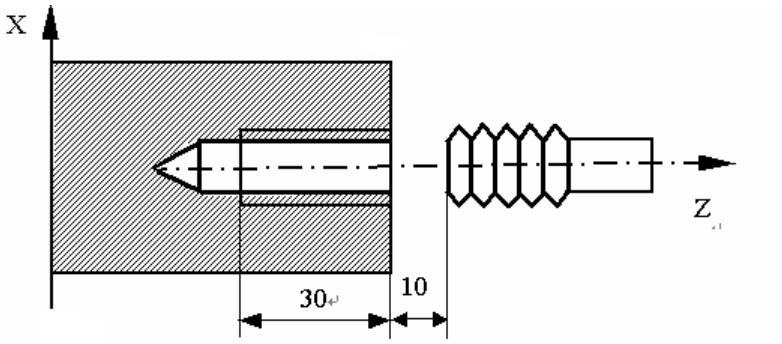


Fig. 2-78

G98；	Feed per minute mode
G0 X0 Z200；	X and Z position to the start point
M3 S800；	The spindle rotates CW and the spindle speed is 800 rev/min. After the block is execute, the spindle starts rotation
G84 Z160 P1000 F1600；	The start point is X0 Z200，the hole position and the start point are the same, the dwell time is 1sec and the thread's lead is 2 according to F value and S value. When the rigid tapping mode is not specified in advance, G84 is the common tapping cycle. After the block is executed, the spindle stops rotation
N G80；	The fixed cycle is cancelled

M30； End of program

2.25 Automatic Chamfering Function

Command function: Automatic chamfering function is defined to automatically insert chamfering block or coring R block between machining blocks.

Blocks where the automatic chamfering can be inserted:

- Between linear interpolation and linear interpolation
- Between linear interpolation and arc interpolation
- Between arc interpolation and linear interpolation
- Between arc interpolation and arc interpolation

Command format: , C_ ；（chamfering）

, R_ ；（corning R）

Command explanation: one chamfering block or corning R block is inserted when the above format is specified at the end of the specified linear interpolation (G01) or arc interpolation (G02, G02) block.

Note: The system can continuously specify more than two chamfering blocks and corning R blocks.

+	Incremental system	Metric input（mm）	Inch input (inch)
,C	ISB system	-99999.999～99999.999 mm	-9999.9999～9999.9999 inch
	ISC system	-9999.9999～9999.9999 mm	-999.99999～999.99999 inch
,R	ISB system	-99999.999～99999.999 mm	-9999.9999～9999.9999 inch
	ISC system	-9999.9999～9999.9999 mm	-999.99999～999.99999 inch

Chamfering： The numerical value following C specifies the distance from chamfering starting point to end point of the imaginary cornering intersection which is defined to the imaginary existing cornering when the chamfering is not executed.

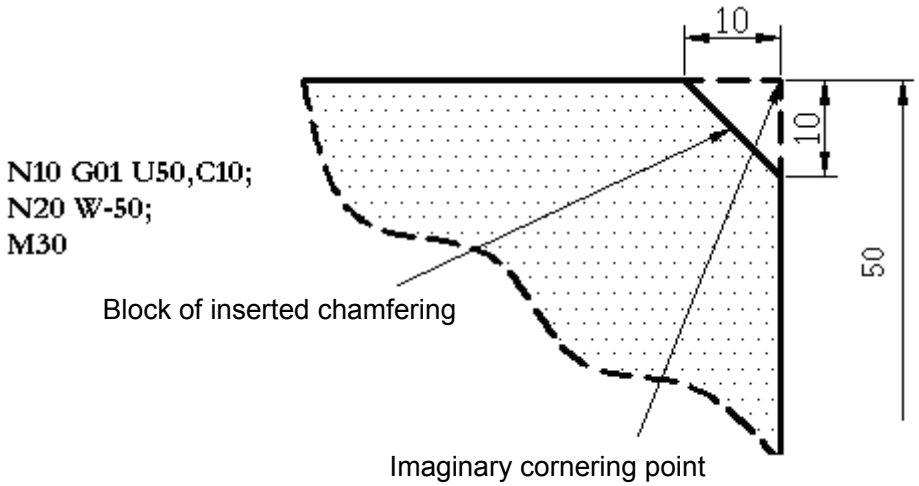


Fig. 2-79

Corning R: The numerical value following R specifies corning R radius.

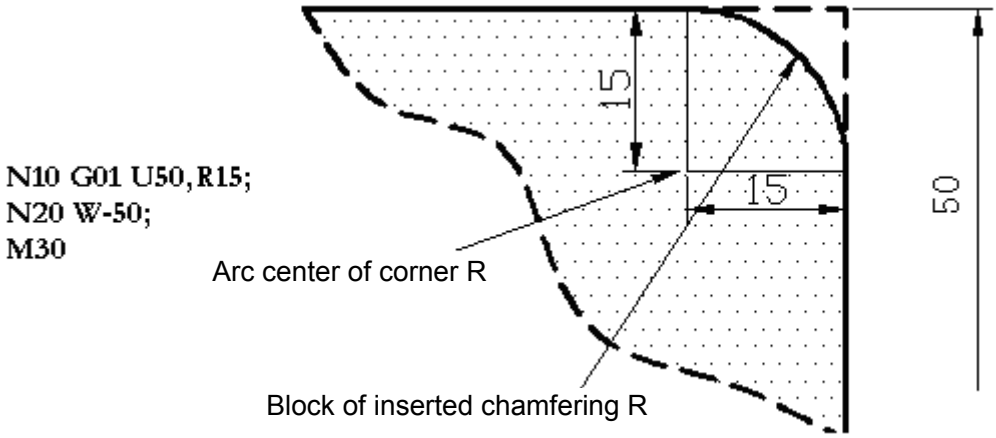


Fig. 2-80

Note 1: Even if the chamfering (, C) or corning R(,R) is specified in other blocks besides G01 and G02/G03 (except for G32, G34), it is ignored.

Note 2: The block following chamfering or corning R for the chamfering or corning operation must be the one of G01 or G02/G03. The alarm "no movement after chamfering/corning R" occurs when other codes are specified.

But, only one G04 (dwell) block can be inserted between these blocks. The system pauses after the inserted chamfering/corning R block is executed.

Note 3: When the system exceeds the previous interpolation movement range caused by the inserted chamfering or corning R block, the alarm "executed movement being excessive small in the block following chamfering/corning R" occurs.

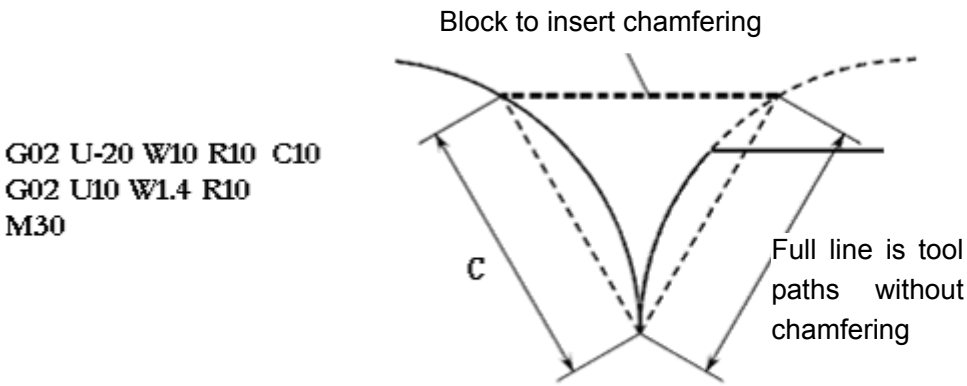


Fig. 2-81

Note 4: The chamfering or corning R block can be inserted into the movement codes included in the same plane.

When the plane selection (G17, G18, G19) in the next block after the chamfering or corning R is specified, the alarm occurs "the plane selection code is specified after chamfering or corning R".

Note 5: When two linear interpolation operations are executed and their angle difference is within ±1, the

movement of chamfering/coring R block is 0. When linear interpolation and circular interpolation operations are executed and angle difference of their tangent at the intersection point is within ±1, the movement of corning R block is 0. When two circular interpolation operations are executed and the angle difference of their circular tangent is within ±1, the movement of corning R block is 0.

Note 6: When the chamfering or corning R block is specified in a single block, the operation runs until it reaches the end point of new chamfering/corning R block, the machine stops in feed hold mode at the end point.

Note 7: The following G codes cannot be used with the chamfering/corning R code in the same block, as well as the blocks of chamfering/corning R of the defined continuous graph.
G codes in Group 00 (except for G04)

Note 8: When ",C" or ",R" is executed in the thread cutting block, the alarm occurs "cannot code the chamfering or corning R in the current block".

Note 9: The last is valid when the many ",C" and ",R" are specified in the same block.

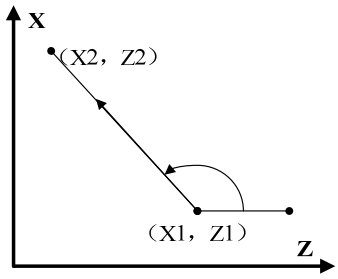
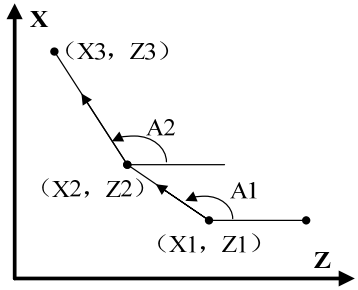
2.26 Function of Directly Inputting Graphic Dimension

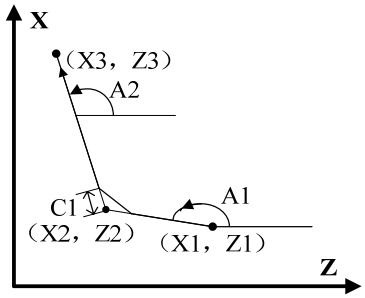
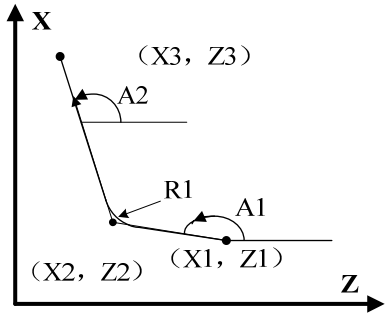
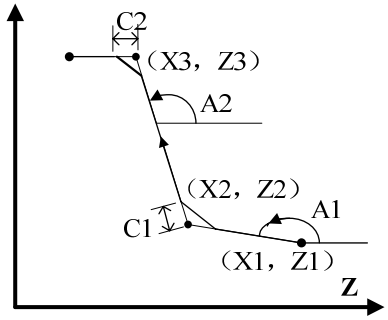
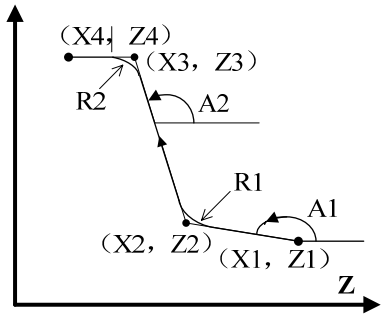
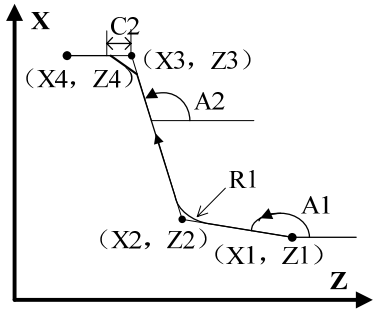
Command function: the function of directly inputting graphic dimension can make the user directly use the linear angle, chamfering value and corning R value in the machining drawing to program.

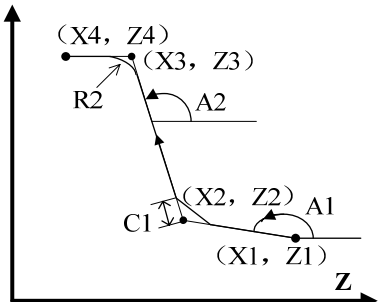
Command format: can specify the plane in G17 plane（XY plane）, G18 plane（XZ plane）, G19 plane（ZY plane）. Taking example of G18 plane（XZ plane），the format changes when G17/G19 plane code is used：

G17 plane：“Z”→“X”，“X”→“Y”

G19 plane：“Z”→“Y”，“X”→“Z”

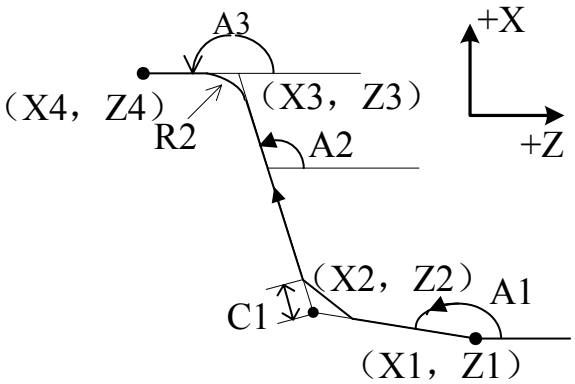
	Command format	Command path
1	X2_ (Z2_) ， A_；	
2	， A1； X3_ Z3_ ， A2_；	

	Command format	Command path
3	X2_ Z2_ , C1_; X3_ Z3_; Or , A1_ , C1_; X3_ Z3_ , A2;	
4	X2_ Z2_ , R1_; X3_ Z3_; Or , A1_ , R1_; X3_ Z3_ , A2;	
5	X2_ Z2_ , C1_; X3_ Z3_ , C2_; X4_ Z4_; Or , A1_ , C1_; X3_ Z3_ , A2, C2_; X4_ Z4_;	
6	X2_ Z2_ , R1_; X3_ Z3_ , R2_; X4_ Z4_; Or , A1_ , R1_; X3_ Z3_ , A2, R2_; X4_ Z4_;	
7	X2_ Z2_ , R1_; X3_ Z3_ , C2_; X4_ Z4_; Or , A1_ , R1_; X3_ Z3_ , A2, C2_; X4_ Z4_;	

	Command format	Command path
8	X2_ Z2_ , C1_; X3_ Z3_ , R2_; X4_ Z4_; Or , A1_ , C1_; X3_ Z3_ , A2，R2_; X4_ Z4_;	

Command explanations:

X2_ Z2_ , C1_;
X3_ Z3_ , R2_;
X4_ Z4_;
Or
, A1_ , C1_;
X3_ Z3_ , A2，R2_;
X4_ Z4_;



A line is specified by specifying one or two of X, Z, A. When one is specified, the linear must be defined by a code in the next block.

When the angle, chamfer or corning R value of a line, it is specified by a code with the following comma（,）.

,A_
,C_
,R_

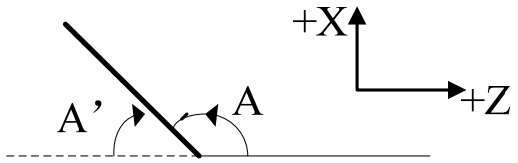
Note: when end coordinates and linear angle of two axes are specified, the supplementary angle function is disabled. The codes in the above figure are programmed as follows:

X1_ Z1_;
X2_ Z2_ , A1_ C1_ ; (at this time, the supplementary angle A1 is invalid because of the end coordinate X2 Z2 has confirmed the path instead of supplementary angle to count).

- **A supplementary angle specifying a angle**

When DDP(No.3405#5) is set to “1”，the supplementary angle can specify the angle.

Suppose that the supplementary angle is A’，the actual code angle is A，so, the following relationship is true.



A = 180 – A'

Note 1：The code for directly inputting graphic dimension is valid in only Auto mode, and an alarm occurs in MDI, DNC mode.

Note 2：The following G codes cannot be used in the blocks which are the same those of directly inputting graphic dimension, also in the blocks which define continuous graph's directly inputting graphic dimension, as well as in the blocks which the directly inputting graphic dimension is executed in their mode.

G codes in group 00（except for G04 ）

G codes except for G00, G01, G32 in group 01

G codes in group 10（fixed cycle for drilling ）

G codes in group 16（plane selection ）

G22, G23

Note 3：Cannot insert the chamfering C or corning R code in thread cutting blocks.

Note 4：In single block run, when the system uses the code for directly inputting continuous graphic dimension, and the next block determines the previous block's end point, the machine does not stop at the end point of the previous block in single block stop mode rather than in feed hold mode.

Note 5：In the following programs, angle tolerance of counting intersection point is ±1 degree.（the counted movement value is too big at intersection point.）

***X_,A_;*（an alarm occurs when the angle code A is specified to 0 degree or within *180±1* degree.）**

Z_,A_;(an alarm occurs when the angle code A is specified to 90 degree or within *270±1* degree.）

Note 6：When the intersection point is counted and angle difference between two lines is within *±1* degree, an alarm occurs.

Note 7: The chamfering or corning R is ignored when angle difference of two lines is within ±1.

Note 8：The system must specify a coordinate code value (absolute code value) and an angle code value in a block which follows the block only having specified an angle code.

（Example）

N1 X_ ,A_ ,R_;

N2 ,A_;

N3 X_ Z_ ,A_;

(Besides the coordinate code value, must specify an angle code in N3, otherwise, an alarm occurs. Besides, the coordinate code value must not be an absolute code, otherwise, an alarm occurs.）

Note 9：In tool nose radius compensation, blocks of the angle code for directly inputting graphic dimension have no movement blocks.

Note 10：In continuous graphic dimension directly input codes, when there are 2 or more movement blocks between codes, an alarm occurs.

Note 11：When CCR(No.3405#4) is set to "1" , the block address A of G76（multiple thread cutting cycle）is the tool nose angle code. Besides, when A or C is an axis name, A and C cannot be used in the angle codes and chamfering codes of graphic dimension directly input. Please use ,A_ and ,C_

（CCR(No.3405#4)=“0”）。

Note 12：In compound fixed cycle, although the program for graphic dimension directly input is used in the blocks between the serial number specified by P or Q, the last block specified by Q cannot be in the code for the graphic dimension directly input.

2.27 Macro Code

GSK988TA/TB provides the macro code which is similar to the high language, and can realize the variable assignment, and subtract operation, logic decision and conditional jump by user macro code, contributed to compiling part program for special workpiece, reduce the fussy counting and simplify the user program.

2.27.1 Variable

（1）variable use

The variable can specify the address value in the program. The variable value is assigned by the program code or is set directly by the keyboard. One program can use many variables which can be distinguished by their variable number.

- **Variable expression**

  Use “#”+variable number to express;

  Format：# i（i=200，202，203，……）;

  Example：#205，#209，#225.

  Besides, the expression can be used to specify the variable number. At the moment, the expression must be in the brackets.

  Example：#[#20+#30/4]

- **Variable reference**

1. Use variable to permute the number following address.

   Format: ＜address＞+“# i” or ＜address＞+“—# i” means to take the variable value or the negative value of value of the variable as the address value.

   Example: F#203…#203=15: it is the same those of F15 functions;

   Z-#210…#210=250: it is the same those of Z-250 functions;

   G#230…#230=3: it is the same those of G3 functions.

   When the variable value is used in program, the decimal point can be omitted. Example: #1=123: the actual value of #1 is 123.000.

   When the variable value followed the axis code address has the decimal point, the data less than the least setting unit executes the rounding. For example: #1=1.23456; the axis least setting unit is 0.001, the tool to execute G00 X#1 positions to 1.235 position.

2. Use variable to permute variable number.

   Format: “#”+[variable number]

   Example: 5 uses #30 to execute the permutation in #5, is written to #[#30].

3. Refer the undefined variable.

   When the variable is not defined, it becomes the “Null” variable. When the variable #0 is Null, it is only read instead of being written.

When the system refers to the undefined variable, it ignores the variable and the word.

Example: when the variable #10 value is 0, the variable #!1 value is Null and the system executes G00 X#10 Y#11, the execution result is G00 X0，Y#11 to be ignored.

In course of operation, besides using null variable assignment, null variable value is the same that of 0 in other cases.

When #2=<Null> , #1=#2, #1=<Null>;
#1=#2*3, #1=0;
#1=#2+#2, #1=0;

Beside using the Null to assign, the variable value is 0 in other conditions.

When #2=< Null >, #1=#2, #1=<Null>;
#1=#2·3, #1=0;
#1=#2+#2, #1=0;

<Null> in conditional expression is different with 0.

When #2=<Null>, #2 EQ #0，#2 NE 0, the condition is false.

When #2=0, #2 EQ #0，#2 NE 0, the condition is false.

（**2**）Variable Type

The variable is divided into the different variable types according to the variable number, their use and prosperity are different below:

Variable range	Variable type	Function
#0	Null variable	The variable is null and is not assigned.
#1～#33	Local variable	The local variable is used to transmit parameters (when the system uses G65, G66 to call subprograms every time, it initializes the local variable value by parameters), and store data (for example: store operation result) in the macro program. When the system is turned off, the local variable is initialized to be null. When the macro program is called, the argument assigns to the local.
#100～#199 #500～#999	Share variable	The share variable has the same meaning in the different macro program. When the system is turned off, the variable #100~#199 is initialized to be null, #500~#999 is saved and is not lost.
#1000--	System variable	The system variable is used to read all types of data when CNC runs.

（3）Variable range

The input range of the local variable and common variable is -99999999~99999999 which integer part and decimal part are up to 8-digit number. When the assignment exceeds the valid range, an alarm occurs. When the operation result exceeds -99999999~99999999 and is stored into the variable, the system window displays “***” which means overflow. When the variable is used in code, the relevant operation result is displayed to “***”; when the digit exceeds 8-digit, its former 8 digits are valid, and its ninth digit is rounded; the middle result in the macro variable count can be more than the valid input digital.

Note 1: The variable cannot be referred to address O and N. The system cannot use O#200，N#220 to execute the programming;

Note 2: When the variable exceeds the max. code value defined by the address, it cannot be used; for example: #230 = 120: M#230 exceeds the max. code value;

Note 3: The system cannot identify –0 and + 0. # 4 = - 0: X # 4 is taken as X 0;

Note 4: When the variable is used to the address data, the other except for the valid digit is rounded.

Note 5: The number followed by the address can use <Formular> to replace. The system takes "Word address [<Formular>]" or word address-[<Formular>]" as a program, and take <Formular> value or its negative value as the code value of the address.

Note 6: The decimal point which defines the variable in a program can be omitted. For example, #1=123 is defined, the actual value of #1 is 123.000;

Note 7: The negative sign of variable value which changes the reference should be placed in the front of #, such as G00X-#1;

Note 8:The variable #1～#33, #100～#199 are cleared out after they reset, which are set by NO.6001Bit7（CLV）and Bit6（CCV）, and which cannot be executed in MDI mode;

Note 9: When the variable value overflows, the code address referring to the variable is ignored.

Note 10: NO.6000 Bit5（SBM）sets whether the single block stop is valid in user macro program.

Note 11: The number in expression (including brackets) can be omitted. For example, X[10] actual value is X10.000.

2.27.2 System Variable

The system variable is used to read and write NC internal data. For example, some system variable only read the tool offset value and current position data. The system variable is the base of the automatic control and general machining program development.

（1）Interface signal

The interface signal can program the exchange message between the machine controller and user macro programs, i.e. it completes the exchange with PLC by G, F signals and the interfaces with I/O are defined by PLC.

The input signal can be only read, and the output signal can be read and written.

System variable of interface signal		
Variable number	Function	Corresponding G, F signals
#1000--#1031	Read the 32-bit signal according to its bit from PLC to user macro program	Corresponding to G54.0～G57.7 signal state
#1032	Read 32-bit signal one time.	Corresponding to G54～G57 signal state

#1100--#1131	Write 32-bit signal from macro programs to PLC according its bit (its corresponding signal is 0 or 1 based on its macro variable value which is rounded off.	Corresponding to F54.0 ～ F57.7 signal state
#1132	Write 32-bit signal to PLC one time. It is specified in the range: -99999999～99999999	Corresponding to G54～G57 signal state

（2）Tool compensation value

The system variable can read/write the tool compensation value. The system variable of the tool compensation storage area is 2001～2999. The variable numbers divided exactly by 100 in the above range are illegal. The concrete ranges are referred to the following table.

Compensation number	X		Z		Radius compensation value R		Tool nose T
	offset	wear	offset	wear	offset	wear	
1	2701	2001	2801	2101	2901	2201	2301
…	…	…	…		…	…	…
99	2799	2099	2899	2199	2999	2299	2399

Compensation number	Y	
	offset	offset
1	2401	2501
…	…	…
99	2499	2599

Note: Range of #2301-#2399: 0-9, and is rounded when it is with decimal point.

（3）Marco program alarm

Alarms and alarm messages specified by the user can exist in programs. The variable can only be written instead of being read.

Variable	Function
#3000	When the system executes the assignment statement of #3000=XXX, it stops the run and alarms. The alarm message only displays 31 characters (15 Chinese characters), and the system only displays the first 31 characters when there are more than it. The value of the alarm number being #3000 adds 3000, the alarm range is 3000 to 3200. When #3000 value is less than 0, the alarm number is 3000, when #3000 value is more than 200, the alarm number is 3200.

Example:
#3000=6; the tool has not found

When the system executes the block, it stops and alarms and the alarm number is 3006. The alarm message is “TOOL NOT FOUND”, The system maybe alarm in advance because of the buffer exists.

The alarm message can use the small brackets. For example, #3000=6(TOOL NOT FOUND). When the small brackets and the semicolon are in the block, the latter specified message is valid, such as #3000=6(TOOL NOT FOUND); TOOL NOT FOUND，the displayed message is “TOOL NOT FOUND”.

（4）Stop message

The program execution is interrupted and the system displays one message. i.e. the single stops after the system executes the block, and the system displays only one prompt. The variable is only be written instead of being read.

Variable	Function
#3006	When the system executes the assignment statement of #3006=1, it stops the run and displays only one prompt message. The alarm message only displays 26 characters (13 Chinese characters), and the system only displays the first 26 characters when there are more than it. The value of the alarm number being #3006 adds 3200, the prompt number range is 3201 to 3500. When #3006 value is less than 1, the alarm number is 3201, when #3006 value is more than 300, the alarm number is 3500.

For example:

#3006=3; wait for run

When the system executes the block, it stops and displays one prompt and the prompt number is 3203. The prompt message is “WAITING FOR RUN”. The format of the prompt message is the same that of description in the macro program alarm.

（5）Machine workpiece quantity

The required workpiece quantity (the target quantity) and machined workpiece quantity (completed quantity) are read and written

Required workpiece quantity and machined workpiece quantity	
Variable	Function
#3901	Machined workpiece quantity(completed quantity)
#3902	Required workpiece quantity(target quantity)

When #3901 value is changed, the workpiece quantity displayed in POSITION window also changes.

When #3902 value is changed, No.6713 value also changes.

（6）Modal message

The previous modal message which is being processed can be read.

Variable number	Function	
#4001	G00, G01, G02, G03, G32, G34, G90, G92, G94	Group 1
#4002	G96, G97	Group 2

#4003		Group 3
#4004		Group 4
#4005	G98, G99	Group 5
#4006	G20, G21	Group 6
#4007	G40, G41, G42	Group 7
#4008	G25, G26	Group 8
#4009	G22, G23	Group 9
#4010	G80, G84,G88	Group 10
#4011		Group 11
#4012	G66, G67	Group 12
#4013		Group 13
#4014	G54, G55, G56, G57, G58, G59	Group 14
#4015		Group 15
#4016	G17, G18, G19	Group 16
…		…
#4022		Group 21
#4109	F code	Group 22
#4113	M code	
#4119	S code	
#4120	T code	

Example:

When the system executes #1=#4016, #1 value is 17, 18 or 19.

An alarm occurs when the reading/writing modal value is G code which cannot be used by the system.

（7）Current position

The position message is only read instead of being written.

Variable number	Position signal	Coordinate system	Tool compensation value
#5001--#5005	End point of block(absolute coordinate)	Workpiece coordinate system	Not including
#5021--#5025	Current position(machine coordinate)	Machine coordinate system	including
#5041--#5045	Current position(relative coordinate)	Workpiece coordinate system	including
#5061--#5065	Skip signal position	Workpiece coordinate system	including
#5081--#5085	Tool length compensation value		

The read is the position value after the last block execution.
The unit digit from 1 to 5 of variable number corresponds the No. n axis.

（8）Compensation value of workpiece coordinate system

The workpiece zero offset value can be read and written.

Variable number	Function
#5201--#5205	External zero offset value
#5221--#5225	G54 workpiece zero offset value
#5241--#5245	G55 workpiece zero offset value
#5261--#5265	G56 workpiece zero offset value
#5281--#5285	G57 workpiece zero offset value
#5301--#5305	G58 workpiece zero offset value
#5321--#5325	G59 workpiece zero offset value

The units digit from 1 to 5 of variable number corresponds the No. n axis.

（9）Note

The system variable is the state value of the system, and is buffered in advance when multi cycles are executed, so, the attained system variable is the value before the multi cycle code instead of the current value to avoid using the system variable in the cycle body of the multiple cycles.

2.27.3 Operation and Jump Code

(1) Operation code

Variables can execute all kinds of operations, and their operation command format is as follows.

#i=＜Expression＞

The right <expression> of an operation code is a compose of constant, a variable, function and operator.

GSK988TA/TB defines the following operations and logic codes:

Function	Format	Use
assignment	#i=#j;	Assignment statement assigns #j value to #i； #i is Null when #j is Null;
addition	#i=#j+#k;	Addition. When #j value is Null, it it taken as 0.0 value, and the following functions are the same that of it;
Subtraction	#i=#j-#k;	Execute subtraction operation;
Multiplication	#i=#j*#k;	Execute division operation;
Division	#i=#j/#k;	Execute addition;
Sine	#i=SIN[#j];	Execute sine operation; Angle unit is degree;
Arc sine	#i=ASIN[#j];	Execute arc sine operation; #j value is -1～1;
cosine	#i=COS[#j];	Execute cosine operation ; Angle unit is degree

Arc cosine	#i=ACOS[#j];	Execute arc cosine; #j value is -1～1 Function range: 0°～180°.
Tangent	#i=TAN[#j];	Execute tangent operation ; Angle unit is degree #j value cannot be 90, 270
Arc tangent	#i=ATAN[#j]/[#k];	Specify the lengths of two sides, execute the arc tangent, #j is opposite with "/" to partition;
Square root	#i=SQRT[#j];	Execute square root operation; #j cannot be less than zero
Absolute value	#i=ABS[#j];	Execute absolute value operation;
Rounding	#i=ROUND[#j];	Execute rounding operation; In macro program, execute the rounding of one-digit of No., in NC statement, execute the rounding of the next digit of the least increment
FUP	#i= FUP [#j];	Floating UP integer ; In puls quantity, #i is more than or equal to #j，in the negative,#i is less than or equal to #j
FIX	#i= FIX [#j];	Floating FIX integer; In puls quantity, #i is less than or equal to #j，in the negative,#i is more than or equal to #j
Natural logarithm	#i=LN[#j];	Execute natural logarithm ; The system alarms when #j is zero or less than zero
Exponential function	#i=EXP[#j];	Execute #j exponent ; #j value cannot be more than 80
OR	#i=#j OR #k;	Execute the binary logic operation of input data #j, #k cannot be less than zero When there are the decimal points in #j, #k, the decimal parts are rounded
XOR	#i=#j XOR #k;	
AND	#i=#j AND #k;	
BCD to BIN	#i=BIN[#j];	Converse the decimal data into the binary; The system alarms for the data which cannot the converse
BIN to BCD	#i=BCD[#j];	Converse the binary into the decimal

Command explanation:

（1）operation sequence:

Priority	Operator and function
5	" [" , "]"
4	"#"
3	"SIN", "SI", "ASIN", "AS", "COS", "CO", "ACOS", "AC", "TAN", "TA", "ATAN", "AT", "SQRT", "SQ", "ABS", "AB", "ROUND", "RO", "FIX", "FI", "FUP", "FU", "LN", "EXP", "EX", "BIN", "BI", "BCD", "BC",

2	"AND", "AN", "*", "/",
1	"OR", "XOR", "XO","+", "-",

（2）EXP function input value cannot be more than 80, otherwise, an alarm occurs;

（3）"/" character in <expression>(in the right of assignment"=" or in the bracket []) is taken as the division operator instead of optional block skip code;

(4) A bracket "[]" is used to 5 levels, including the bracket used in the function. An alarm occurs when it exceeds 5 levels.

（5）The angle units of the triangle function SIN, COS, ASIN, ACOS, TAN and ATAN are degrees, for example: 90°30′ is 90.5 degree;

（6）#i=ASIN[#j] value range:

When NO.6004 No. 0-digit NAT is set to 0: 90°～270°

When NO.6004 No. 0-digit NAT is set to 1: -90°～90°

When #j exceeds between -1 and 1, the system alarms and #j can be a constant.

（7）#i=ACOS[#j] range: 0°～180°。

When #j exceeds between -1 and 1, the system alarms and #j can be a constant.

（8）In #i= ATAN[#j]/[#k], ATAN #j and #k are the weight length of two right-angle sides as follows:

When NO.6004 No. 0-digit NAT is set to 0: 0°～360°

Example: when #1=ATAN[-1]/[-1] is specified, #1=225°.

When NO.6004 No. 0-digit NAT is set to 1: -180°～180°

Example: when #1=ATAN[-1]/[-1] is specified,#1=-135°

#j，#K can be the constant.

In division or TAN[90], the division is specified to 0, P/S alarms;

（9）The function ROUND is used to NC code or macro statement, which rounds the data with the decimal point. It is used to NC statement, which rounds according to the least setting unit; when it is used to the macro statement, which rounds No. 1-digit decimal point;

In executing #2=ROUND[#3], when #3=1.2345, the variable #2 value is 1.

In ISB increment metric input, #2=1.2345, #3=2.5456:

G00 X#2； the tool moves to 1.234mm

G00 X#3； the tool moves to 2.545mm

（10）For FUP, FIX, when the absolute value of the integer after execution is more than that of the original, it is FUP; when it is less than that, it is FIX.

When #2=1.2，#3=-1.2

In executing #4=FUP[#2], 2.0 is assigned to #4

In executing #4=FIX[#2], 1.0 is assigned to #4

In executing #4=FUP[#3], -2.0 is assigned to #4

In executing #4=FUP[#3], -1.0 is assigned to #4

（11）Logic operation OR, XOR, AND firstly are conversed the decimal into the binary, and are executed in the binary by one-digit to one digit.

Range: 0～99999999，when it has the decimal point, it is ignored.

Example:

#101=10 （the binary is: 00001010）

#102=12 （the binary is: 00001100）

#103=#101 OR #102 （or the operation result is : 00001110）

The window display result of macro variable is #101=10.000000 #102=12.000000 #103=14.000000

（12）The function BIN converses the decimal into the binary which is displayed in decimal system.

The function converses the binary number displayed in decimal system into 8421 format BCD. The system cannot display and alarms when some digit in BCD code after conversion exceeds 9.

Example 1：

#101=37 （BCD 37 corresponds to the binary : 00110111）

#102=BCD[#101]

Macro variable window display #102=55.000000

(2) Transfer and cycle codes

The transfer and the repetition codes can change the control flow, and there are three kind of transfer and repetition operation: the unconditional transfer GOTO, the conditional transfer IF…GOTO, IF…THEN and WHILE DO repetition.

Command format:

GOTO n;

Command function:

Skip to the line number n without condition;

Command format:

IF <Logical expression> THEN <expression>;

Command function:

When the logical expression is valid, the system executes one following THEN, otherwise, it executes the next block.

Command format:

IF < Logical expression > GOTOn;

Command function:

When the logical expression is valid, the system skips the block with the line number n to execute, otherwise, it executes the next block;

Command format:

WHILE < Logical expression > DOn;

……;

ENDn

Command function:

When the logical expression is valid, the system executes the block between Do and END, otherwise, its execute the block following END. The numerical value n following DO and END is used to specify the execute range label of the specified program, n value is 1, 2, 3. The system alarms when n is not 1, 2, 3.

IF, WHILE logical operation character rules are as follows:

Operator	substitute	character definition
EQ	==	（=）
NE	<>	（≠）

GT	>	（>）
GE	>=	（≥）
LT	<	（<）
LE	<=	（≤）

Note 1: When the system transfers to the block with the serial number n and specifies the another exceeding the serial number range between 1 and 99999, P/S alarms, and the expression can specify the serial number;

Note 2: The conditional expression must include the operator which is inserted in the middle of two variables or the variable and the constant and is closed by the bracket[]. The expression can be replaced by the variable;

Note 3: The number following D0 and the one following END specify the execution range label of the specified program, and the label value is 1, 2, 3. The system alarms when n is not 1, 2, 3;

Note 4: The label (1-3) in the repetition DO—END can be used many times, but P/S alarms when there is the cross repetition (superposition in DO range);

Note 5: When the system specifies D0 instead of WHILE statement, it creates the limitless repetition between DO and END;

Note 6: In using EQ, NE logical operation expression, <Null> and zero have the different result. <Null> is taken as the zero in +, -, * conditional expression;

Note 7: The macro program statement cannot be used with NC statement together, and the macro program statement definition is as follows:

Block including arithmetic or logical operation(=);

Block including the control statement (such as TOTO, DO, END);

Block including macro program call code (such as G65, G66, G67 or other G codes, M code call macro program) ;

Any blocks except for macro program statements are NC statements;

Note 8: Any blocks except for macro program states are NC statements.

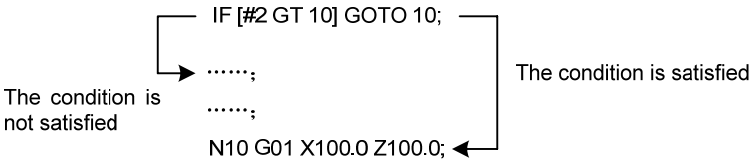
Note 9: The system can use the substitution character which is easily understood to replace the operator. '>', '<' can be edit in PC instead of on MDI keyboard and are uploaded into the system;

Note 10: When macro statement needs a line number, the line number must be compiled in the front the statement;

Note 11: In MDI mode, the system cannot execute the skip statement, otherwise, it alarms.

Example:

（1）GOTO example

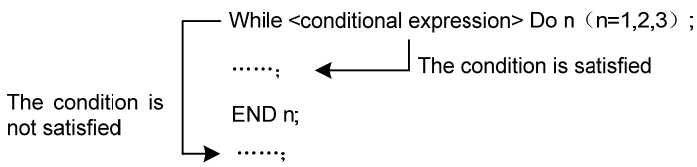


（2）IF <Logical expression> THEN <expression> example

IF[#2 EQ #3] THEN #4=0；

When #2 value is same that of #3, #4 value is 0.

（3）WHILE <Logical expression> DOn;…；ENDn example



2.27.4 Macro Program Statement and NC Statement

The following blocks are macro program statements:
Including arithmetic or logical operation (=);
Including control statement(such as GOTO, DO, END);
Including macro program call code (G65, G66, G67).
Any NC blocks except for macro program statement are NC statements.
In Single Block mode, when No.6000 Bit5 (SBM) is set to 0, the system directly skips the macro program statement and the machine does not stop, but it is set to 1, the system stops run and enters the stop state.
One block cannot have the macro program statement and NC statement simultaneously.

2.27.5 Macro Program Call

（1）Non-modal call of macro program G65

Command format: G65 P __ L __ 〈argument list 〉;

Command function: The system calls macro program L times specified by P and transfers the argument to the called macro program.

Command explanations: P: specify the macro program to be called;
L: times of calling the macro program, and its default is 1 and its range is 1～9999;
Argument list: data transferred to macro programs.

Argument specification:

Two types of argument specification are available. Argument specification I uses letters other than G, L, O, N and P once each. Argument specification II uses A, B and C once each and also uses Ii, Ji and Ki. i is 1~10, and the types of argument specification are determined automatically according to the letters used.

Argument specification I

Address	Variable No.	Address	Variable No.	Address	Variable No.
A	#1	I	#4	T	#20
B	#2	J	#5	U	#21
C	#3	K	#6	V	#22
D	#7	M	#13	W	#23
E	#8	Q	#17	X	#24
F	#9	R	#18	Y	#25
H	#11	S	#19	Z	#26

Addresses G, L, N, O and P cannot be used in arguments;
Addresses that need not be specified can be omitted and local variables corresponding to an omitted address are set to null;

Addresses do not need to be specified alphabetically. They conform to word address format, but I, J , K are specified according to the letter order;

Example: B_A_D_…J_K_ Correct

B_A_D_…K_J_ Incorrect

Argument specification II uses A, B and C once each and uses I, J, and K up to ten times. Argument specification II is used to pass values such as three-dimensional coordinates as arguments.

Argument specification II

Address	Argument No.	Address	Argument No.	Address	Argument No.
A	#1	K3	#12	J7	#23
B	#2	I4	#13	K7	#24
C	#3	J4	#14	I8	#25
I1	#4	K4	#15	J8	#26
JI	#5	I5	#16	K8	#27
K1	#6	J5	#17	I9	#28
I2	#7	K5	#18	J9	#29
J2	#8	I6	#19	K9	#30
K2	#9	J6	#20	I10	#31
I3	#10	K6	#21	J10	#32
J3	#11	I7	#22	K10	#33

Note 1: G65 must be specified before any argument;

Note 2: After G65, specify at address P and L. when P or L is repeated and No.3403 Bit6 (AD2) is set 0, the specification later takes precedence, otherwise, the system alarms;

Note 3: Subscripts of I, J, K in the argument specification II for indicating the order of argument specification are not written in the actual program;

Note 4: The CNC internally identifies argument specification I and argument specification II. If a mixture of argument specification I and argument specification II is specified, the type of argument specification specified later takes precedence;

Note 5: Calls can be nested to a depth of four levels including simple calls G65 and modal calls G66. This does not include subprogram call M98.

Note 6: Whether the units used for argument without a decimal point correspond to the least input increment of each address is related to the parameter DPI (No.3401#0);

Note 7: G65, G66 cannot be in the same block with NC code, otherwise, the system alarms;

Note 8: In macro program nesting call, the local variables from level 0 to 4 are provided for nesting. When the level of the main program is 0, each time a macro is call, the local variable level is incremented by one. The values of the local variables at the previous level are saved in the CNC. When M99 is executed in a macro program, control returns to the calling program. At that time, the values of the local variables saved when the macro was called are restored.

Note 9: The line number of the code line of the macro statement must be home, otherwise, the system alarms.

Macro program nesting example

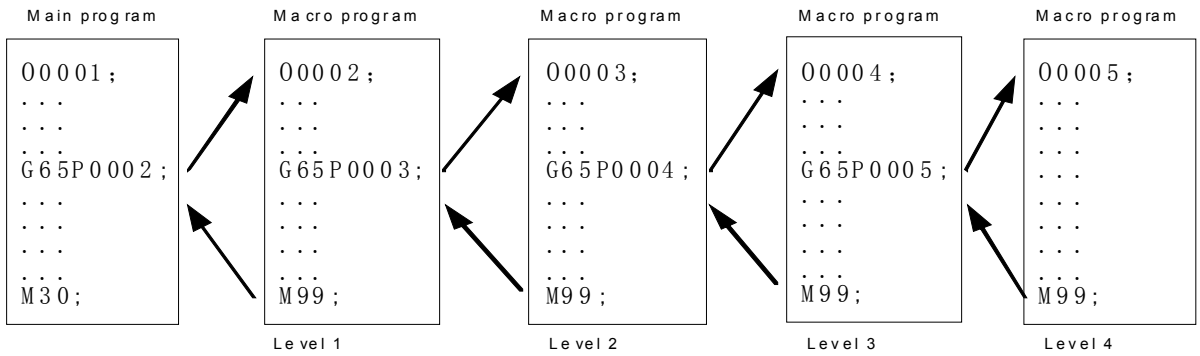


Fig.2-82 Nesting macro program

（2）Modal call of macro program G66, G67

Command function: set the modal message of the specified macro program L times for calling P, send the argument to the called macro program.

Command format：G66 P __ L __ 〈argument list 〉；

……;

G67;

Command explanation:

G66：modal macro program call needs one line to be specified;

G67：call macro program call mode;

P：specify many called macro programs;

L：times for calling the macro program. It is default to 1, its range is 1—9999;

Argument list: data sending to macro program is referred to the explanations of G65.

Note 1: In G66, call a movement code in the same CNC file or M98, G65 or G66 cannot be executed in non-movement blocks;

Note 2: G66 is specified before P_, L_ and argument, and the use methods of P, L, the argument are the same those of G65;

Note 3: G66 can't call a macro program in G66 blocks but it can execute the call only when its mode is called; without movement codes, it cannot call a macro program in miscellaneous function or dry run mode;

Note 4: The local variable (argument) is specified only in G66 block, and the system does not set it again when each modal call is executed;

Note 5: Cannot specify the macro call code in MDI mode;

Note 6: When the reset is executed by setting the parameter, whether the common variables of the local variables from #1 to #33 and from #100 to #149 are cleared to the Null value.

Note 7: The system clears the call state of all user macro programs and subprograms and DO state, and returns to the main program;

Note 8: In executing the macro program statement, when the feed pause is valid, the machine stops after the macro statement is executed, and the machine also stops when the system resets or alarms.

Application example:

（1） G65 example

Program: O0001
N001 G0 X40.0 Z5.0；
…；
…；
N006 G65 P9000 L3 B20.0 C10.0；
…；
…；
N009 M30；

Macro program: O9000
N001 #4=#3-#2；
N002 IF[#4 GT 40] GOTO 9
…；
N006 G01 X100.0 Z100.0；
…；
…；
N009 M99；

（2）G66, G67 example
Program: O0002
G00 X100 Z50；
G66 P0100 L2 A2 B20 C20 I30 J20 K20；execute the block instead of call, writing local variables, but only change the mode
G01 X80 Z50；after executing the block, call No.P0100 program two times(refresh the local variable according to the argument)
G0 U0 W0; have no subprograms to be called
G01 U1; after executing the block, call No.P0100 program two times(refresh the local variable according to the argument)
G67； G66 mode call is cancelled
G01 X20 Z50； after executing the block, the system does not call No.P0100 program
M30；

2.28 Slant Axis Control

The slant axis control function is defined that the slant axis relative to a quadrature axis is installed with exceeding 90°, which makes each axis' movement amount be controlled according to a slant angle.

Distribute any axes to one group of slant axis and quadrature axis by a parameter.

The actual movement is controlled according to a slant angle, but, a program is compiled according to a vertical intersection between a slant axis and quadrature axis. Here, the used coordinate system is called a program coordinate system. (the following contents sometimes changes the program coordinate system into a cartesian coordinate system, but the actual movement coordinate system is called a slant coordinate system or a machine coordinate system.)

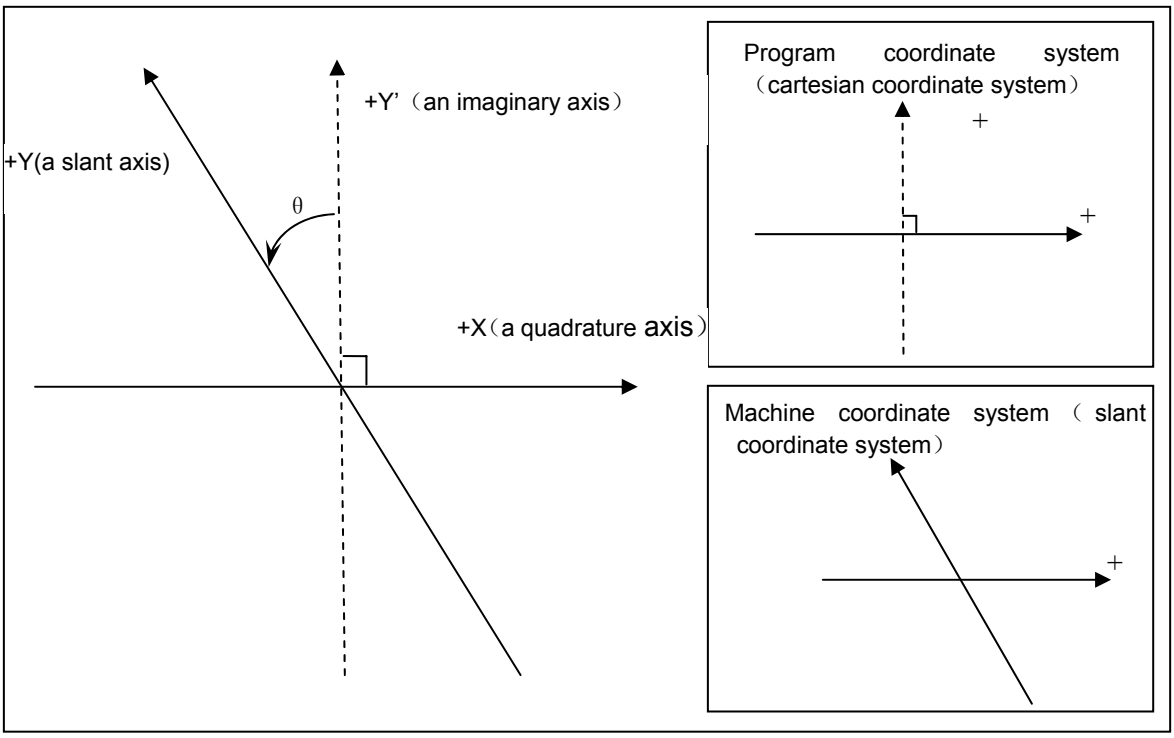


Fig. 2-83

● Each axis' movement formular

When the slant axis' movement amount is Ya，and the quadrature axis' movement is Xa, the system realizes the control by the following expressions.

The slant axis' calculation formular:

$$Ya = \frac{Yp}{\cos\theta}$$

the quadrature axis' calculation formular: $Xa = Xp - C \times Yp \times \tan\theta$

Note: The coefficient C becomes 2 when the quadrature axis is specified with a diameter, and becomes 1 when the quadrature axis is specified with a radius.

Relationship between a slant axis and a quadrature axis is shown in Fig. 2-84:

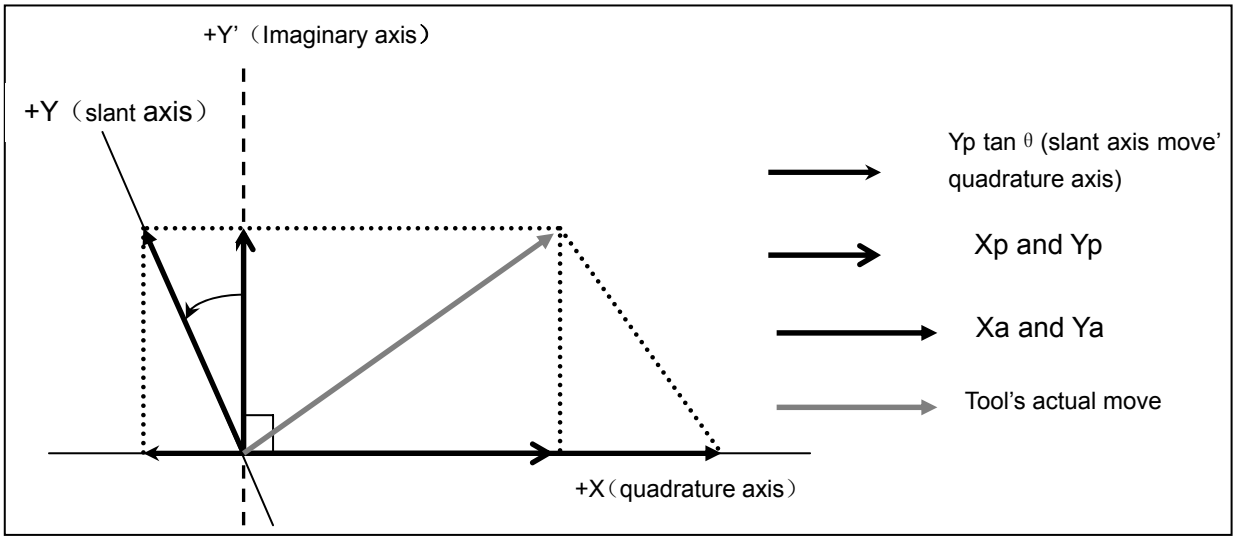


Fig. 2-84

The absolute position and relative position is expressed by the programmed cartesian coordinate system, and the machine position is expressed by the actual movement's machine coordinates according to the slant axis.

● Feedrate of each axis

When Y is a slant axis and X is a quadrature axis, to get the tangent direction's speed Fp, the controlled axis' feedrate is shown below:

Y's actual speed: $Fay = \frac{Fp}{\cos\theta}$ Fa: actual speed，Fp: programmed speed

X's actual speed: $Fax = Fp - Fp \times \tan\theta$

● Position coordinate display

The absolute position display and relative position display: is expressed by the programmed Cartesian coordinate system.

Machine position display: is expressed by the actual movement's machine coordinates according to the slant axis.

● Usage

The slant axis and quadrature axis used to control a slant are set in No.8211 and No.8212 in advance. But, when some parameter is set to 0, it is set to the same serial number or beyond the controlled axis quantity, selecting a slant axis and a quadrature axis is shown in the following table.

Slant axis	3 reference axes' X（the axis of No.1022 is set to 1）
Quadrature axis	3 reference axes' Z（the axis of No.1022 is set to 3）

1）AAC（No.8200#0）makes the slant axis control valid/invalid. When it is valid, the system executes the control according to the slant angel parameter（NO.8210）.

2）When the manual reference point return operation along the slant axis is executed, AZR（No.8200#2）selects whether to make the quadrature axis move by the slant axis.

3）Setting the invalid control signal NOZAGC（G63.5）of quadrature axis' slant axis to "1" makes the slant axis control function is valid to only the slant axis. At the moment, the slant axis' movement code can be transformed into slant coordinates, but the quadrature axis is not influenced by a slant axis' movement code. The signal is used when each axis individually moves.

2.29 G Code System B

GSK988TA/B has two set of G code, including G code system A and G cod system B. The previous described G codes use G code system A. Here, introduce their differences between programs and uses.

2.29.1 Differences of G Codes

2 set of G code system are referred to Table 2-23, Section 2.26.1. Their differences are described in Table 2-23.

Table 2-23

G codes		Group	Function	Classification
A	B			
G50	G92	00	Workpiece coordinate system	Non mode

			setting or max. spindle speed setting	
G90	G77	01	Axial cutting cycle	Mode
G92	G78		Thread cutting cycle	
G94	G79		Radial cutting cycle	
*G98	*G94	05	Feed per minute	Mode
G99	G95		Feed per rotation	
—	*G90	03	Absolute code	Mode
—	G91		Incremental code	
—	*G98	11	Fixed cycle return to initial plane	Mode
—	G99		Fixed cycle return to point R plane	

2.29.2 Absolute Code and Incremental Code G90, G91

Command function: G90 and G91 determine to use the absolute code or incremental code. G90 is an absolute code and G91 is an incremental code.

Command format:

Absolute code： G90 IP_;

Absolute code： G91 IP_;

Command explanation: main differences between A set of G code system and A set of G code system are to use the incremental code and absolute code. The A set of G code system using U, V, W and H separately means X, Y, Z, C's incremental code, but B set of G code system using G90/G91 means the currently used incremental value or absolute code value.

Example:

In A system, block：G01 X100 W100

In B system, block：G90 G01 X100

G91 Z100

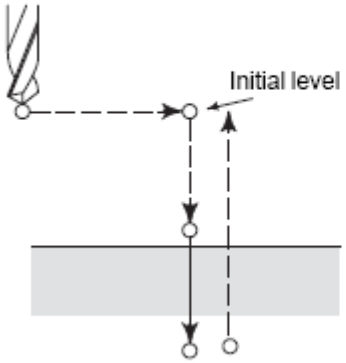
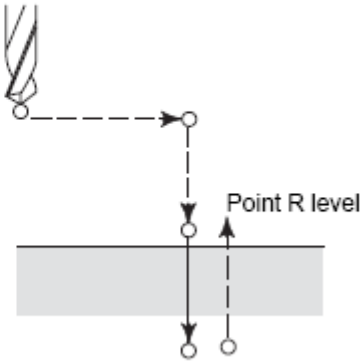
2.29.3 Cycle Code Processing

In B set of G code system, U, W set in G71~G76 blocks are taken as a parameter set by a cycle code to use, and is not taken as a axis movement code, even if U,W is set to a parallel axis.

The position code set in the cycle code is only specified by a corresponding axis code, and using G90/G91 means the current programmed position is set by an absolute vaue or incremental value.

2.29.4 Drilling Fixed Cycle's Return Operation G98, G99

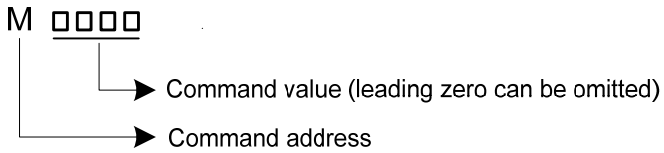
In A set of G code system, the tool returns to the initial plane from the hole bottom. In B or C set of G code, specifying G98 can make the tool return to the initial plane from the hole bottom and specifying G99 can make the tool return to point R plane from the hole bottom. The following figure specifies G98, G99 operations. Generally, the initial drilling uses G99 and the last uses G98.

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

Chapter 3 MSTF Codes

3.1 M (Miscellaneous Function)

M code consists of code address M and its following digits (the digit is set by No.3030), used for controlling the flow of executed program or outputting M codes to PLC .



There is one valid M code in one block. There are most specified 3 M codes in one block(set by NO.3404 Bit 7 (M3B)). The corresponding relationship between M codes and their functions are determined by the machine manufacturer. CNC sends M code signal and one strobe signal to PLC in executing M codes.

Except for M98, M198, M99, all M codes are executed in PLC. Their functions, meanings, control sequence and logic are referred to the machine manufacture's manual books.

3.1.1 End of Program M02

Command format: M02 or M2

Command function: In Auto mode, after M02 is executed and other codes of current block are executed, the automatic run stops. Whether the cursor returns the home of program is set by No.3404 Bit5 (M02). The cursor must return to the start of program when the program is executed again.

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

3.1.2 End of Program Run M30

Command format: M30

Command function: In Auto mode, after other codes 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).

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

3.1.3 Program Stop M00

Command format: M00 or M0

Command function: the system stops the automatic run after M00 block is executed, which is

same that of the single block pausing to save the previous modal message, i.e. which is equal to the program pause function. Press the CYCLE START key on the operation panel to execute the follow block and the CNC continuously automatically runs.

When M00 are other G code are in the same block, the system executes the code in the block, then M00, and last stops running.

3.1.4 Optional Stop M01

Command format: M01 or M1

Command function: after the block containing M01 is executed, the system stops the automatic run and the single block stopping signal lights. M01 is valid when the OPTIONAL STOP on the machine operation panel is pressed.

3.1.5 Subprogram Call M98

Command format：M98 P○○○○□□□□

Called subprogram number

Call times

Command function: In Auto mode, after other codes in the current block are executed in M98, CNC calls subprograms specified by P.

When the subprogram is called one time, ○○○○ can be omitted in inputting the number“○○○○□□□□” behind P, at the same time, the leading zero of the called subprogram number can be omitted and the system does not alarms. Example: M98 P12; it expresses to call the subprogram O0012 one time; the leading zero cannot be omitted when the subprogram call times are more than one.

The called subprogram name in M98 must be the program in the system and be less than 9999, and the subprogram name must be input.

The specified call times in M98 is 1～9999.

The called subprogram format in M98 is the following. The last end of the subprogram must be M99 instead of M30, its program compiling format is the same that of the main program compiling format.

Subprogram： O□□□□；（subprogram name）

…；

…；

M99；（return from subprogram）

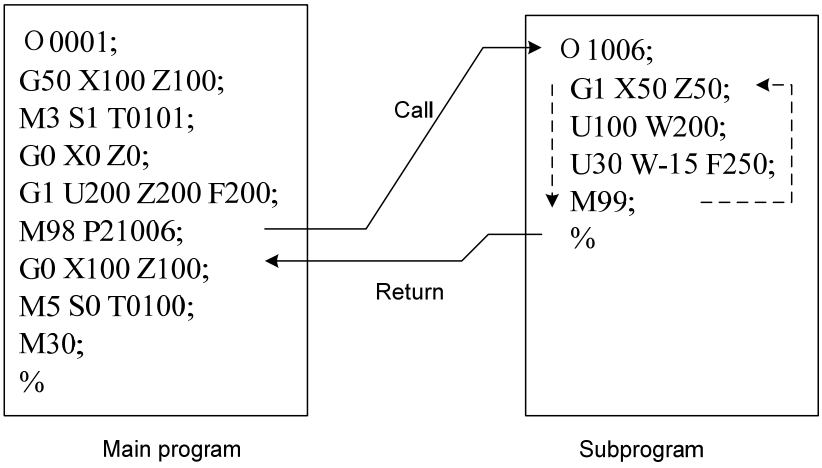


Fig.3-1 subprogram call

The called subprogram can call other subprograms. The subprogram called by the main program is called as the one-embedded subprogram, and the one called by the one-embedded subprogram is called as the two-embedded subprogram and so forth. One main program can call 12-embedded subprogram (including macro program call). The following is the four-embedded subprogram.

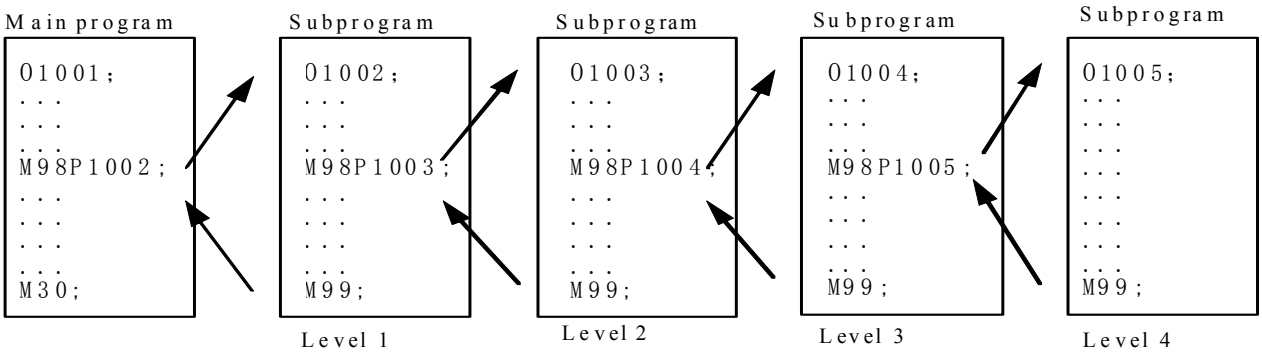


Fig. 3-2 Subprogram nesting

Note 1: An alarm occurs when the system has not searched the subprogram specified by P;

Note 2: In MDI mode, inputting M98P_ cannot call a subprogram, otherwise, an alarm occurs;

Note 3: An alarm occurs when P98P__ call itself;

Note 4: An alarm occurs when M98 is executed and the subprogram is called without P code.

3.1.6 Subprogram Call M198

Command format：M198 P○○○○ □□□□

Called subprogram number

Repetitive call times

Command function: in Auto mode, when M198 is executed and the other codes in the current block has been performed, the CNC calls the subprogram in the external input/output device (usually, it is U disk) specified by P.

The code usage and notes are those of M98, and the operations are referred to M98.

3.1.7 Return from Subprogram M99

Command format: M99 P○○○○○

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

Command function: After other codes 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).

Example: Execution path of calling subprogram (with P in M99) as Fig. 3-3.Execution path of calling subprogram (without P in M99) as Fig. 3-4.

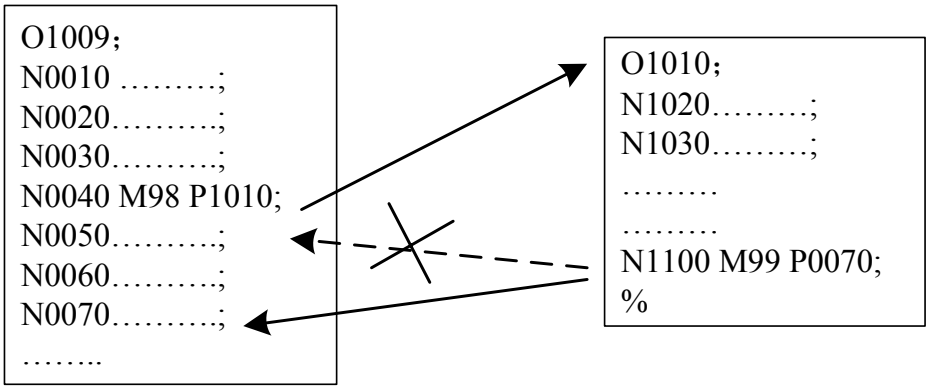


Fig. 3-3

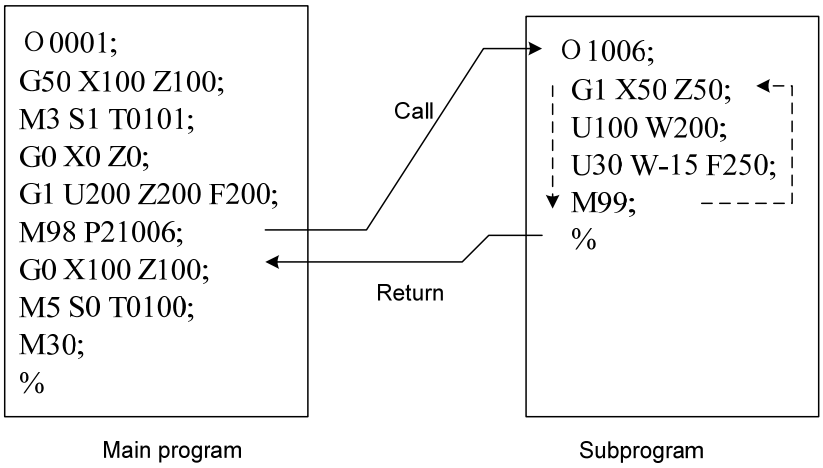


Fig. 3-4

Note 1: M99 does not need to be specified in the alone block. Example: G00 X100 Z100 M99;

Note 2: An alarm occurs when M99 has executed the block number which does not exist;

Note 3: In Auto mode, the specified block number following M99 is repetitive, and the system executes the followings: 1. when the two block numbers are front of M98 block, the program returns to the later

repetitive block; 2. when the two block numbers are behind of M98 block, the program returns to the top repetitive block; 3. when the two block numbers are separately in front of or behind of M98 block, the program returns to the later repetitive block;

Note 4: In Auto mode, the system ignores the line and returns to the beginning of the file to perform the execution when the main program ends in M99 and specifies the line number following P.

3.1.8 Standard M Codes for Standard Ladder

Some functions may be relevant with the system's allocation, and the used M codes for some machine are referred to the machine manufacture's user manual.

Code	Function	Remark
M00	Program pause	
M03	Spindle clockwise (CW)	Functions interlocked and states reserved
M04	Spindle counterclockwise (CCW)	
*M05	Spindle stop	
M08	Cooling ON	Functions interlocked and states reserved
*M09	Cooling OFF	
M10	Tailstock forward	Functions interlocked and states reserved
*M11	Tailstock backward	
M12	Chuck clamping	Functions interlocked and states reserved
M13	Chuck releasing	
M14	Spindle position control	Functions interlocked and states reserved
*M15	Spindle speed control	
M16	The 2nd spindle position control	Functions interlocked and states reserved
*M17	The 2nd spindle speed control	
M18	The 3rd spindle position control	Functions interlocked and states reserved
*M19	The 3rd spindle speed control	
*M20	Positive rigid tapping	Functions interlocked and states reserved
M21	Reverse rigid tapping	
*M24	The 1st spindle rigid tapping	Functions interlocked and states reserved
M25	The 2nd spindle rigid tapping	
M26	The 3rd spindle rigid tapping	
M29	Rigid tapping	
M32	Lubricating ON	Functions interlocked and states reserved
*M33	Lubricating OFF	
M35	Spindle HOLD start	Functions interlocked and states reserved
*M36	Spindle HOLD close	
M37	Chip cleaner rotation CCW	Functions interlocked and states reserved
M38	Chip cleaner rotation CW	
*M39	Chip cleaner stop	
M41,M42 M43,M44	Spindle automatic gear shifting	Functions interlocked and states reserved
M51 ~ M58	Spindle 8-point orientation	Functions interlocked and states reserved

M63	The 2nd spindle rotation CCW	Functions interlocked and states reserved
M64	The 2nd spindle rotation CW	
*M65	The spindle 2nd spindle stop	
M73	The 3rd spindle rotation CCW	Functions interlocked and states reserved
M74	The 3rd spindle rotation CW	
*M75	The 3rd spindle stop	

3.1.9 Notes for M Codes

1. M00, M01, M02, M30, M98, M99 is separately specified in one block. When it with other M code are specified, the system ignores the other M code and the above M code is executed; when the above seven M codes are in the same block, the first executed M code is valid.
2. When M05, M11, M13, M33, M9 and G codes are in the same block, there are two execution methods:

   a) The motion codes and M miscellaneous function codes are executed simultaneously.
   b) The miscellaneous function codes following the motion codes are executed.

   Refer to the tool manufacturer's user manual. The second method is executed for GSK's standard ladder.
3. CNC permits there are up to specified 3 codes in one block (when NO.3404 Bit7 M3B is set to 1), some M codes cannot be specified simultaneously because of machinery operation, such as the spindle's automatic gear change codes: M41, M42, M43, M44.
4. No.3010 sets the delay time of the strobe signal MF, SF, TF signals.
5. No.3011 sets the width of M, S, T function end signals (FIN).

3.2 Spindle Function

S code is used to controlling spindle speed. In GSK988TA/TB spindle speed control, NC outputs 0~10V analog voltage signal to spindle servo device or inverter to realize the gradeless spindle speed.

3.2.1 Spindle Speed Analog Voltage Control

Command format: S □□□□□

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

Command explanation: spindle speed analog voltage control code

□□□□□ means the set spindle speed, its value range is referred to Table 1-4, and the leading zero can be omitted. When the value exceeds the range set by No.3772, the most spindle speed limit is specified in the program, and S value is specified to the most spindle speed; when it is not specified, the upper and lower limit of S value is specified. The system alarms when the decimal is input to the specified of the S value. The system can set the digit number by No.3031.

The first spindle of the CNC can execute 4-gear spindle speed, and the second spindle has 2-gear spindle speed. In executing S code, the system counts the analog voltage value corresponding to the specified speed according to setting value(corresponding to No.3741～No.3744) of max. spindle speed (analog voltage is 10V)of current gear, and then outputs to spindle servo or converter to ensure that the spindle actual speed and the requirement are the same.

After the CNC 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 code 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.

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 code(r/min), and is invariant without changing S code 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 code, 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.

3.2.2 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 should be referred when using it. Refer to the following functions of GSK988TA/B standard PLC ladder.

The spindle actual speed specified by GSK988TA/TB 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 Ⅱ OPERATION.

3.2.3 Multi-Spindle Control

GSK988TA/TB has a multi-spindle control function (controlling up to 3 spindles). Besides the 1st spindle control, S code from CNC can control the 2nd and the 3rd spindle. The spindle code is the same with the previous, using one S code. PLC sending a signal or address P code determines to select one of 3 spindles.

Like the 1st spindle, the 2nd spindle and the 3rd spindle can execute 2-level gear switch.

Besides, max. speed set by each spindle can be clamped at their separate speed. (it is determined by the setting of No.3772.)

The system can select the 2nd, 3rd spindle's position encoder interface; the 1st ～3rd position encoder selection is determined by the signal from PLC.

Note: the multi-spindle control is valid when No.3710 CNC controllable spindle quantity exceed 1.

● Control method

Multi-spindle speed specifies 2 methods: 1：S code and the spindle rotation M code are executed (supported by PLC), 2：P code specifies the speed to the spindle.

1：S code and the spindle rotation M code executing relevant parameters (take the standard ladder defaulted by the CNC as the standard)

Relevant parameter setting:

	Parameter number	Parameter definition	Setting value
System parameter	3703#3	In multi-spindle control, whether to use SWS to perform the spindle selection: 0 No，1 Yes	0
	3709#2	In multi-spindle control, SIND is valid 0: it is valid only to the 1st spindle 1: it is valid to all spindles	1
PLC parameter	K16.2	In multi-spindle control, the spindle speed S 1：it with M are executed，0：it is specified by P	1

Relevant M command explanation:

M03, the 1st spindle rotation (CW)，M04, the 1st spindle rotation (CCW)，M05, the 1st spindle stop

M63, the 2nd spindle rotation (CW)，M64, the 2nd spindle rotation (CCW)，M65, the 2nd spindle stop

Usage: using the 1st spindle can directly use M03/M04 S****，using the 2nd spindle can use M63/M64 S****. When S is executed but M code for the spindle rotation is not specified in running programs, PLC outputs the speed code to the spindle which is the last one to start rotation (CW/CCW), but when the system is turned on and it does not code any spindle rotation (CW/CCW), it codes S code to output to the 1st spindle.

Programming example:

```
N10   M03 S1000   (start the 1st spindle rotation(CW), speed: 1000 rotations)
N20   T0101
 *****
 *****
N50   S1500  （the 1st spindle speed: 1000 rotations）
 *****
 ****
N100   M63 S2000（start the 2nd spindle rotation(CW), speed: 2000 rotations）
 ******
 ******
N150   S1800（change the 2nd spindle speed into 1800 rotation）
 *****
 *****
```

N200 M05 (stop the 1st spindle)
N210 M65 （stop the 2nd spindle）
N220 M30

2：**P code specifying parameters relevant with speed**

	Parameter number	Parameter definition	Setting value
System parameter	3703#3	In multi-spindle control, whether to use SWS to select the spindle: 0 No, 1 Yes	1
	3706#2	In multi-spindle control, when the system sets to select the spindle by address P, it specifies P without S: 0: an alarm occurs 1: use the last P executed by S_ P_；. After the system is turned on, No.3775 value is used but P is never executed	To be set
	3709#2	In multi-spindle control, SIND is valid 0: it is valid to only the 1st spindle 1: it is valid to all spindles	1
	3713#6	In multi-spindle control, when a program code based on address P executes the spindle selection, whether the position encoder feedback used in thread cutting/feed per rotation is automatically switched in accordance with the selected spindle 0: do not switch 1: switch	0
	3775	In multi-spindle, the defaulted spindle selects P code value (MPD): in multi-spindle control, set the defaulted P code value when S_P is not executed one time after power-on	1
	3781	In multi-spindle, select the spindle's P code (MPS): the parameter sets to select each spindle's P code in multi-spindle control. Specify P in the block in which the S code is	S1：1 S2：2 S3：3
PLC	K16.2	In multi-spindle control, the spindle speed S 1：it is executed with M，0：it is specified by P	0

Example：

N10 M03 P1 S1000 (start the 1st spindle rotation(CW), speed: 1000 rotations)
N20 T0101
*****
*****
N50 P1 S1500 （the 1st spindle speed: 1500 rotations）
*****
****
N100 M63 P2 S2000（start the 2nd spindle rotation(CW), speed: 2000 rotations）
******

******
N150 P2 S1800（change the 2nd spindle speed into 1800 rotation）
*****
*****
N200 P1 S500 (the 1st spindle speed is executed to 500 rotations)
N210 P2 S700 （the 2nd spindle speed is executed to 700 rotations）
N220 M30

Note 1：No matter what the spindle uses the method, the spindle override is valid to the spindle which is the last to start the rotation (CW/CCW).

Note 2：The spindle constant surface speed control is valid to the 1st spindle.

3.3 Tool Function

3.3.1 Tool Offset

Tool functions (T code) of GSK988TA/TB: automatic tool change and executing tool offset. Control logic of automatic tool change is executed by PLC and tool offset is executed by NC.

Command format:

T □□ ○○

Tool offset number（the leading zero cannot be omitted）

Target tool number（the leading zero can be omitted）

Command function: The automatic tool post rotates to the target tool number and the tool offset of tool offset number executed 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 code is the state before the system is switched off, the tool offset number is in the cancelling state(i.e. 00 state).No. 3032 sets T code digit, and No.5002 Bit 0(LD1) sets the digit of tool offset number.

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 code 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.

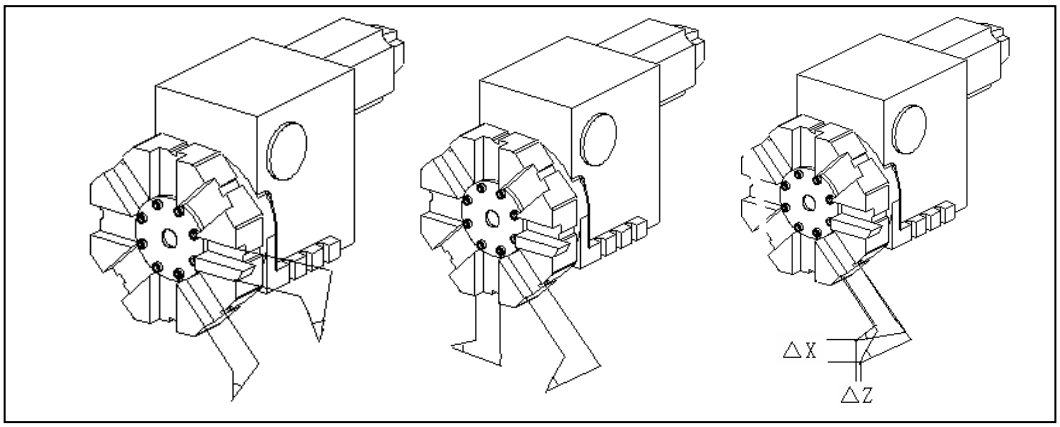


Fig.3-5 Tool offset

The tool offset is used for the programming. The offset corresponding to the tool offset number in T code is added or subtracted on the end point of each block. X tool offset in diameter or radius is set by No.5004 Bit1(ORC). For X tool offset in diameter or radius, the external diameter is changed along with diameter or radius when the tool length compensation is changed.

Example: When the state parameter No.5004 Bit1 is set to 0 and X tool length compensation value is 10mm, No.5004 Bit1 is set to 1 and X tool length compensation value is 10mm the diameter of workpiece external diameter is 20mm.

Fig. 3-6 is to create, execute and cancel the tool offset in movement mode.

Example: When the state parameter No.5004 Bit1 is set to 0 and X tool length compensation value is 10mm, No.5004 Bit1 is set to 1 and X tool length compensation value is 10mm the diameter of workpiece external diameter is 20mm.

Fig. 3-6 is to create, execute and cancel the tool offset in movement mode.

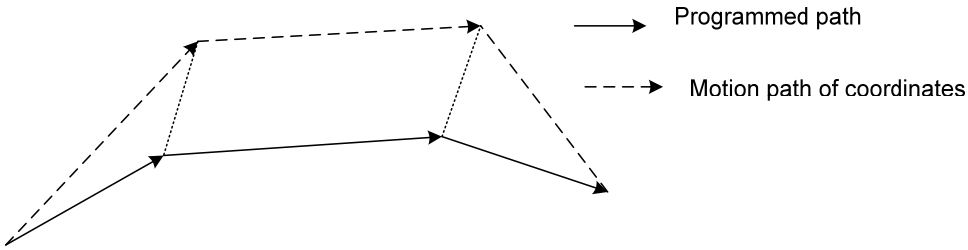


Fig. 3-6 Creation, execution and cancellation of tool length

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

G01 W150； （Block 2, tool offset）

G01 X50 Z300 T0100； （Block 3, canceling tool offset）

There are two methods to execute the tool offset(they are set by No.5002 Bit4(LGT)):

（1）The tool length compensation is executed by the tool traversing;

（2）The tool length compensation is executed by modifying the coordinates;

Example:

Table 2-2

Tool offset number		X	Z
01	Offset	0.0000	0.0000

	Wear	0.0000	0.0000
02	Offset	12.0000	-23.0000
	Wear	0.0000	0.0000
03	Offset	24.5600	13.4520
	Wear	0.0000	0.0000

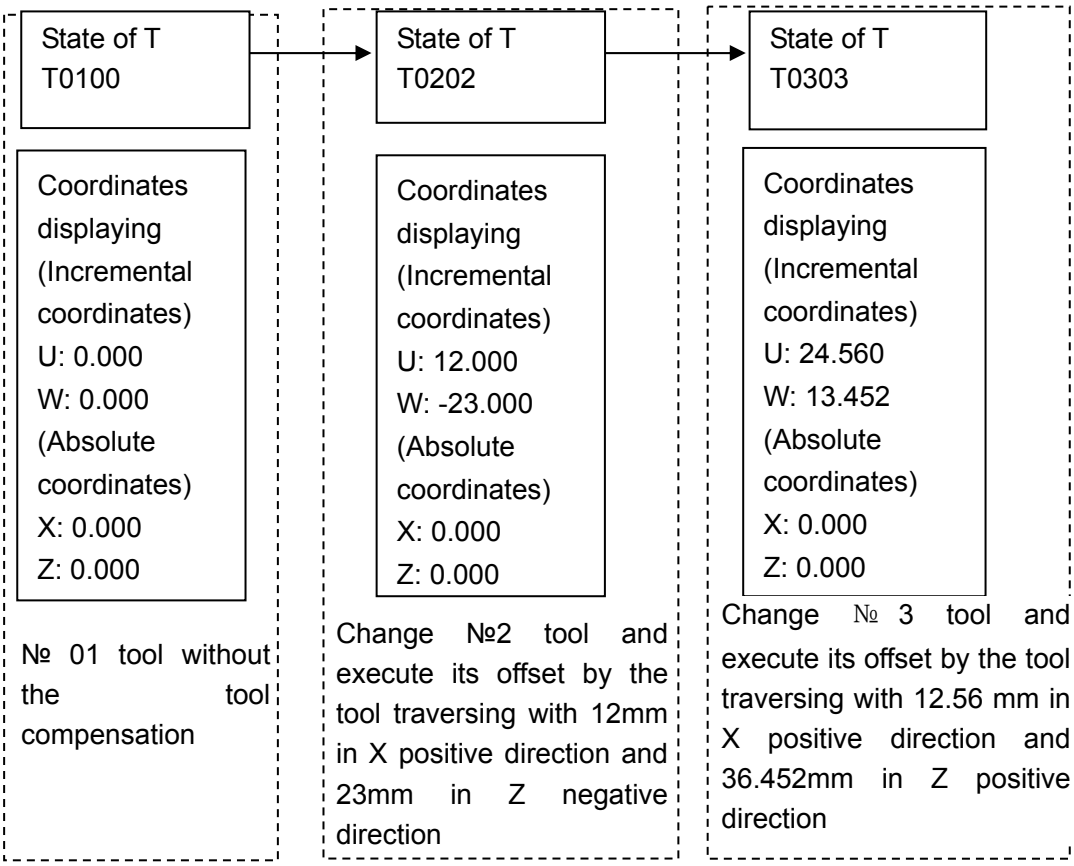


Fig. 3-7 Tool traversing mode to execute the tool offset

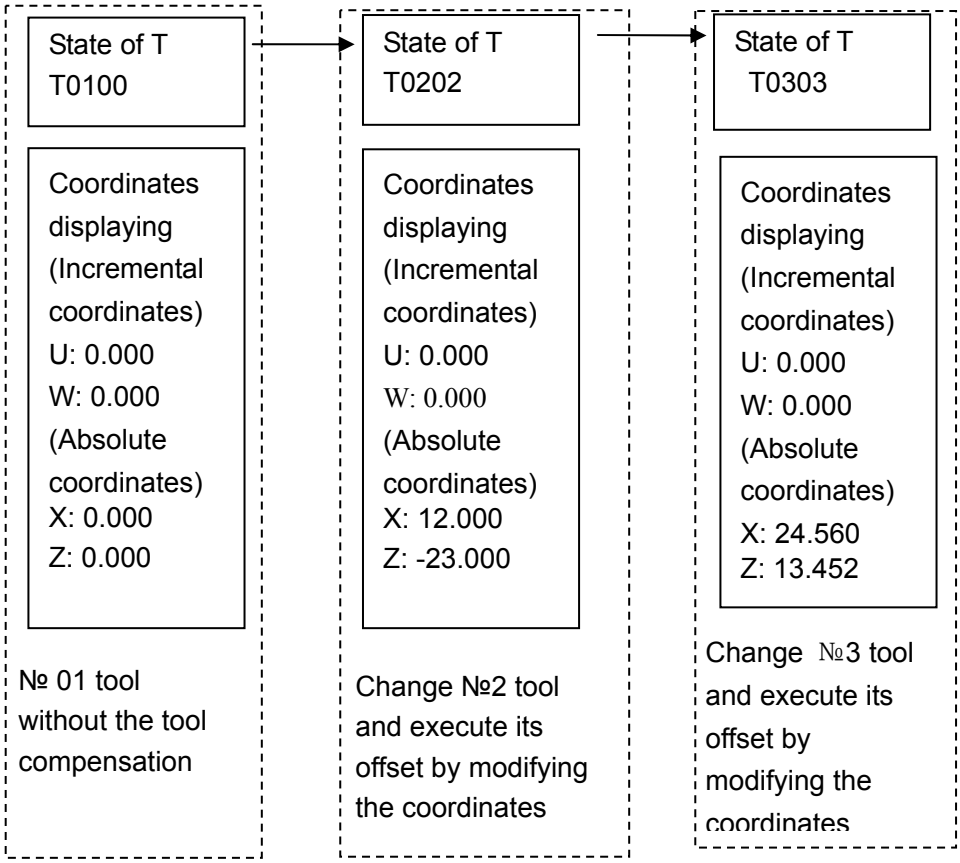


Fig. 3-8 Modifying the coordinates mode to execute the tool offset

When T code and the motion code are in the same block, they are executed simultaneously, in executing tool change, the system executes by adding the current tool offset to coordinates of motion code and whether the cutting feedrate or the rapid traverse speed is defined by the motion code.

Note 1: In tool traversing compensation mode, when the system executes the tool offset, NO. 5002 Bit6 (LWM) sets the valid method of the tool offset:

LWM=0: it is valid in the block of T code

LWM=1: it is valid in the axis movement block

Note 2: No.5001 Bit6（EVO）sets the valid method of modifying tool wear tool when the system executes the program:

EVO =0: it is valid in the next specified T code

EVO =1: it is valid in the next buffer block

Note: After the tool wear value is modified, the system executes the wear value according to No.5002.2 setting method to avoid the too big wear value.

Note 3: It is suggested that the program should be complied according to the normative method, that is, the tool change is executed in the safe position and then the absolute value code is used to position to the starting point of the machining.

Note 4: In the coordinate offset compensation mode, when the system executes the tool offset and executes T function code instead of movement code, it uses G50 to set the coordinate system, the displayed absolute coordinate value is that the coordinate value set by G50 adds or subtracts the unexecuted tool compensation value.

Note 5: T code can use the leading zero. When T00□□ is executed or only tool offset number is executed in the program and the current tool number is not changed, the system only correspondingly modifies the current tool offset value.

Note 6: After executing the miscellaneous function lock is valid, the system does not execute the tool change when T code is executed but the tool offset is executed. When using the miscellaneous function lock checks the program function, it is executed in the safety position.

3.3.2 Tool Life Management

When the tool life management is used, TLF (No.8132#0) is set to "1".

The tools are divided into many groups, each group specifies a corresponding tool life (used time or times). The tool is used one time, the used time or times is added up total. When the current tool life reaches, the system selects the next tool in the same group according the set tool number order in advance. An alarm occurs when all tool lives are run out of in the group.

3.3.2.1 Tool Life Management Data

Set the used max. group number in No.6813. No.6800 bit0 and bit1（GS1 andGS2）set the actual group quantity and the usable max. tool quantity in each group.

GS1	GS2	Group quantity	Tool quantity
0	0	1/8 of 1~max. group quantity（NO.6813）	1~16
0	1	1/4 of 1~max. group quantity（NO.6813）	1~8
1	0	1/2 of 1~max. group quantity（NO.6813）	1~4
1	1	1~ max. group quantity（NO.6813）	1~2

Note 1：After No.6813 or No.6800 bit 0 and bit 1（GS1 and GS2）, executing the tool life data' s input program sets the tool life data again.

Note 2: The same tool number exists in any place at any time in the program of tool life data.

Note 3：T code for the tool registration is composed of the tool selection number and tool offset number.

Note 4：When the tool life management function is used, No.5002 bit0 must be zero.

3.3.2.2 Tool Life Time Count

When the tool life count mode is not specified in the tool life data input program, setting LTM (No.6800#2) value determines the time or times to specify the tool life.

1. Specify the tool life by the used time

FGL（#6805.1) specifies the tool life's unit（0 : 1 minute as the unit；1 : 0.1 sec as the unit）.

When the tool group code（T□□99）is specified, the tool which life has reached in the group, and the system starts executing the selected tool's tool life management. The required time in single block stop, feed hold, rapid traverse, dwell, machine lock and interlock is not counted into the currently used tool life.

The tool life can be specified up to 4300 minutes. The set max. life is 4300（minutes）or 2580000（0.1 sec） minutes according to #6805.1.

2. Specify the tool life by the used times

When the tool group code（T□□99）is specified, a tool which life has not reached in the group is selected, which makes the selected tool life count is added to 1. But the tool life count is not executed and M code is started, only when the tool life count is not executed to start M code, the

reset state is in the initial tool group number code and tool change code of automatic run start state, the system can execute the new tool selection and count.

The tool life can be set up to 65535 times.

Note: the same tool group number in a program is executed many times, the used times is not added up and a new tool is not selected.

3.3.2.3 Tool Life Count Restarting M Code

When the tool life count is specified by the times, and the tool life count restarting M code is executed, the tool group which life has reached even if there is one tool outputs the tool change signal. In the tool group code (T code) after the tool life count restarting M code, a tool which life has not reached in the specified tool group is selected, and the tool life count is added up to 1. So, even if the CNC enters other conditions except for the initial tool group code (T code) following the reset state into the automatic run start state. The tool life count restarting M code is specified by No.6811.

3.3.2.4 Tool Life Management Code in Machining Program

The tool life is used in the machining program, T code codes the tool group according to the following format.

Command format:

……

T□□99；end the previous group's tool life count, start to use the tool which life has not reached in group □□ and output T code signal, and start to execute the tool life count in group □□.

……

T□□88；end the tool life management in group □□，cancel the tool offset which is being used, output the tool number and T code signal

……

M02（M30）； end the machining program；

Command function:

Execute the machining according to the specified group and the tool life management

Example：suppose that the digit of the offset number is 2	
T0199； ： ： ： ： T0188； ： ： ： ： T0299； ： ： ： ： T0299； ： ： ： ： T0301； ： ： ：：	Select the tool which life has not reached in group 1 （Suppose that T1001 is selected，the tool number is 10，and theoffset number is 01.） Select the tool life count in group 1 （count the tool life which tool number is 10） Cancel the tool post's offset which is being used in group 1 （because the tool which is being used is T1001，the tool number is 10，and the tool number is 00.） Select the tool which life has not reached in group 2 （Suppose that T2002 is selected，the tool number is 20，and theoffset number is 02.） Select the tool life count in group 2 （count the tool life which tool number is 20） Many offset numbers in the tool being used in group 2 are executed, the next offset number is selected. （suppose there is T2002 T2003 in the tool number 20, T2003 is selected. When the tool number is 20，the offset number is 03.） The tool life count ends in group 2，which is executed as a common T code. （the tool number is 03，and the tool number is 01.）

Note: When T□□99 is not executed before T□□88, an alarm occurs.

3.3.2.5 Automatically Inputting a Tool Life Data

Using G10/G11 can input the tool life management data and its format is shown below:

（1）delete data in all groups when log-in：

Format	Symbol description
G10 L3； P- L-； T-； T-； . P- L-； T-； T-； .	G10 L3: delete all groups when log-in P-：group number L-：tool life value T-：tool number and tool offset number G11：log-in ends

G11； M02(M30)；	

After the logged-in all tool life management data are deleted, the system logs-in the programmed tool life management data.

（2）Change the tool life management data

Format	Symbol description
G10 L3 P1； P- L-； T-； T-； . P- L-； T-； T-； . G11； M02(M30)；	G10 L3 P1: start to change the group data P-：group number L-：tool life value T-：tool number and tool offset number G11：log-in ends

Set the tool life management data or change the logged-in tool life management data in the tool life management data group which is not logged in.

（3）Delete the tool life management data:

Format	Symbol description
G10 L3 P2； P- ； P- ； P- ； P- ； . G11； M02(M30)；	G10 L3 P2: start to delete the group data P-：group number G11：the deletion ends

（4）Set the tool life group's count type

Format	Symbol description
G10 L3 ； （G10 L3 P1）； P- L- Q-； T-； T-； . G11； M02(M30)；	Q: life count type （1：times, 2：time）

Note 1：P following G10 being 1 means the group's data change, being 2 means to the group's data, omitting P means to delete all groups and log-in the tool life group. Cannot mix with P tool life group number executed internally by G10

Note 2：When Q is omitted, the life count type is set according to LTM（No.6800#2）'s setting value.

3.3.2.6 Process when the Tool Life End

When the tool life count is executed and the last tool life in the group has reached, a tool change signal is output. When the tool life count is specified by time, a tool change signal is immediately output once the last tool life in the group has reached. When the tool life count is specified by times, the last tool life in the group has reached, the CNC is reset by M02 or M30, or the tool life count restarting M code is executed, a tool change signal is output.

When LFI(No.6804#6) is set to 1，the tool life count's invalid signal LFCIV can switch the tool change life count to be valid/invalid.

When the tool life count's invalid signal LFCIV is set to 1, the tool life count's invalid signal LFCIF being 1 means the life count to be invalid.

When the tool life count's invalid signal LFCIV is set to 0, the tool life count's invalid signal LFCIF being 0 means the life count to be invalid.

3.3.2.7 Tool Life's Relevant Signal

Tool change signal TLCH＜Fn064.0＞

[Classification] output signal

[Function] the signal informs the last tool life in the group has reached.

[Output condition] when the followings become '1'.

The tool life has reached, the next tool in the group is selected orderly, and the last tool life in some group has reached.

When the followings become '0'.

When there is no tool group which life has reached.

The signal becomes '1'，the CNC sends the tool change reset signal TLRST or MDI operation by PLC side, the signal becomes '0' when the CNC informs the tool change has completed for the all groups which life have reached.

Tool change reset signal TLRST＜Gn048.7＞

[Classification] input signal

[Function] the group life count, all execution data with *, @ is cleared.

After all tools in the group which life have reached displayed in the window are changed into new tools, the tool group number selection signal (TL1～TL512) specifies the group number and the singal is input. When GRS(No.6800#4) is set o "1", the tool group number selection signal cannot be input, the logged-in all groups' data are cleared.

Besides, operations by MDI mode are cleared.

Signal for tool change one by one TLCHI＜Fn064.2＞

[Classification] output signal

[Function] when the tool life count is specified by time, it informs the currently used tool life has reached. A tool change program is inserted by the signal, the program can be restarted after the tool change is executed.

Reset signal for tool change one by one TLRSTI＜Gn048.6＞

[Classification] input signal

[Function] the tool signal TLCHI for the tool change one by one is set to '0'.

Tool skip signal TLSKP＜Gn048.5＞

[Classification] input signal

[Function] the CNC skips the tool which life has not reached to forcibly select the next tool. Select one of the following two methods by SIG (No.6800#3).

(i) Using the tool group number selection signal specifies the group number (SIG='1')

The tool group number selection signal (TL1～TL512) specifies the tool's group number and the tool skip signal is set to '1'. By the operation, the system selects the next tool in the next T code having specified the skip group.

(ii) Specify the group number not by the tool group number selection signal (SIG='0')

The tool skip signal TLSKP is set to '1' without specifying the group number, which is taken as the system has specifies the currently selected tool's group. By the operation, the system selects the next tool in the next T code having specified the skip group. But, when the tool skip signal TLSKP is set to '1' relative to the last tool, the tool change signal TLCH becomes "1".

A new tool selection signal TLNW＜Fn064.1＞

[Classification] output signal

[Function] the signal informs the PMC side has selected the group's new tool. When the new tool is selected, and automatically measuring the tool's tool length compensation is performed, it is used.

[Output condition] when the followings become '1'.

When the tool group number is executed by T code, the used tool's life has reached in the group and the next new tool is selected. After a new tool code signal is sent, the signal becomes '1' while the system sends the tool function strobe pulse signal TF.

When the followings become '0'.

When the signal is '1'，the strobe pulse signal TF completion signal FIN becomes '1'.

Tool group number selection signal TL01～TL128 ＜Gn047.0~Gn047.7＞

[Classification] input signal

[Function] specify the tool group number. When the tool change reset signal TLRST or the tool skip signal TLSKP is input, the signal specifies some group to execute the tool change reset or tool skip.

Tool life count override signal *TLV0～*TLV9＜Gn049.0~Gn050.1＞

[Classification] input signal

[Function] when the tool life count type is specified by time, LVF(No.6801#2) is set to "1"，i.e., the tool life count is applied with override. Its 10 binary code signals and the override value are executed according to the following method

$$Overrideva\ \ lue = \sum_{i=0}^{9} \{2i \times Vi\ \ \}times$$

Use it in the override value range.

*TLVi is '1'，Vi=0

*TLVi is '0'，Vi=1

*TLV7, *TLV6, *TLV3 are '0': the override value is counted according to the following formular:

12.8+6.4+0.8=20.0

So, the life count is twentyfold of the previous.

When all signals are '1'，the override value becomes 0 times. Set it within 0～99.9 times in every step 0.1 times.

It is clamped at 99.9 times when it exceeds 99.9 times.

The tool life management count time is time that the actual cutting time based on the life count by time multiplies the signal's selected override value. Example: when the override value is 0.1 times, the actual cutting time is set to 1000 seconds, the tool life count time is 100 seconds.

Tool life anticipating signal TLCHB ＜Fn064.3＞

[Classification] output signal

[Function] by setting the redesigned value before selecting a new tool, using the life count, when the life surplus (life value- life count value) in the group is the "same" with the set redesigned value, or in the "following", the system outputs the tool life anticipating signal to anticipate in advance. The tool life anticipating signal is output by ARL(No.6802#4) selecting a tool one by one or when there is the last tool in the group. The CNC redesigns the count value and uses (No.6844,No.6845).

[Output condition] When the followings become '1'.

RMT(No.6802#7)=0:

Life surplus（life value－life count value）≦redesigned count value

RMT(No.6802#7)=1:

Life surplus（life value－life count value）= redesigned count value

When the followings become '0'.

RMT(No.6802#7)=0:

Life surplus（life value－life count value）> redesigned count value

RMT(No.6802#7)=1:

Life surplus（life value－life count value）≠redesigned count value

Tool life count's invalid signal LFCIV<Gn048.2>

[Classification] input signal

[Function] make the tool life count invalid when selecting.

Tool life count being invalid's signal LFCIF<Fn093.2>

[Classification] output signal

[Function] the signal informs the being selected tool life count is invalid.

[Output condition] when the followings become '1'.

The tool life count's invalid signal LFCIV is set to '1' and the life count is invalid.

When the followings become '0'.

The tool life count's invalid signal LFCIV is set to '0' and the life count is valid.

Chapter 4 Tool Nose Radius Compensation

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 generally regarded as the tool nose A point in an imaginary state (there is no 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. 4-1). Its nose of turning tool is not the imaginary point but one arc owing to the processing and other requirement in the practical machining. There is an error between the actual cutting point and the desired cutting point, which will cause the over- or under-cutting affecting the part precision. So a tool nose radius compensation is needed in machining to improve the part precision.

Fig. 4-1 Tool

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

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

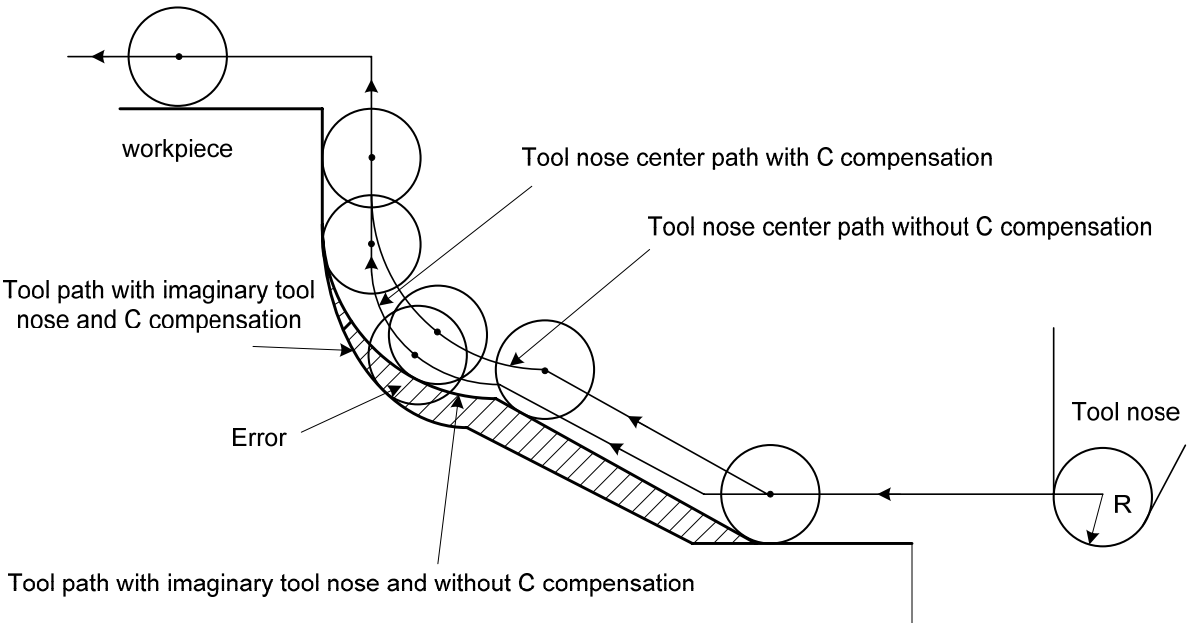


Fig. 4-2

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.

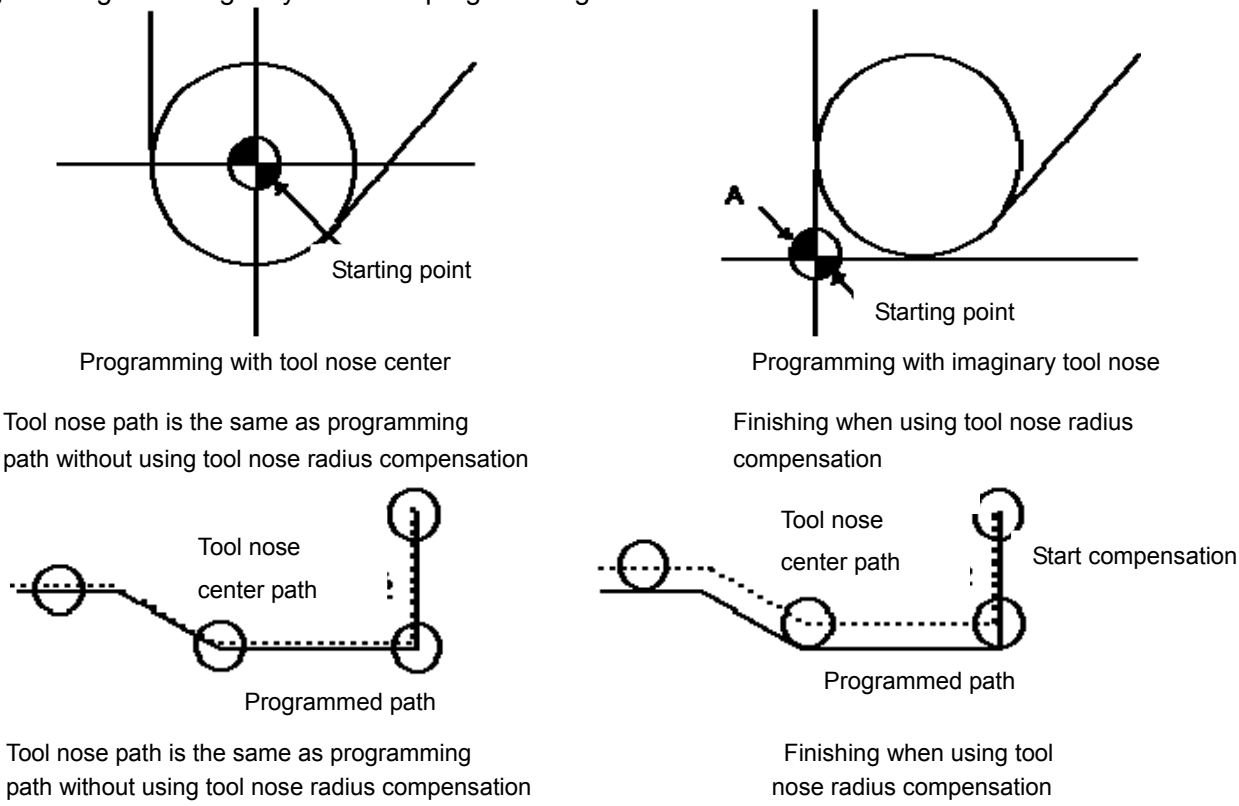


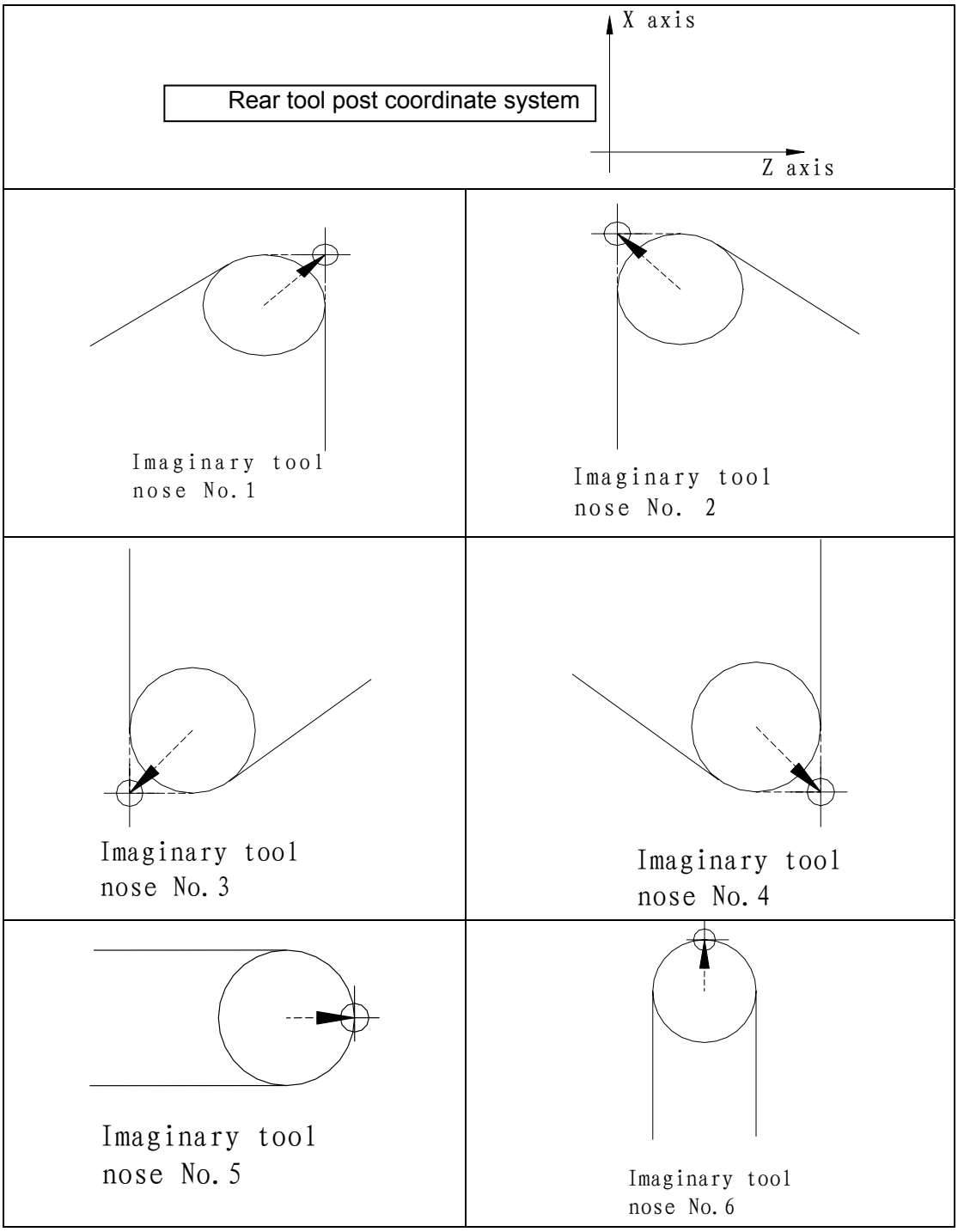
Fig. 4-5 Tool path in tool nose center programming

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 circular, machining error is caused which can be deleted by tool nose circular radius compensation. In actual machining, suppose that there are different position relationship between tool nose point and tool nose circular center point, and so it must create its correct 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（T0～T9） kinds of tool nose setting and 9 directions for position relationship. The tool nose directions are different in different coordinate system (rear tool

post coordinate system and front tool post 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 tool post coordinate system is as Fig. 4-7; T1～T8 in front tool post coordinate system is as Fig. 4-8. The tool nose center and starting point for T0 and T9 as Fig. 4-9.



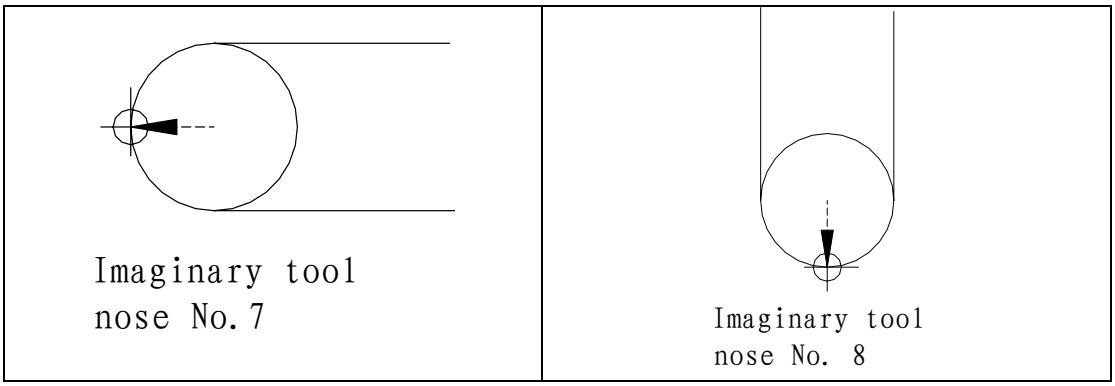
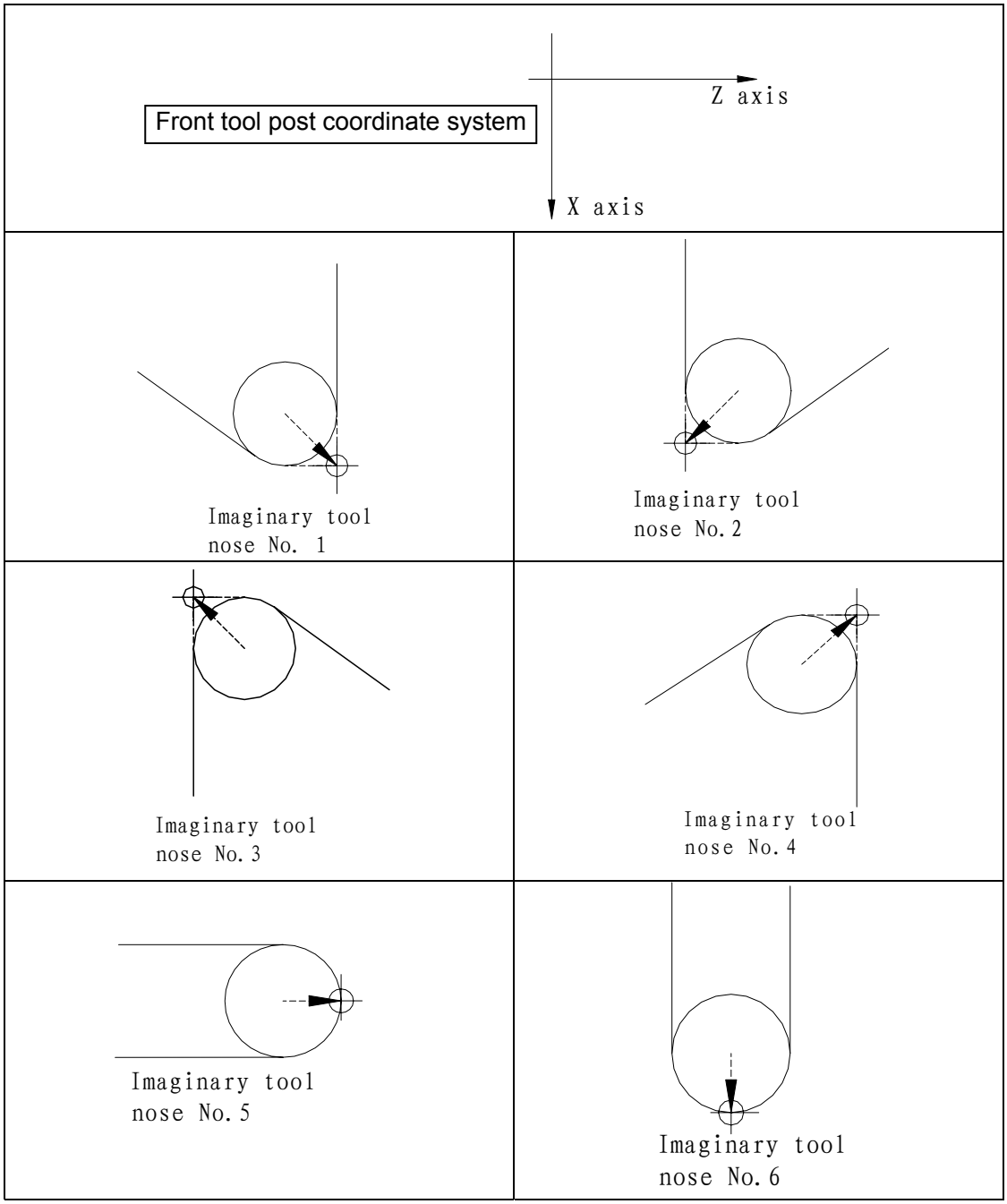


Fig. 4-7 Imaginary tool nose number in rear tool post coordinate system



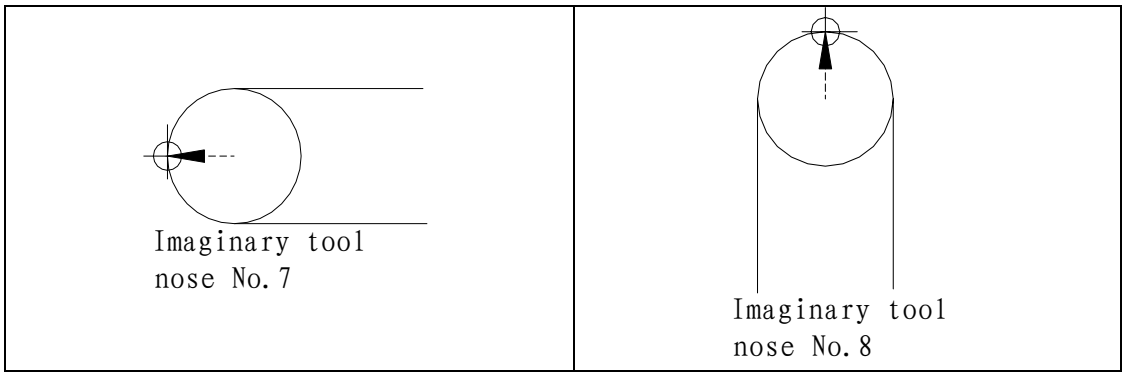


Fig. 4-8 Imaginary tool nose number in front tool post coordinate system

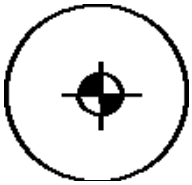


Fig. 4-9 Tool nose center on starting point

Note: The general imaginary tool nose direction 1～8 are used to G18 level, the imaginary tool nose 0 or 9 is used to G17 and G19 levels. The imaginary tool 0 or 9 used to G18 is valid, but the imaginary tool nose direction 1～8 are used to G17 and G19 levels, the system uses the nose 0 to execute the compensation.

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 **"TOOL OFFSET&WEAR"** window (see Fig. 4-1), R is tool nose radius compensation value, T is imaginary tool nose number, and the radius compensation value is the sum of offset radius and wear radius.

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

Tool offset No.		X	Z	…	R	T
001	Offset	0.000	0.000	…	0.380	3
	Wear	0.000	0.000	…	0.000	
002	Offset	10.000	10.000	…	0.250	3
	Wear	0.020	0.040	…	0.000	
003	Offset	14.000	15.000	…	1.200	3
	Wear	1.020	0.123	…	0.000	
…	Offset	…	…	…	…	…
	Wear	…	…	…	…	
099	Offset	10.000	12.000	…	0.300	0
	Wear	0.050	0.058	…	0.000	

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 tool post coordinate system, taking tool post 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).

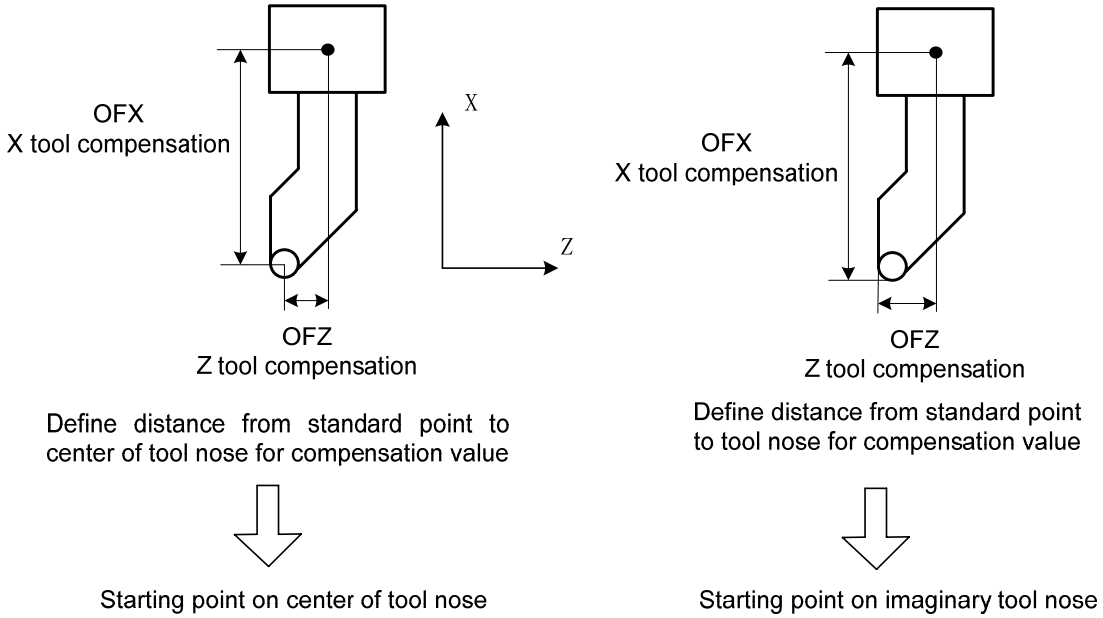


Fig. 4-10 Tool offset value of tool post center as benchmark

4.1.4 G40/G41/G42 Command function

Taking the previous and the current position increment as the programmed path can cancel the tool compensation mode, and its direction is the compensation direction of the previous. When the system specifies(I, J),(I, K)or(J, K), the vector defined by it can replace the current position increment to execute the count.

Command format:

$$\begin{Bmatrix} G17 \\ G18 \\ G19 \end{Bmatrix} \begin{Bmatrix} G41 \\ G42 \end{Bmatrix} \begin{Bmatrix} G00 \\ G01 \end{Bmatrix} Xp\_Yp\_Zp\_$$

In machining workpiece, the tool offset cannot easily compensate the precise workpiece because of the tool nose circle degree but the tool nose radius compensation function can automatically compensate the error.

G40 Xp Yp Zp I J K

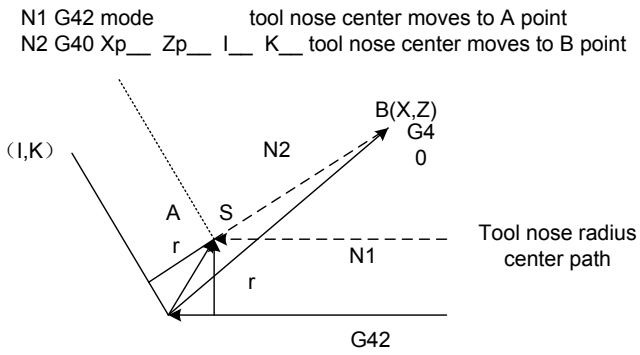


Fig. 4-11 G40 execution process

Command explanation:

Table 4-1

Codes	Function specifications	Remark
G40	Cancel the tool nose radius compensation	See Fig.4-11, Fig.4-12, Fig.4-13
G41	Tool nose radius left compensation is specified by G41 in rear tool post coordinate system and tool nose radius right compensation is specified by G41 in front tool post coordinate system	
G42	Tool nose radius right compensation is specified by G42 in rear tool post coordinate system and tool nose radius left compensation is specified by G42 in front tool post coordinate system	
Xp	X and its parallel axis	
Yp	Y and its parallel axis	
Zp	Z and its parallel axis	
I	X and its parallel axis' cancellation vector (radius value), their value range are the same those of X，and their values are positive and their directions are in X's positive direction, their value are negative and their directions are in X's negative direction.	
J	Y and its parallel axis' cancellation vector (radius value), their value range are the same those of Y，and their values are positive and their directions are in Y's positive direction, their value are negative and their directions are in Y's negative direction.	
K	Z and its parallel axis' cancellation vector (radius value), their value range are the same those of Z，and their values are positive and their directions are in Z's positive direction, their value are negative and their directions are in Z's negative direction.	

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. 4-12 and Fig.4-13.

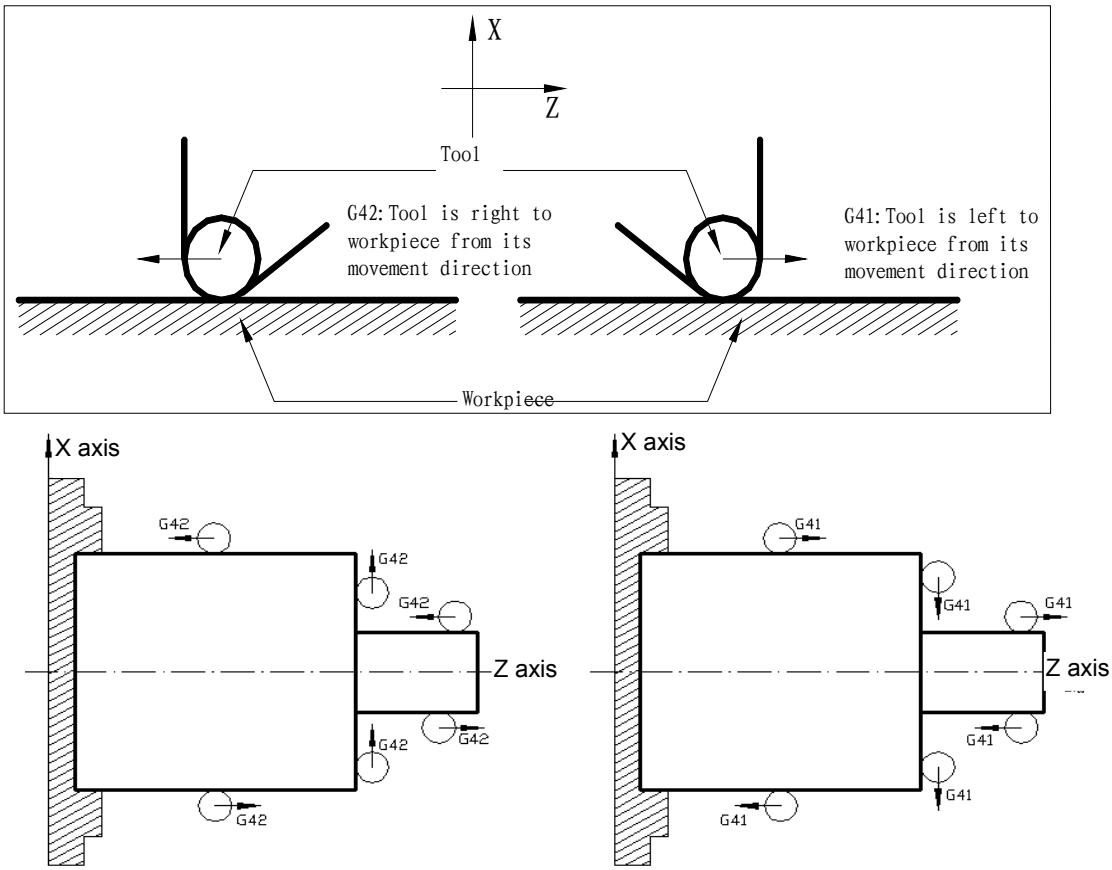


Fig. 4-12 Compensation direction of rear coordinate system

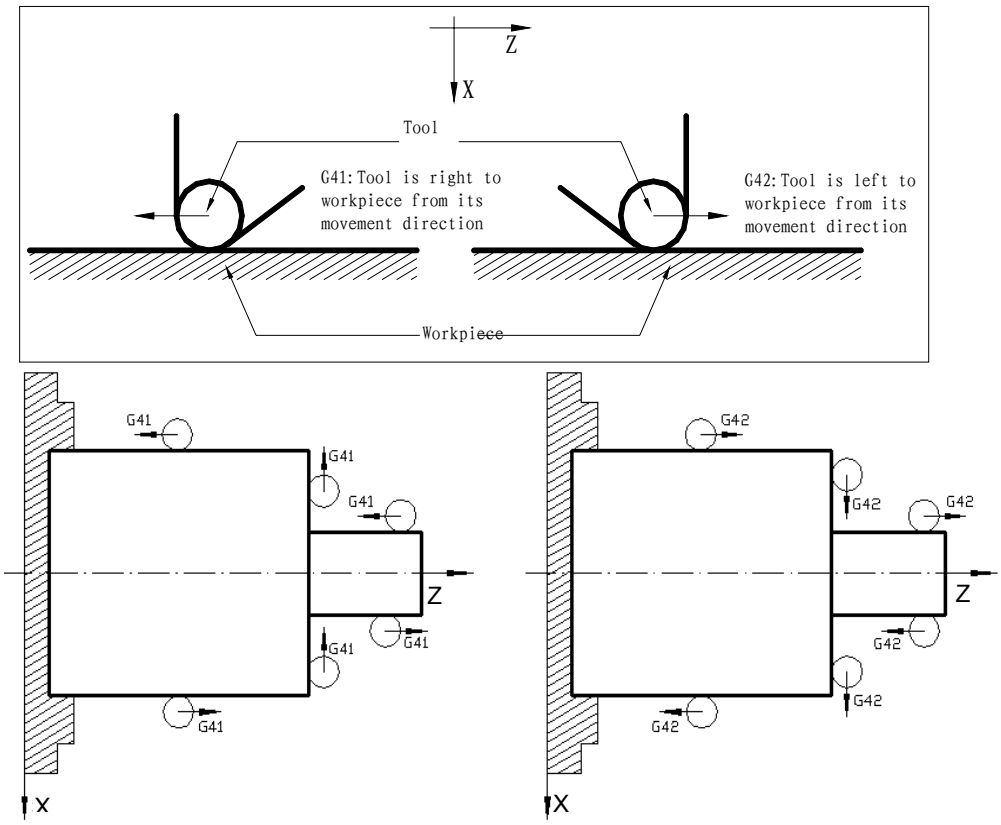


Fig. 4-13 Compensation direction of front coordinate system

4.1.6 Notes

Note 1: In initial state, when the system is in the tool nose radius compensation cancel mode, and the offset compensation number is not 0 in G41 or G42, the system starts creating the tool nose radius compensation offset mode; when the offset compensation number is 0, G modal is the G40 state.

Note 2: In creating or cancelling tool compensation, the workpiece machining must not be executed, otherwise, it causes the overcut or undercut. The system takes the created first movement and the last movement code before being cancelled as the cutting code in normally machining workpiece.

Note 3: The tool does not create the offset and starts compensation in the next movement code when there is no movement code in creating the tool compensation. When there is no movement code in cancelling tool compensation, the tool does not create the offset and the system cancels the compensation vector in the next movement code.

Note 4: The next block to create the tool compensation block has the tool compensation cancel modal code, the system does not execute the tool compensation creation process, but at the moment, the modal code will change normally.

Note 5: I/J/K/R is not specified in G02/G03 and #3403.5=0 (in circular interpolation, I, J, K is not specified, and R is not done, the linear moves to the end point), the system establishes the tool radius compensation. In other cases, the tool nose radius compensation creation and cancel only use G00 or G01 instead of G02 or G03. When it is specified, No.252 alarm occurs.

Note 6: In tool nose radius compensation, the tool nose center moves to the end point of the last block and is vertical with the programmed path of the last when the system executes 3 or more than 3 blocks without movement code. At the moment, the overcut or undercut creates and the system should not machine the workpiece in the next block in programming. When 3 or more than 3 blocks without movement code following the movement code to create the tool nose radius compensation, the system does not create immediately the tool nose radius compensation but does it after the non-movement code.

Note 7: The system does not execute the tool nose radius compensation in G50, G52, G32, G34, G92, G71, G72, G73, G74, G75, G76, G83-G85, G87-G89 and temporarily cancels the compensation mode. Before the system temporarily cancels the compensation execution and when the system modal is G02 or G03, No.262 alarms.

Note 8: In G40, for the inner or outer machining, the system moves to the intersection of two paths, and executes the tool nose radius compensation cancel here, and then moves to the target point after the cancel. When there is no intersection and the tool reaches the normal line position of the end point of the last block, the system cancels the tool nose radius compensation and then moves the target point after the cancel. At the moment, the overcut creates, the workpiece must not be machined.

Note 9: In tool nose radius compensation mode, the system must not be switched to other levels, otherwise,

No.253 alarm occurs.

Note 10: In tool nose radius compensation mode, when RESET key is pressed or M30/M02 is executed, the CNC cancels the tool compensation mode. When the CNC does not code G40 (cancel radius compensation) to execute M30/M02 (end of program), the tool nose center moves to the end point of the previous movement block and is perpendicular to the block's programmed path position. When M30/M02 has movement codes, the CNC normally cancels the radius compensation.

Note 11: In MDI mode, the system cannot execute the tool nose radius compensation creation and its cancel. When the system specifies the tool nose radius compensation code, it executes the code according to No.5008 Bit4(MCR). When the parameter is set to 1, an alarm occurs.

4.1.7 Application

Machine a workpiece in the front tool post coordinate system as Fig. 4-14. Tool number: T0101, tool nose radius R=2, imaginary tool nose number T=3.

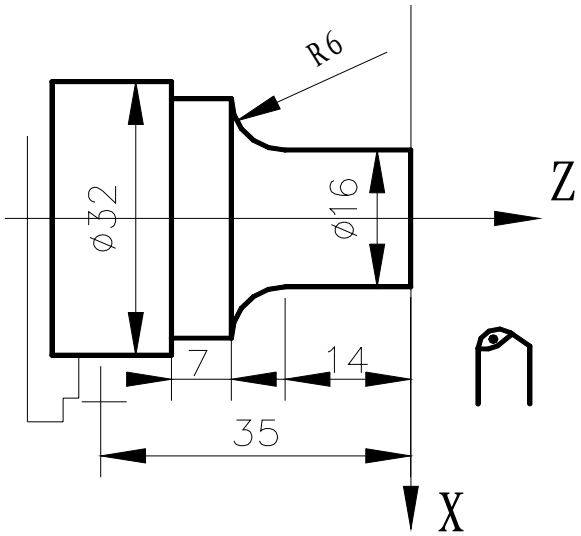


Fig. 4-14

Set the correct imaginary tool nose direction for executing the toolsetting in offset cancel mode, Set the tool nose radius R and imaginary tool nose direction in **TOOL OFFSET & WEAR** window as following:

Table 3-7

No	X	Z	R	T
001			2.000	3
002	…	…	…	…
…	…	…	…	…
007	…	…	…	…

Program:

G00 X100 Z50 M3 T0101 S600;（Position, start the spindle, execute the tool change and the tool compensation）

G42 G00 X0 Z3; （Set the tool nose radius compensation）

```
G01 Z0 F300;                    （Start the cutting）
X16;
Z-14 F200;
G02 X28 W-6 R6;
G01 W-7;
X32;
Z-35;
G40 G00 X90 Z40;                （Cancel the 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°, which is shown 4-15.

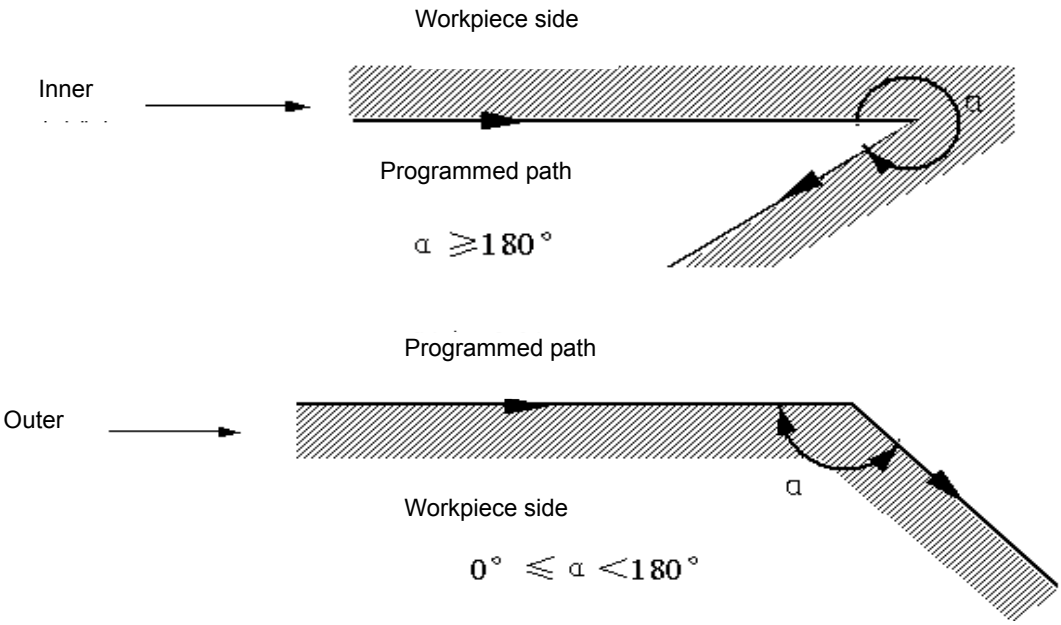


Fig. 4-15

4.2.2 Tool Traversing when Start-up 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 (start-up 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——linear; C——circular, R——tool radius compensation; α——angle between two blocks.

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

1) linear —linear

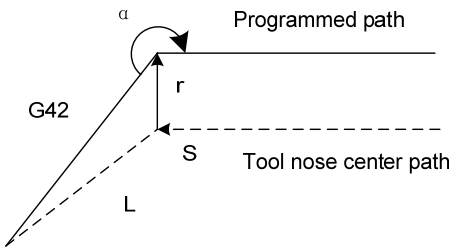


Fig.4-16 Linear —linear(start-up tool inside)

2) linear —circular

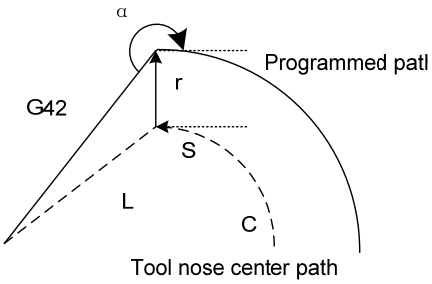


Fig. 4-17 Linear —circular（start-up tool inside）

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

1) linear —linear

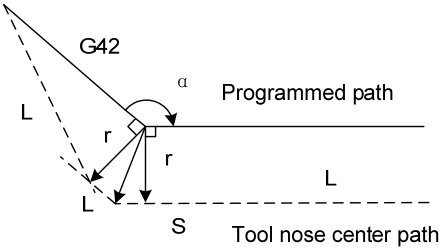


Fig.4-18 Linear —linear(start-up tool outside)

2) linear —circular

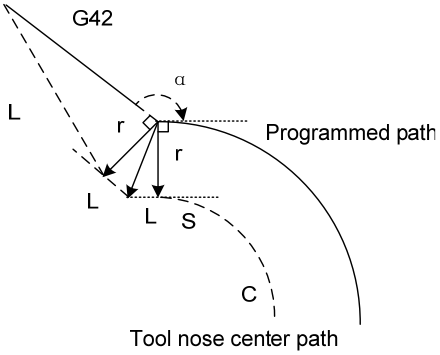


Fig.4-19 Linear—circular(start-up tool outside)

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

1) linear —linear

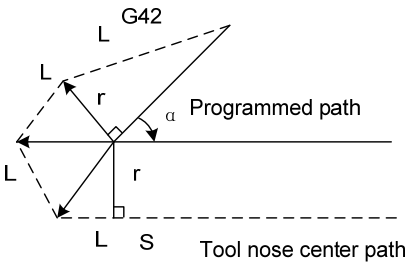


Fig.4-20 Linear —linear（start-up tool outside）

2) linear —circular

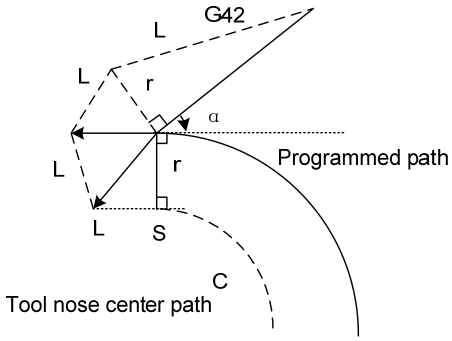


Fig. 4-21 Linear—circular (start-up tool outside）

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

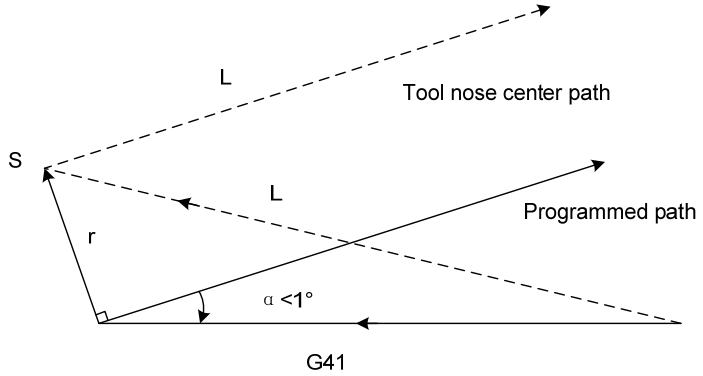


Fig. 4-22 Linear—linear（α＜1°, start-up tool outside）

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) linear —linear

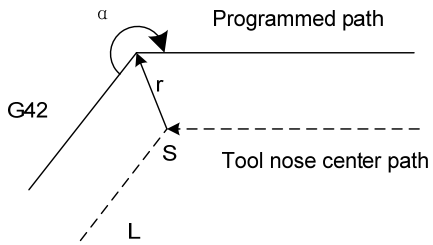


Fig. 4-23 linear—linear（moving inside）

2) linear —circular

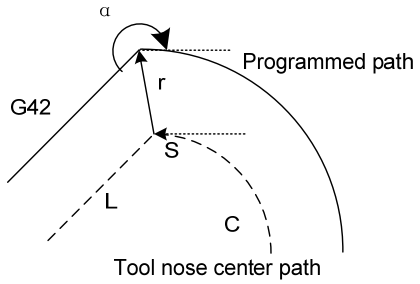


Fig. 4-24 linear—circular（moving inside）

3) circular—linear

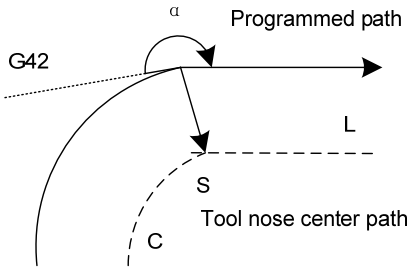


Fig. 4-25 Circular—linear（moving inside）

4) circular —circular

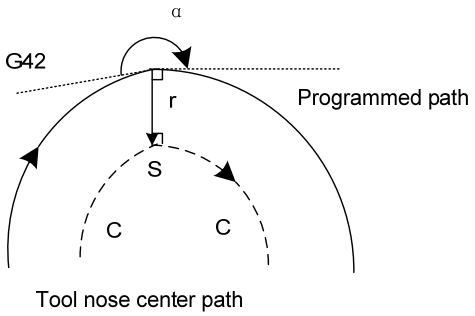


Fig. 4-26 Circular—circular（moving inside）

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

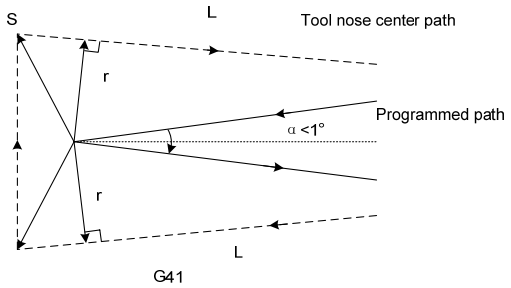


Fig. 4-27 Linear —linear（α＜1°, moving inside）

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

1) linear —linear

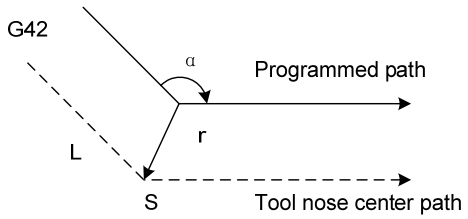


Fig. 4-28 Linear —linear（moving outside）

2) linear —circular

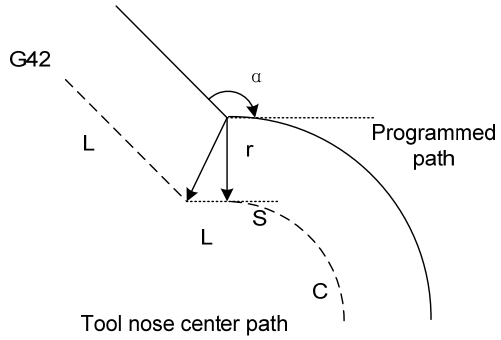


Fig. 29 Linear—circular（moving outside）

3) circular—linear

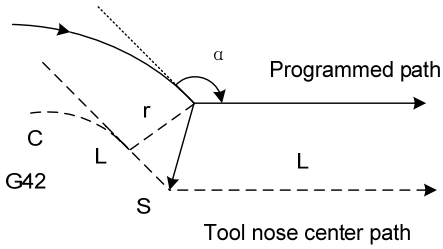


Fig.4-30 circular—linear（moving outside）

4)circular —circular

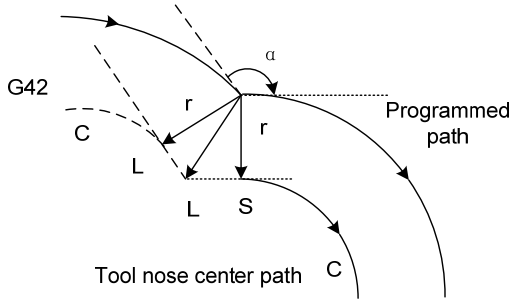


Fig.4- 31 circular—circular（moving outside）

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

1) linear —linear

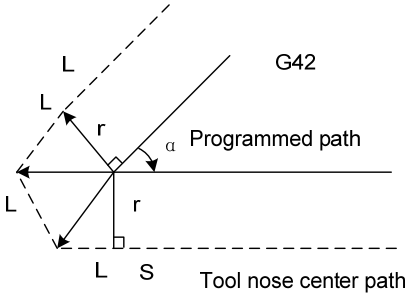


Fig. 4-32 Linear—Linea（moving outside）

2) linear —circular

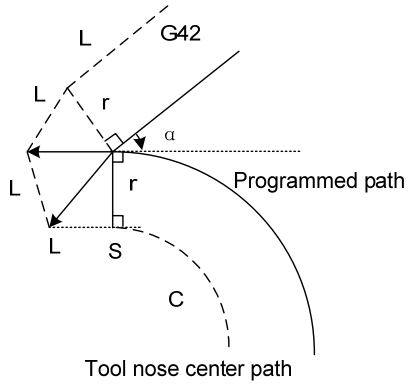


Fig. 4-33 Linear—circular（moving outside）

3)circular—linear

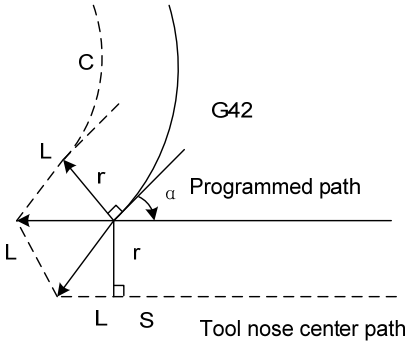


Fig.4-34 Circular—linear（moving outside ）

4) circular—circular

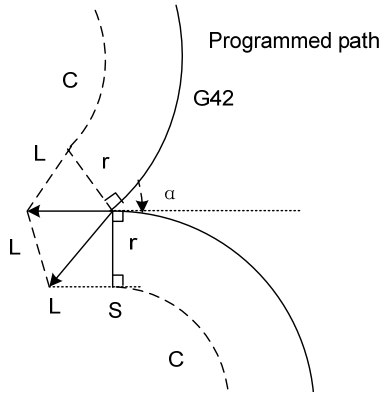


Fig.4-35 Circular—circular（moving outside ）

(d) Special cutting

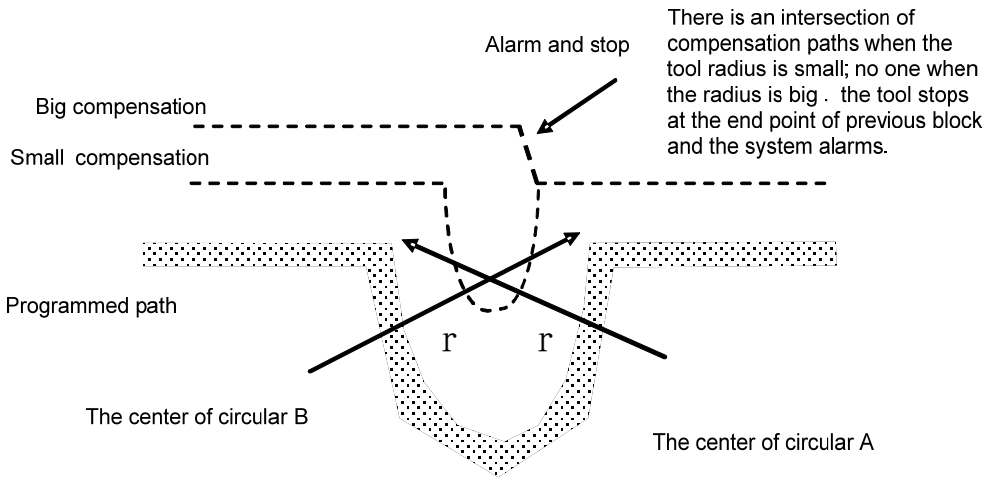


Fig. 4-36 Paths without intersection after offset

2）Center point and starting point of circular being the same one

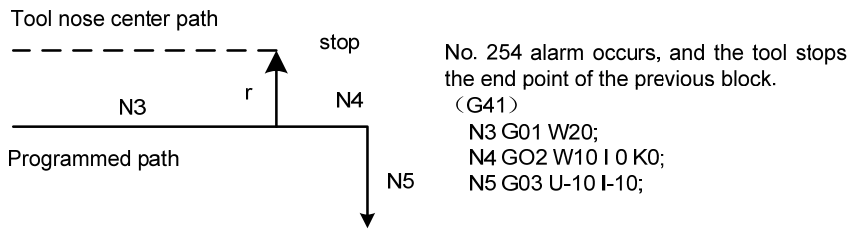


Fig. 4-37 Center point and starting point of circular being the same one

- **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 4-2

Compensation value sign / G	+	-
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)linear—linear

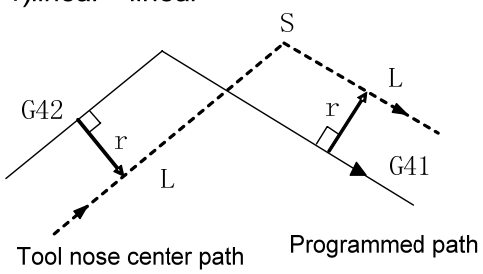


Fig. 4-38 Linear—linear
（changing compensation direction）

2) linear—circular

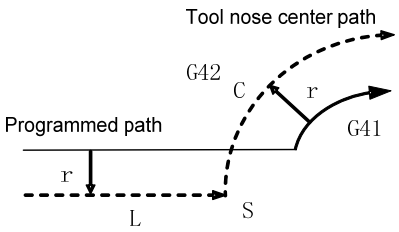


Fig. 4-39 Linear—circular
（changing compensation direction）

3)circular—linear

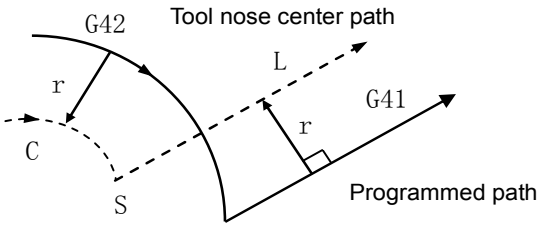


Fig. 4-40 circular—linear（changing compensation direction）

4) circular—circular

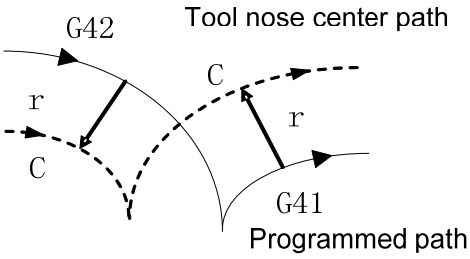


Fig. 4-41 circular—circular
（changing compensation direction）

5）The compensation is executed normally without an intersection point

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) Linear----Linear

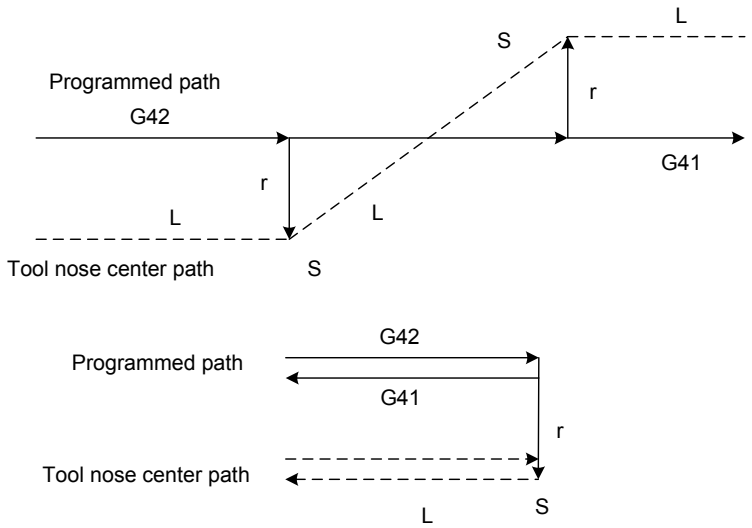


Fig. 4-42 Linear—linear, no intersection（changing compensation direction）

ii) Linear ---circular

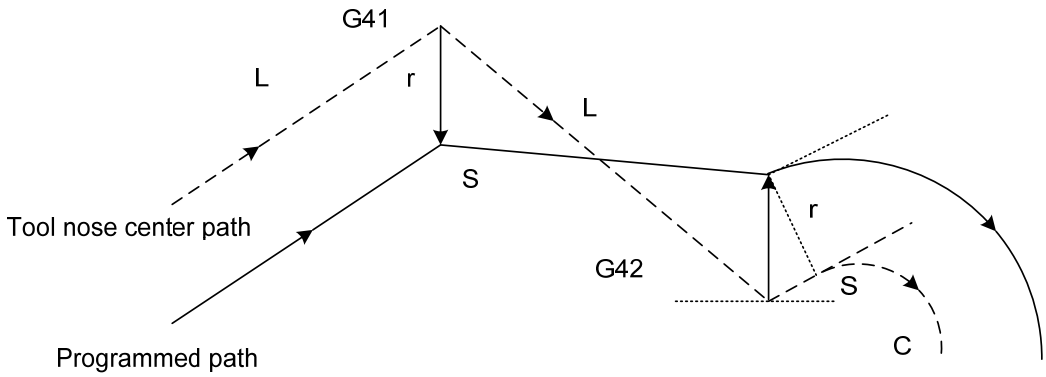


Fig. 4-43 Linear—circular without intersection（changing compensation direction）

iii) Circular-----circular

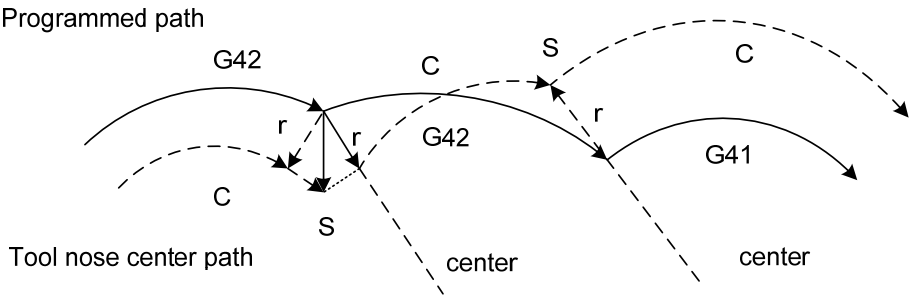


Fig. 4-44 Circular—circular without intersection (changing compensation direction)

4.2.4 Tool Traversing in Offset Canceling Mode

In compensation mode, when the system executes G04, it enters the compensation canceling mode, which is defined to compensation canceling of block. The system cannot execute the circular code(G02 or G03) in canceling tool compensation mode, otherwise the system alarms and stops run.

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

1) linear —linear

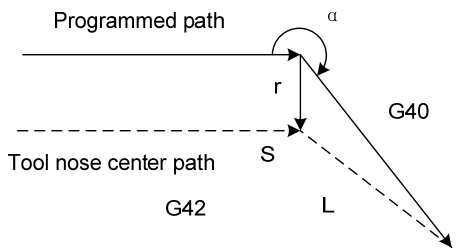


Fig. 4-45 Circular-linear (moving inner and canceling offset)

2) circular—linear

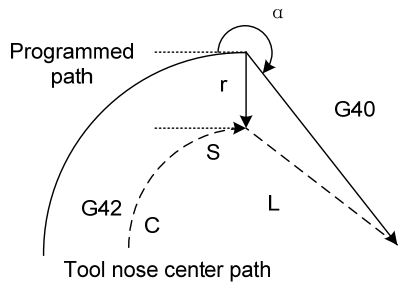


Fig. 4-46 Circular-linear (moving inner and canceling offset)

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

1) linear —linear

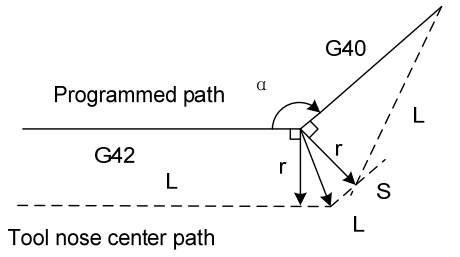


Fig. 4-47 Circular—linear（moving outside and canceling offset）

2) circular—linear

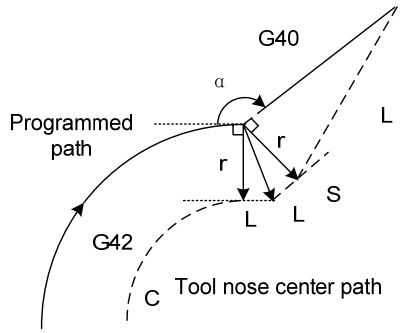


Fig. 4-48 Circular—linear（moving outside and canceling offset）

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

1) linear —linear

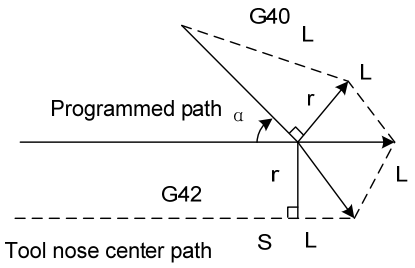


Fig. 4-49 Linear—linear (cutting outside and canceling offset)

2) circular —linear

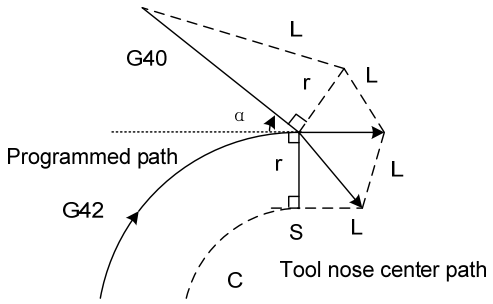


Fig. 4-50 Linear—linear (cutting outside and canceling offset)

(d) Tool traversing outside along corner(α＜1°) ; linear→ linear

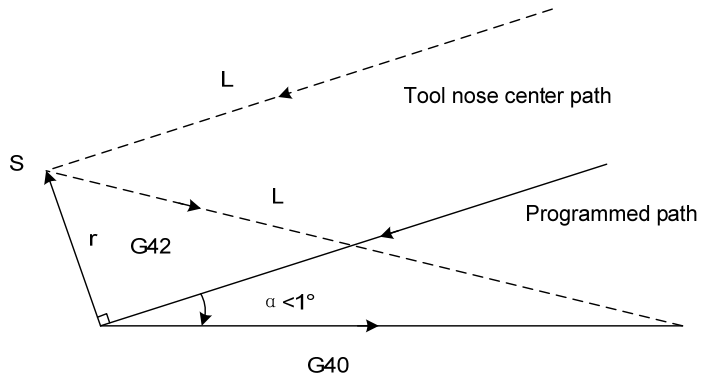


Fig. 4-51 Linear—linear (α＜1°cutting outside and canceling offset)

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, at the moment, No.257 alarm occurs (the angle is 90°~270°). The alarm does not occur when No.5008 Bit1 (CNC) is set to 1.
2) In machining arc, there is great difference the two angles(α＞180°), the one is between the starting point and the end point of the tool center path, and the other is between the starting point and the end point of the programmed path, or the system cuts the inner of the arc (α＞180°) , and the tool cannot pass the entrance, No.256 alarm occurs.

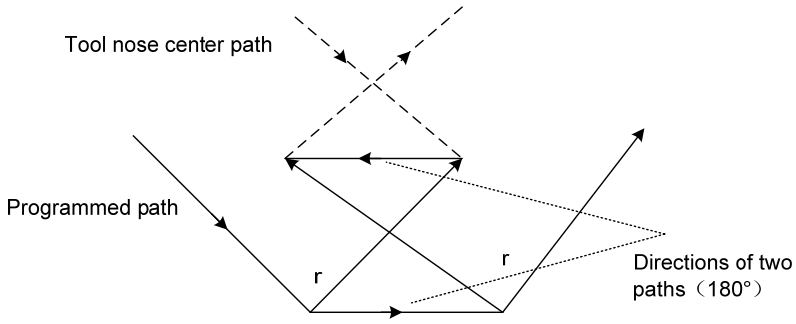
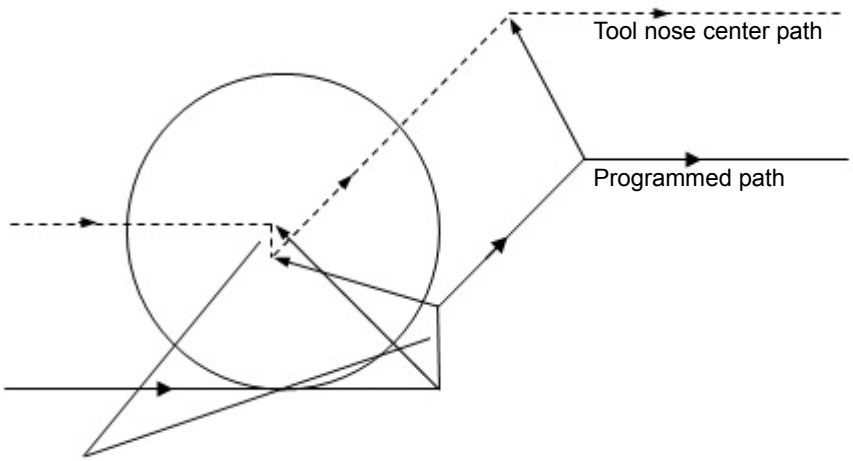


Fig. 4-52 Machining interference, No.257 alarm occurs



Direction difference of two paths being big（180°）

Fig. 4-53 No. 260 alarm occurs when machining interference

(2) Executing it without actual interference

1) Concave groove less than compensation value

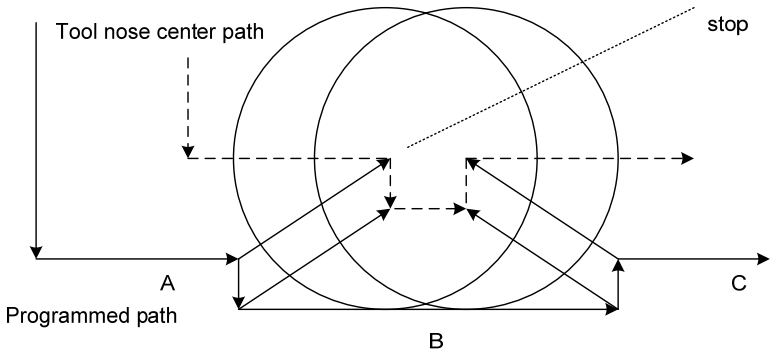


Fig. 4-54 Executing interference (1)

Actually, without interference, but there are an excess of workpieces not be machined in the interior angle, the tool stops and No. 260 alarm occurs.

2) Concave channel less than compensation value

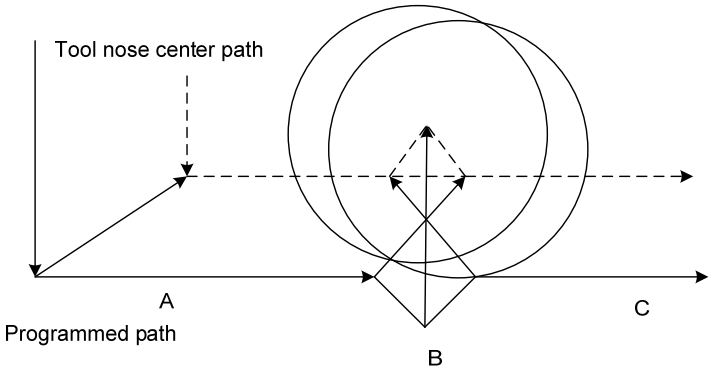


Fig. 4-55 Executing interference (2)

Directions of block B and tool nose radius compensation path are opposite without interference, the tools stops and No.257 alarm occurs.

(3) Automatic interference vector clear

The system has the automatic interference vector clear function. For example, when the neighbor three blocks N10, N20, N30 execute the tool radius compensation, the section between N10 and N20 creates the vector V1, V2, V3 and V4, and the section between N20 and N30 creates V5, V6, V7, V8. The system executes the interference check to the last vectors in the above two group of vector, i.e. V4 and V5. V4 and V5 are ignored when there is the interference; the system checks V3 and V6, and they are ignored when there is the interference; the system does V2 and V7, and they are ignored when there is the interference. When the system executes the interference check to the last vectors V1 and V8, and there is the interference, they cannot be ignored, the tool stops movement and the system alarms. Based on the above process, the system executes the interference check, and has checked the vector which is not interfered, the followings are not check, and the tool runs according to the path of the first group vector which does not create the interference. When the last group of vector creates the vector, they cannot be ignored, the tool stops movement and No.257 alarms.

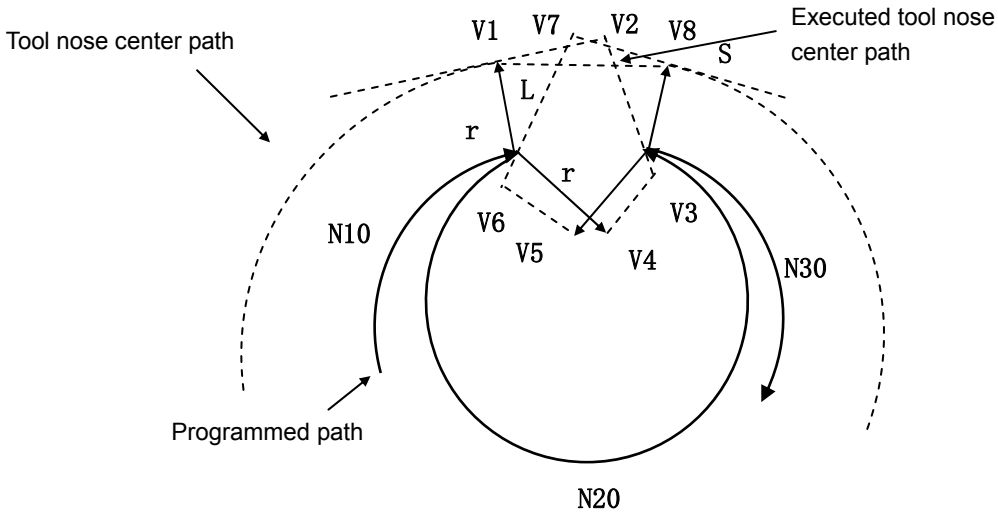


Fig. 4-56 interference vector clear

Note 1: NO.5008 Bit 0（CNI）can set whether the interference check is executed in tool nose radius compensation mode.

Note 2: NO.5008 Bit 1（CNC）can set whether the system alarms when the difference 90°-270° between the movement direction and offset direction.

Note 3: NO.5008 Bit 3（CNV）can set whether the system executes the interference check and the vector clear.

4.2.6 Codes for Canceling Compensation Vector Temporarily

In compensation mode, when the system specifies G28, G30, G50, G52, G32, G34, the fixed cycle (including rigid tapping), multi cycle, drilling cycle code, the compensation vector is cancelled temporarily and is automatically resumed after executing the codes. 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.

Note: do not machine workpieces in the movement block prior to the code for temporarily canceling radius compensation, otherwise cause overcut or downcut of workpieces when canceling or creating compensation.

- **Setting coordinate system in G50, G52**

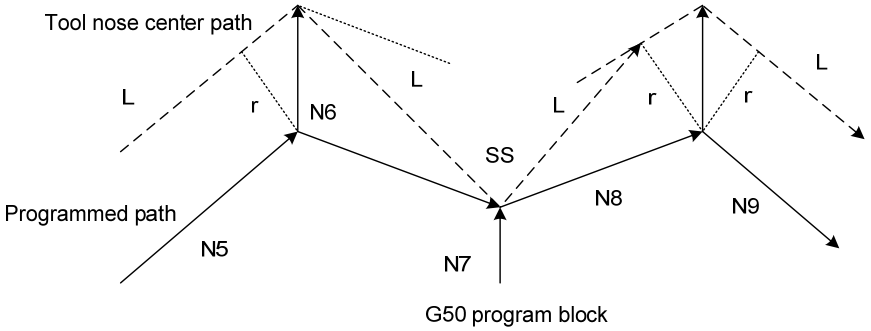


Fig. 4-57 Temporary compensation vector in G50, G52

Note: SS indicates a point at which the tool stops twice in Single mode.

- **Reference position automatic return G28, G30**

In compensation mode, the compensation is cancelled in a middle point and is automatically resumed after executing the reference position return when G28/G30 is executed. Refer to #5003.2.

Note: when the middle point is the same with the end point of the previous movement code, the system defaults to cancel the compensation mode when the tool traverses to the reference point regardless of #5003.2 value.

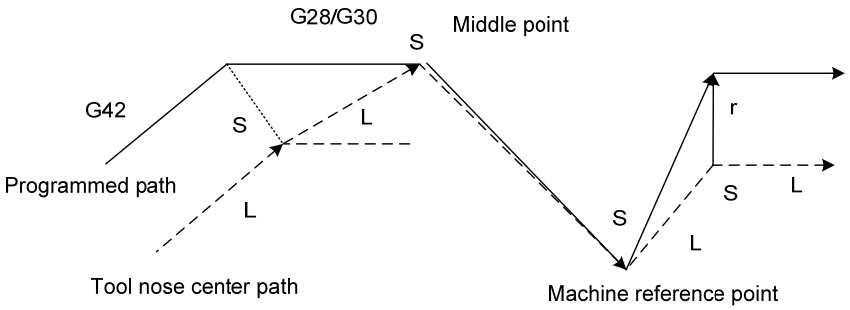


Fig. 4-58 Cancel compensation vector temporarily in G28, G30

- **G53 automatic return to reference position**

In compensation mode, when G53 is executed, the system creates the offset vector which is vertical with the tool motion direction before the end point of the last block. When the tool moves to G53 position, the compensation vector is cancelled. The compensation vector is automatically recovered when the system executes the next movement code.

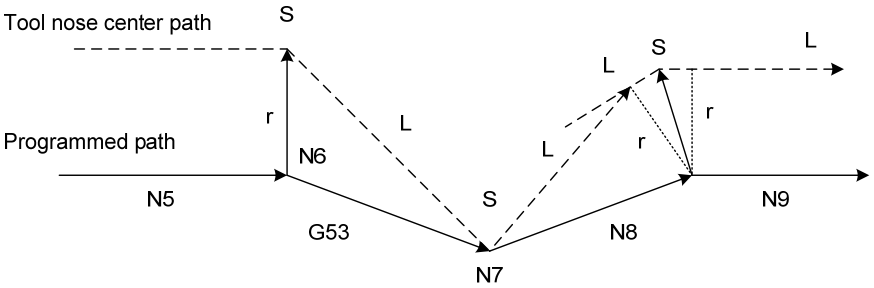


Fig. 4-59 G53 temporarily cancelling compensation vector

- **G71～G76 compound cycle; G92 fixed cycle, G84, G88 drilling cycle**

When executing G71～G76 , G92 fixed cycle, G84, G88 drilling cycle, the system does not execute the tool nose radius compensation and cancel it temporarily, and executes it in the next blocks of
G00, G01, G70, CNC automatically recovers the compensation mode.

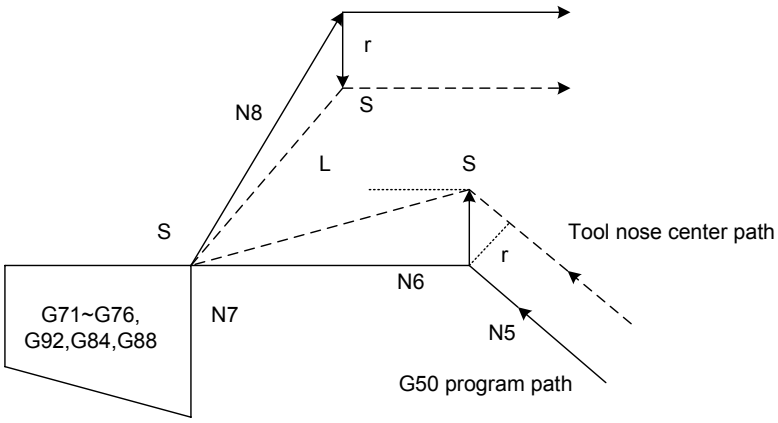


Fig. 4-60 Cancel compensation vector temporarily in cycle pause

- **G32, G34 thread cutting**

The system does not execute the tool nose radius compensation and temporarily cancels the tool nose radius compensation in G32, G34, and it automatically recovers the compensation mode in G00, G01.

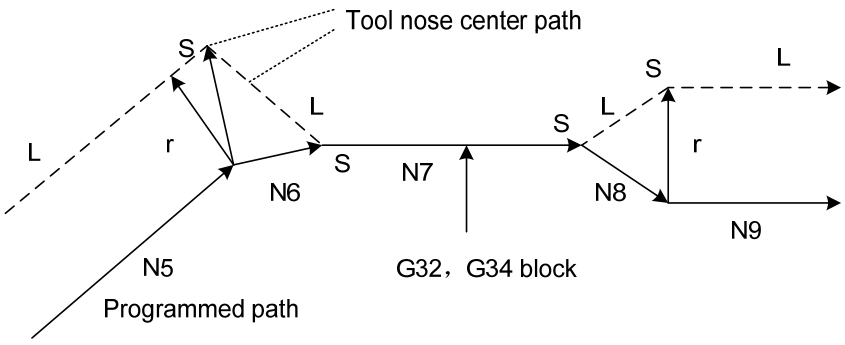


Fig.4-61 cancelling compensation vector in G32, G34 pause

- **G90, G94**

Compensation method of the 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, and the system determines the tool compensation direction according to the UW direction of starting point and end point, and executes the tool compensation according to the direction in the cycle process.
C. In having creating C tool compensation state, the system firstly cancels C tool compensation state in G90, G94, and executes the infeed tool to the intersection point of the tool nose center based on the tool nose center parallel programmed path, and at last to the positioning point. The system creates C tool compensation again in the next G00, G01.

D. After the system cancels the tool radius compensation, the imaginary tool nose point moves to the positioning point, and when the tool is in the cycle inner, the tool diameter exceeds the length of the rapid traverse of the first block, the overcut creates and No.255 alarms.

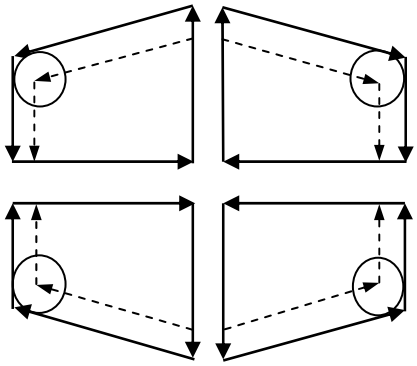


Fig. 4-62 Offset direction of tool nose radius compensation in G90

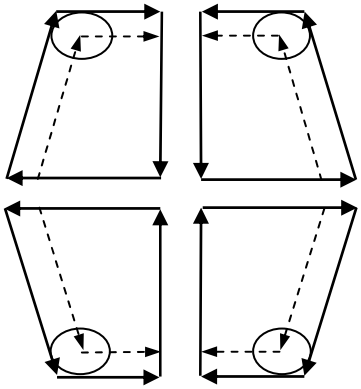


Fig. 4-63 Offset direction of tool nose radius compensation in G94

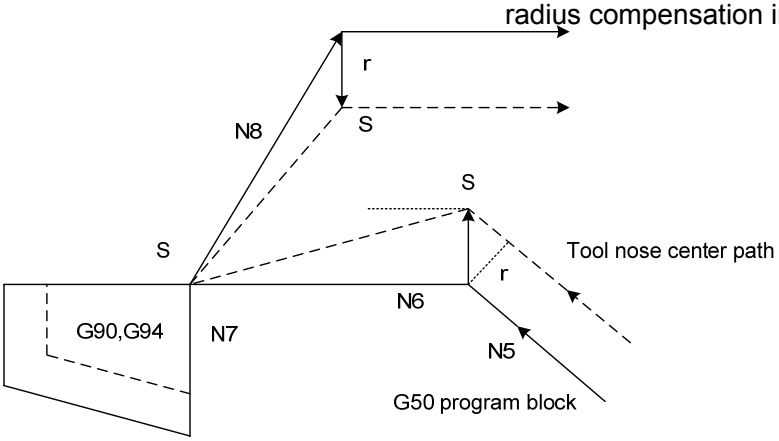


Fig. 4-64 G90, G94 radius compensation mode

- **G70**

When G71～G73 is executed, the system temporarily cancels C tool compensation. When G70 is specified again, the system automatically recovers the compensation mode. Because the system executes G71～G73, it does not execute the radius compensation, there must be the finishing allowance in programming to avoid the overcut in roughing.

When G70 is cancelled, the radius compensation is cancelled firstly at the positioning point, a radius compensation is created at the first block of the cycle body, and a compensation is performed at the middle, and the radius compensation is cancelled in the course of returning to the positioning point. When G70 cycle is executed, when the compensation mode is not cancelled after the cycle ends, the system continuously executes the compensation in the positioning point, which causes the undercut of the finishing cycle in the last block, so, In G70 cycle, the last block should exceed one tool radius value of the workpiece in programming.

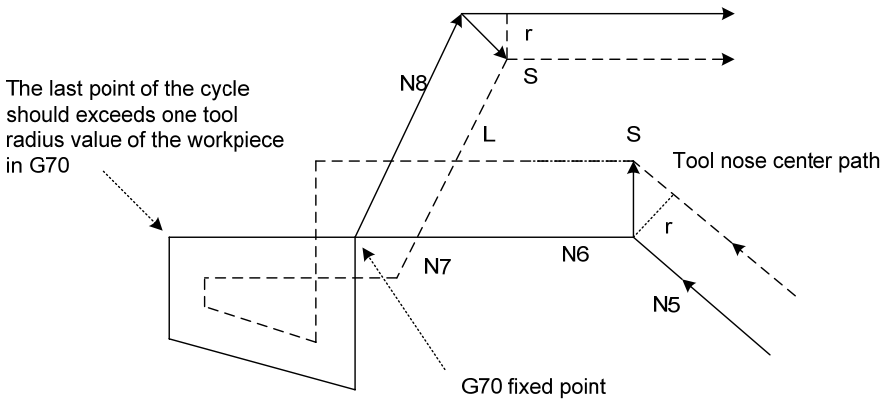


Fig. 4-65 G70 radius compensation mode

4.2.7 Particulars

- **Inside chamfer machining less than tool nose radius**

At the moment, the tool inside offset causes an excessive cutting. The tool stops and No.261 alarm occurs when starting the previous block or chamfer moving. But the tool stops the end point of previous block when **Single** is ON.

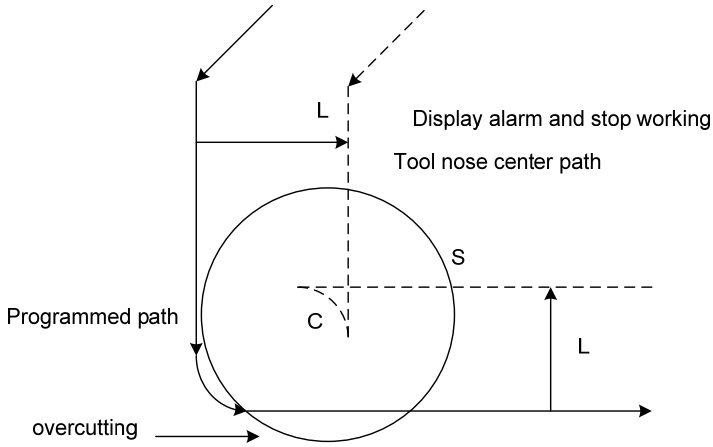


Fig.4-66 inner corner machining less than tool nose radius

- **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 No.257 alarm occurs when starting the previous block or chamfer moving.

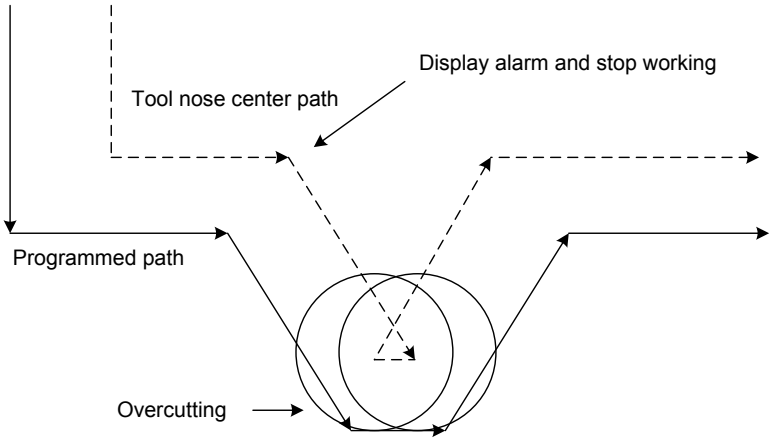


Fig. 4-67 machining a grooving less than tool nose radius

- **Machining a inner sidestep less than 90°**

When the system machines a inner sidestep less than or equal to 90°and the machining path length is less than the tool nose radius, there will be the too much undercut and No. 260 alarm occurs. At the moment, No.5008 Bit6 (CNS) sets whether the system alarms in the condition.

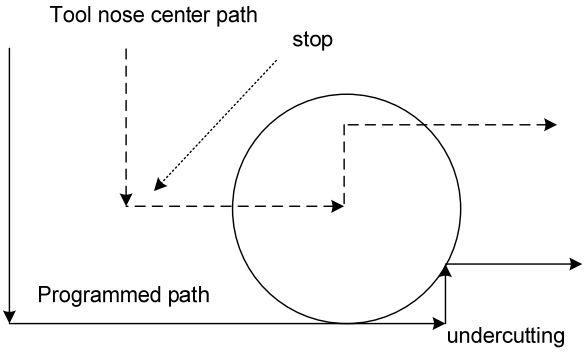


Fig.4-68 machining an inner sidestep less than 90°

- **Corner motion**

When two or more than movement vector in the end point of one block create, the tool moves to another vector from the vector linear, which is called the corner motion. When the single block is valid, the tool stops in the last vector.

When two vectors coincide, the system does not execute the corner motion and the second vector will be ignored. When the two-axis increments of the movement vector in the compensation level are less than the setting values of No. 5010(CLV), the second vector is ignored, but it is not ignored when the interpolation block is the arc.

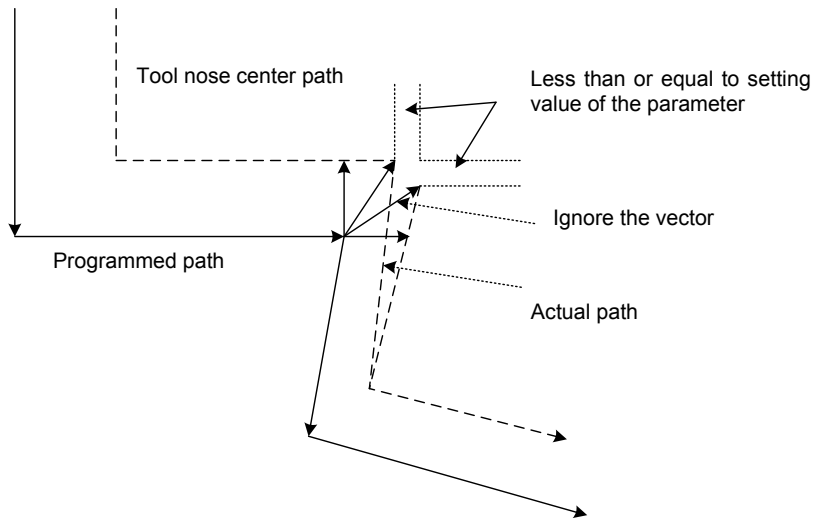


Fig. 4-69 corner motion

- **Changing compensation value**

(a) The system executes the tool change in the compensation cancel mode, the compensation value is changed. When the compensation value is changed in the compensation mode, No.5001 Bit4(EVR) can set whether the compensation value change is valid from the nest T code or the next buffer block.

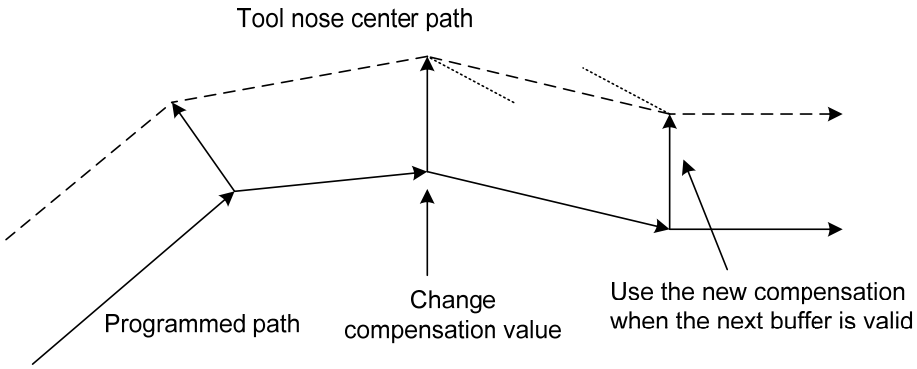


Fig.4-70 changing compensation value

(b) Positive/negative compensation value and tool nose center path
When the compensation value is negative (-), G41 and G42 exchange in programming. When the tool center moves along the workpiece outer, it moves along the inner, and vice versa.

Note: The compensation value is equal to the offset value adding the wear value. When the compensation sign is changed, the tool nose offset direction changes but the imaginary tool nose direction does not change. So, do not change the compensation sign optionally.

- **End point of programmed arc is not in the arc**

In the radius compensation process, when the system uses IJK to specify the circle center and the end point of the arc is not in the arc, the system positions again the circle center position

specified by IJK, and confirms the circle center position according to the radius counted by IJK to execute the radius compensation. When the counted radius is too small not to reach the end point of the arc, No. 254 alarm occurs.

Note: At the moment, there is a difference between the counted arc and the specified in programming, and the function is sued to regulate the error of the radius out-of-tolerance in some range in programming.

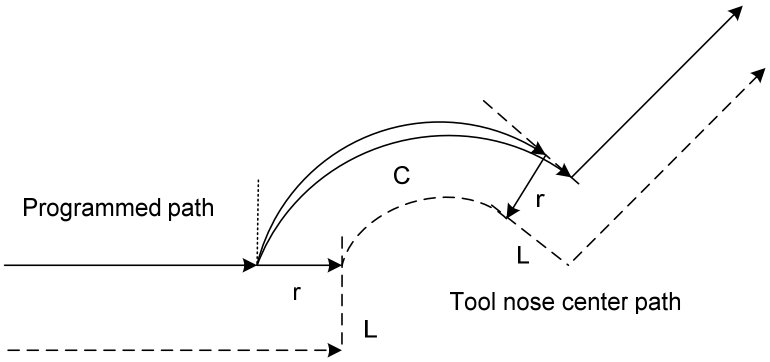


Fig. 4-71 End point of programmed arc be not in the arc

- **Continuous 3 or more than 3 blocks non-movement code**

In tool radius compensation process, when there are 3 or 3 blocks without movement code, the tool nose center reaches the end point of the last block and is vertical with programmed path position of the last, at the moment, which causes the overcut, and the programmer should pay more attention it.

General non-movement code ：

1. M03S300 only have M, S, T, F, O, N codes
2. #100=3 non-NC statement （when 6000#5 SBM is set to 1）
3. G04 X10 pause
4. G00 only have G code and do not specify the position code
5. G01 U0 the infeed distance is 0
6. G01X100 only specify the absolute value which is same with that of the last block
7. G01Y10 only specify the axis in non tool compensation level
8. M98M99 statement for calling subprogram and subprogram return(the block has no axis increment code）
9. G66G67 statement for calling macro program and cancelling macro program call modal
10. ; null block

In non-movement block, when there is a code to cancel the radius compensation, the system does not cancel the vector and execute the code in the vertical vector. It cancels the radius compensation vector when the system cancels the radius compensation in G28, G30, G53, it executes the code in the vertical vector in G50, G52, G32, G34, fixed cycle, multi cycle, drilling cycle and other codes.

When there are 3 or more than 3 blocks without movement code following the block used to create the tool radius compensation, the system does not immediately create the tool radius compensation but does it in the block following the non-movement code.

The system executes the above vertical before the last movement code when there is a optional symbol “/” in tool radius compensation. So, please do not use the optional block function in the tool radius compensation to avoid the overcut.

When No.6000 Bit5 (SBM) is set to 1, the macro statement can stop in single block and is taken as the non-movement block in the tool nose radius compensation at the moment, which causes the abnormal path. It is suggested that No.6000 Bit5 (SBM) is set to 0 when the system uses the macro statement in the tool nose radius compensation mode in the course of normal machining.

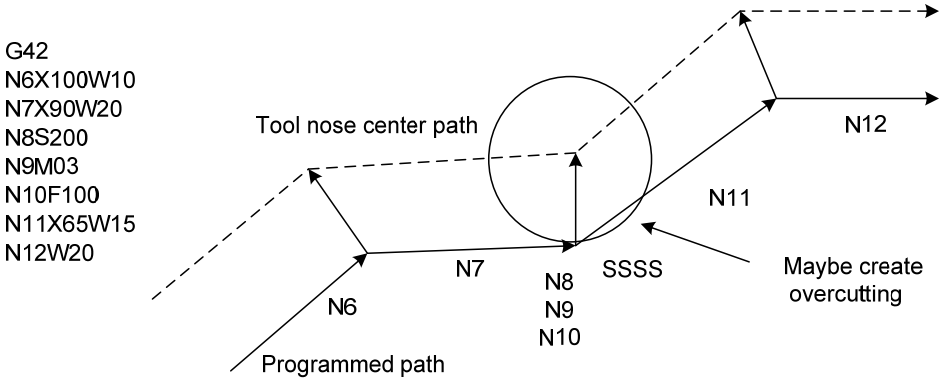


Fig. 4-72 continuous 3 or more than 3 blocks of non-movement code

- **Subprogram call and macro statement call in G code**

In tool nose radius compensation mode, when the system specifies the code for calling the subprogram, it can execute the normal compensation, the compensation method of calling program is transferred to the subprogram which is to execute the corresponding compensation. When the system specifies G code in the radius compensation mode in the subprogram, G code is valid, at the same time, the system cancels the radius compensation mode when the subprogram does not end, the compensation mode is transferred to the called program which will continuously executes the corresponding compensation.

The code for calling subprogram and subprogram return has no movement code, it is taken as the non-movement block.

- **Cutting inner of the whole circle**

In the tool nose radius compensation, when the system machines the inner of the whole circle and the compensation direction is not changed, the overcut or undercut creates, at the moment, it determines whether it alarms based on No.5008 Bit5 (CNF). When Bit5 is set to 0, No.259 alarm occurs.

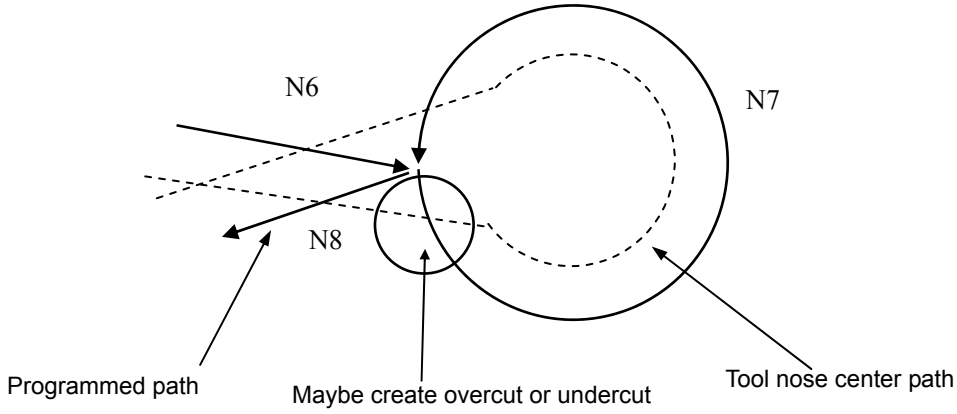


Fig. 4-73 overcut in machining inner of the whole circle

- **Inserting MDI operation in tool compensation**

In MDI mode, the system does not execute the tool nose radius compensation. When the system specifies G41 or G42, the system determines No.5008 Bit4 (MCR). When Bit is set to 1, No.258 alarms. The system does not alarm and ignores the specified G41 and G42 when it is set to 0,
When the system runs in AUTO mode in absolute code programming and the single block run stops to insert MDI mode, and then starts AUTO mode, at the moment, transfers the vector of starting point of the next block, and forms other vectors based on the next two blocks, the offset can be executed from PC, and the tool path is as follows:

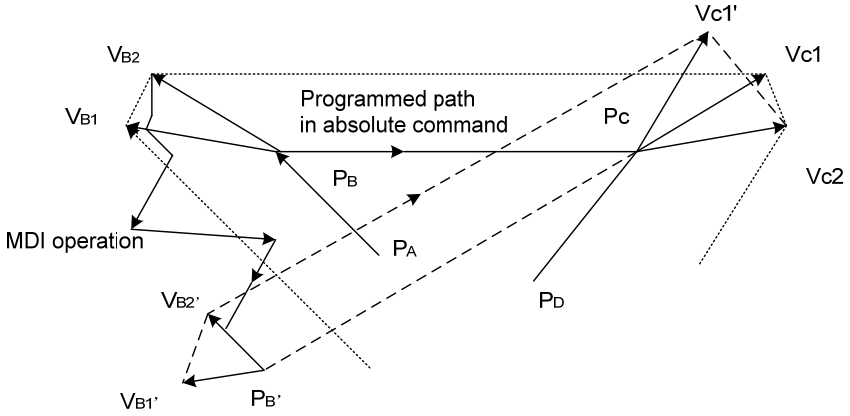


Fig. 4-74 insert tool offset of block in MDI mode

When PA, PB, PC is programmed with absolute code, the single block run stops and the tool is moved in MDI after the block from PA to PB is executed. The vector VB1, VB2 are transferred to VB1' and VB2', VC1' ,VC2 of PB'→PC and PC→PD are calculated again.
But, the system can correctly execute the compensation following PC because the vector VB2 has not calculated again.

OPERATION

Chapter 1 Overview

1.1 Operation Overview

GSK988TA is with the operation modes: Edit, Auto, MDI, Reference position return, MPG/Single step, Manual and DNC operation modes, etc.

- **Editing the program**

The operation is completed with the program editing function, and the edited program is saved in CNC memory, and the program can be corrected and rewritten. (About the details, refer to Chapter Four.)

- **Auto running**

Automatic running is to operate the machine based on the edited program. Once the program is edited into CNC memory, the program can run based on the program codes, and the operation is called running in Auto mode.

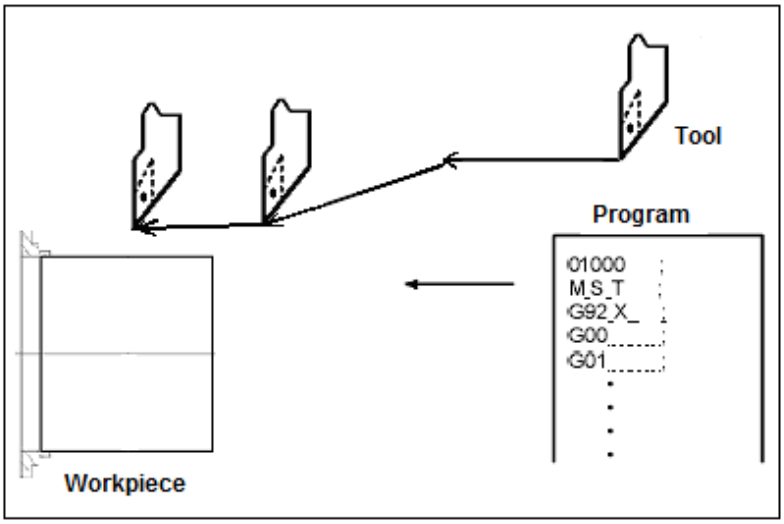


Fig.1-1 Automatic running

- **Manual data input（MDI）mode running:** On MDI page, after inputting the program, the program can run based on the program codes. And the operation is called as running in MDI mode.

- **Reference position return**

A CNC machine tool is provided with a fixed position for setting the position of the machine worktable. And the fixed position is called as the reference position. Normally, tool change or setting the coordinate system is performed at this position. After power on, the tool is moved into the reference position. Manual reference position return is to move the tool into the reference position by the switches and the buttons on the operation panel.

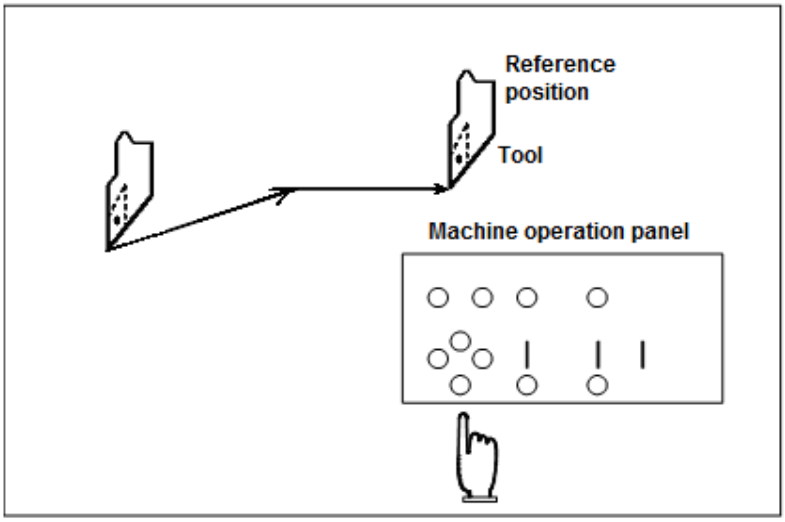


Fig. 1-2 Manual reference position return

Moreover, the tool can be moved into the reference position with the program code, and the method is called as the automatic reference position return.

- **MPG feeding**

The tool can be moved the distance corresponding to the rotation angle by rotating MPG. (See Section 5.4) .

- **Manual operation**

The tool can be moved along each axis by the switches on the machine operation panel, buttons or MPG.

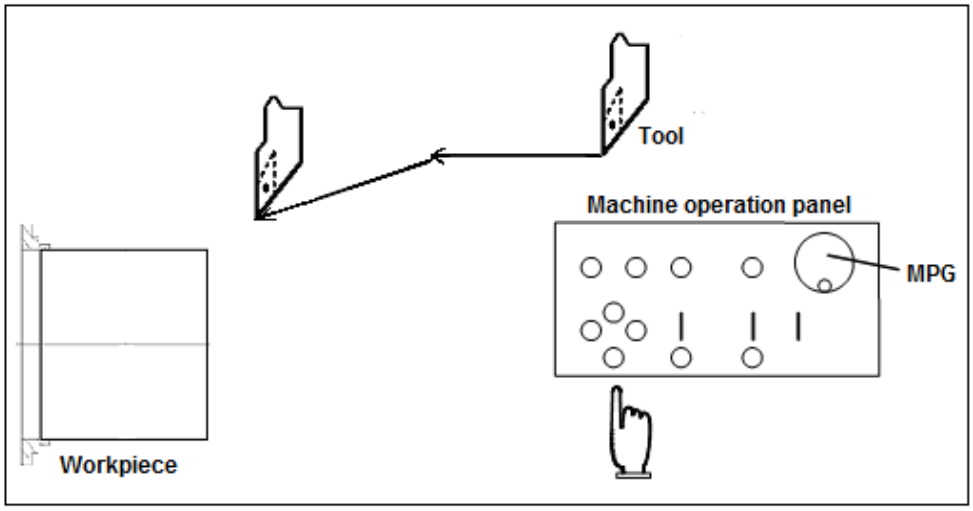


Fig.1-3

（1）Manual (manual continuous) feed (refer to 5.2 for details.)

During the button is pressed, the tool can be continuously moved.

（2）Incremental feed (refer to 5.3 for details.)

After the button is pressed, the tool is only moved for certain distance when the button is pressed once.

- **DNC running:** The program can be directly read from the external input/output equipment for running the machine; however, the program isn't saved into CNC memory, and the operation is called as DNC running (Refer to 6.3 for details.)

1.2 Setting the System

The operator can set CNC by CNC host machine buttons and the normal setting is: the tool offset setting, CNC setting and the macro variable setting.

Tool offset setting: The tool has its own dimension (length, diameter). When the workpiece with certain shape is machined, the tool dimensions vary based on the different movement amounts. If the dimension data of the tools are set in advance, the tool path is automatically given by the same program even the different tools are used, so the workpiece shape specified by the program can be cut by any tool.

The data of the tool dimension are called as the offset amount (Refer to Chapter Seven for details.).

CNC setting: CNC setting includes setting the system, the coordinate, the system time and the system IP.

Macro variable setting: CNC system can support the various macro editing, while the variable required by the macro should be set here.

1.3 Display

Program display:

The content of the program currently being executed is displayed, which is shown in Fig. 1-4.

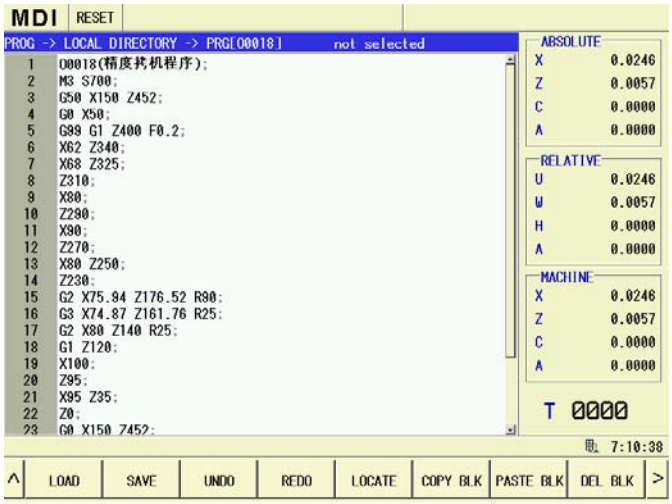


Fig.1-4

Coordinate display

The coordinate value of each coordinate system can be displayed the current tool position, and can be taken as the distance to go from the current position to the target position, which is shown in Fig.1-6. (About details, refer to Chapter 3.1: Position Display Page).

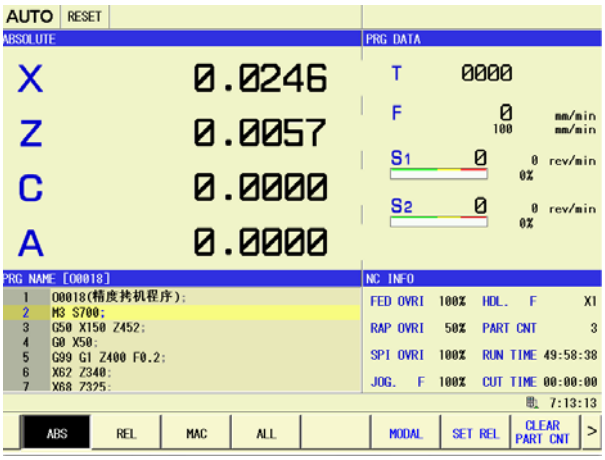


Fig.1-5

Fig.1-6

Alarm display

When the fault occurs during running, the corresponding wrong codes and alarm message will be displayed, which is shown in Fig. 1-6. Please refer to *Appendix One* for the detailed explanation of the alarm message.

Display the machined workpiece number and the operation time

The machined workpiece number, the operation time and the cutting time can be displayed on the current position display page, which is shown in Fig. 1-7:

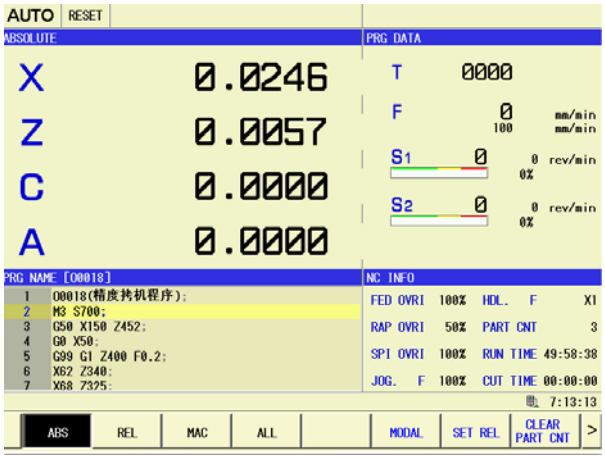


Fig. 1-7

1.4 System Host Machine

1.4.1 System Host Machine Panel

The screen size of GSK988TA is 10.4 inches and its appearance is shown in Fig. 1-8：

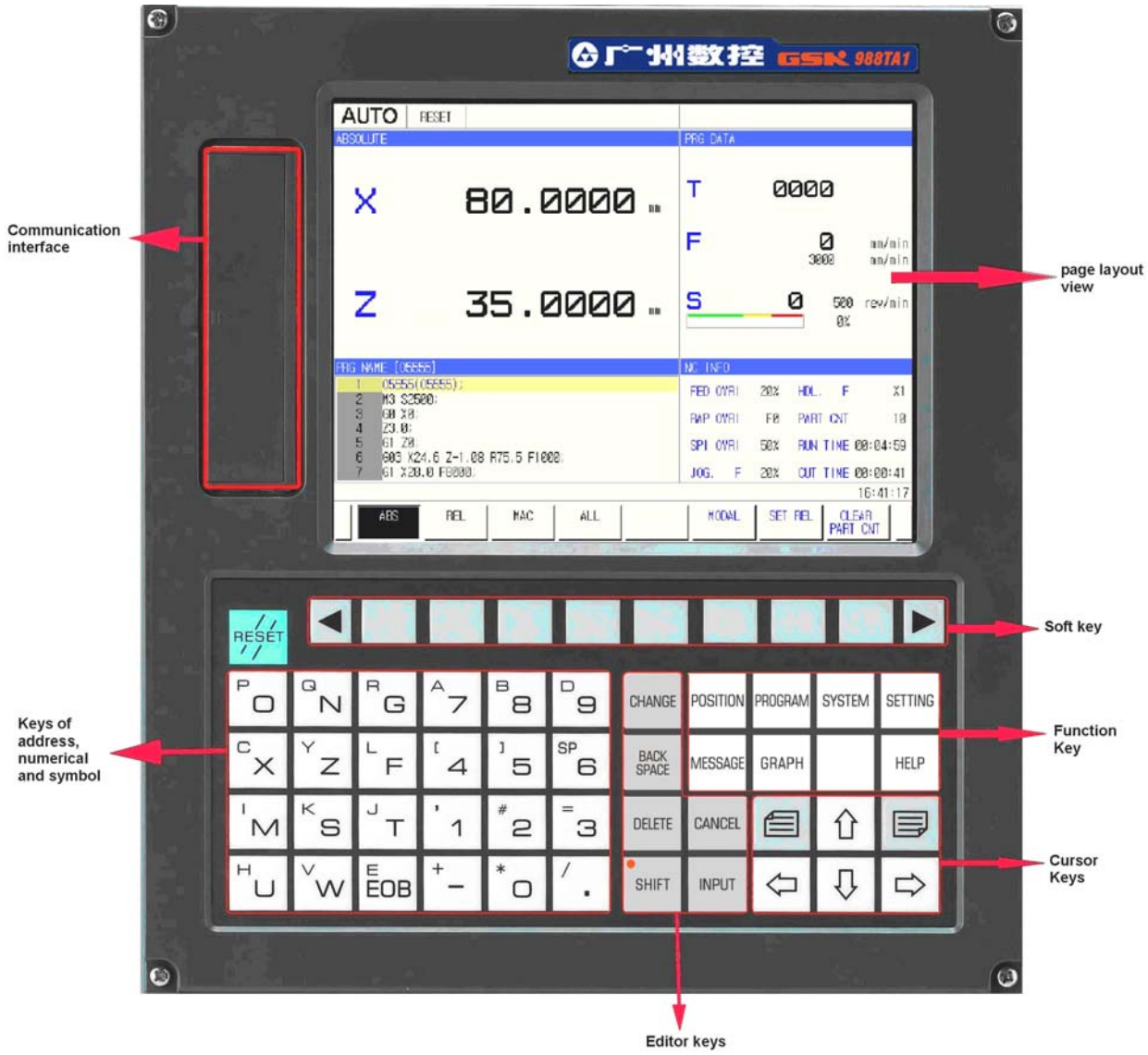


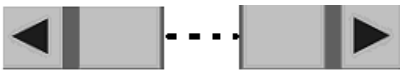
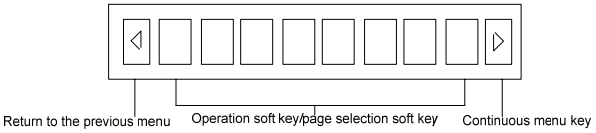
Fig. 1-8

1.4.2 Button Definition

Button	Name	Function
	Reset key	CNC reset, feed, output stop and so on
	Address/ numerical/ sign key	Address input, digit input, symbol input, pressing shift key to switch addresses or symbols

Button	Name	Function
SHIFT	Shift key	Switch the double-address key, double-symbol key, address symbol key and digit address key. Pressing the shift key and it light is ON. Press the address key and the input is the upward address; it combined with the cursor key can select a block
INPUT	Input key	Confirm the data input and line feed during editing the program
CHANGE	Change key	Switch message/display, Tab key function, fast shortkey with other keys when editing programs
BACK SPACE	Backspa ce key	Delete the characters before the cursor
CANCEL	Cancel key	Cancel the current operation
DELETE	Delete key	Delete the character after the cursor
	Cursor move-ment key	Control the cursor left/right/upward/downward
	Page key	Switch the page in the same display interface

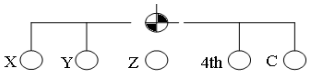
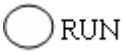
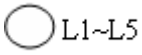
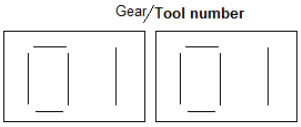
Button	Name	Function
POSITION PROGRAM SYSTEM SETTING MESSAGE GRAPH HELP	Function	POSITION press the function key to switch to the position display set. PROGRAM press the function key to switch to the program set. SYSTEM press the function key to switch to the system set. SETTING press the function key to switch to the setting sets. MESSAGE press the function key to switch to the message set. GRAPH press the function key to switch to the graph set. Custom set. HELP press the function key to switch to the help set.

Button	Name	Function
	Software keys	After using function keys to switch page set, use the corresponding soft key to display the current page set's some sub-page content or input in the current page. GSK988TA/TB's 10 soft keys are at the bottom of the screen, which is shown below.  Soft key functions: ① switch the subpages in the current page set; ② as the currently displayed subpage's operation input, such as editing, modifying data or display content

1.4.3 Key Definition on Machine Operation Panel

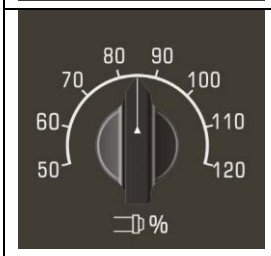
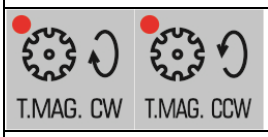
State indicators and key functions on GSK988TA's machine panel are defined by the PLC program (ladder). The state indicators and key functions described in the user manual are based on GSK988TA ladder. Please refer to the machine tool manufacture's user manual if there is difference.

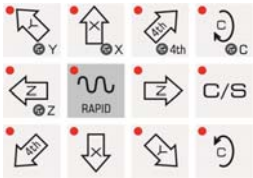
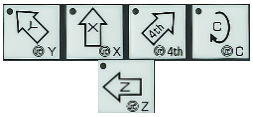
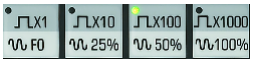
State indicator:

	Each axis reference position return end indicator lamp		Running indicator lamp
	Alarm indicator lamp		Self-defined indicator lamp
	Gear/tool location indicator lamp		

Button definition

Button	Name	Function	Operation mode when the function be valid
	Feed hold key	Program, MDI code operation pause	Auto mode, MDI mode, DNC mode

Button	Name	Function	Operation mode when the function be valid
	Cycle start key	Program, MDI code running start	Auto mode, MDI mode, DNC mode
	Feedrate override button	Adjusting the feedrate or the manual feedrate	Auto mode, MDI mode, Edit mode, Reference point return mode, MPG mode, Single step mode, Manual mode, DNC mode
	Spindle override key	Spindle speed adjustement（the spindle speed analog control mode is valid）	Auto mode, MDI mode, Edit mode, Reference point return mode, MPG mode, Single step mode, Manual mode, DNC mode
	Manual tool change key	Manual tool change	Reference point return mode, MPG mode, Single step mode, Manual mode
	Position record key	Record the current coordinate position and input the tool offset	Auto mode, MDI mode, Edit mode, Reference point return mode, MPG mode, Single step mode, Manual mode, DNC mode
	Jog switch key	The spindle jog state ON/OFF（The concrete use is referred to the standard ladder explanation or machine manufacturer's ladder description）	MPG mode, Single step mode, Manual mode
	Lubrication switch key	Machine lubrication ON/OFF	Auto mode, MDI mode, Edit mode, Reference point return mode, MPG mode, Single step mode, Manual mode, DNC mode
	Cooling ON/OFF key	Cooling ON/OFF	
	Chuck control key	Chuck clamping/releasing	Auto mode, MDI mode, Edit mode, Reference point return mode, MPG mode, Single step mode, Manual mode, DNC mode

Button	Name	Function	Operation mode when the function be valid
	Spindle control key	Spindle rotation（M4） Spindle stop（M5） Spindle rotation（M3） Spindle speed position switch Spindle exact stop(spindle orientation)	MPG mode, Step mode, Manual mode
	Rapid movement key	Rapid traverse speed/feedrate switch	Auto mode, MDI mode, Manual mode, DNC mode
	Manual feed key	Each axis positively/negatively moves in Manual, Single Step mode	Reference point return, Step mode, Manual mode
	MPG control axis selection key	Each axis selection in MPG mode Note: the key in MPU08 is valid, but MPU09's MPG selection key is on the external MPG device	MPG mode
	MPG/Single increment selection and rapid override selection key	MPG movement increment 0.001 mm /0.01 mm /0.1 mm /1 mm（note: the function key is valid only in MPU02A，but MPU02B's MPG times selection key is on the external MPG device.）	MPG mode
		Movement amount of each step 0.001 mm /0.01 mm /0.1 mm / 1 mm	MPG mode(single step increment operation)
		Rapid override Fo, F25%, 50%, F100%	Auto mode, MDI mode, Reference point return mode, Manual mode, DNC mode

Button	Name	Function	Operation mode when the function be valid
SINGLE	Single block switch	Whether the single block running is valid is controlled by the key, When the button indicator lamp is ON, the single block is valid.	Auto mode, MDI mode, DNC mode
SKIP	Block skip switch	Whether the skip code "/" of the program is valid is controlled by the key, when the button indicator lamp is ON, the skip function is valid	Auto mode, MDI mode, DNC mode
MACHINE LOCK	Machine locked switch	When the machine is locked, the machine locked indicator is ON and each axis' output is invalid	Auto mode, MDI mode, Edit mode, Reference point return mode, MPG mode, Single step mode, Manual mode, DNC mode
MST M.S.T. LOCK	Miscellaneous function locked switch	The key is for switching the status of the miscellaneous function locked, M, S, T function output is invalid when the miscellaneous function is locked.	Auto mode, MDI mode, DNC mode
DRY	Dry run switch	When the indicator lamp is ON, dry run is valid. Machining program is executed/MDI block executes dry run.	Auto mode, MDI mode, DNC mode
OPTIONAL STOP	Optional stop switch	When the optional stop is valid, the indicator lamp is ON. When M00 exists in the program, the block stops after running completes.	Auto mode, MDI mode, DNC mode
PROG. RESTART	Program restart switch	The program restart is valid, the program restart indicator is ON, the program restarts at the place where it stops previously.	Auto mode
EDIT	Edit mode key	Enter Edit mode key	Edit mode
AUTO	Auto mode key	Enter Auto mode key	Auto mode
MDI	MDI mode key	Enter MDI mode	MDI mode

Button	Name	Function	Operation mode when the function be valid
	Return to reference point mode selection key	Enter the reference point return mode	Reference point return mode
	Step/ MPG mode key	Enter single step or MPG mode(one of them is selected by the parameter)	Step/MPG/Manual mode
	Manual mode key	Enter manual mode	Manual mode
	DNC mode selection key	Enter DNC mode	DNC mode
	Emergency stop switch	After the key is pressed, the system enters the emergency stop status, all output are OFF.	Auto mode, MDI mode, Edit mode, Reference point return mode, MPG mode, Single step mode, Manual mode, DNC mode
	Power supply ON/OFF switch	System power ON/OFF switch	Auto mode, MDI mode, Edit mode, Reference point return mode, MPG mode, Single step mode, Manual mode, DNC mode
	Program protect switch	Protect program from being changed at will	Auto mode, MDI mode, Edit mode, Reference point return mode, MPG mode, Single step mode, Manual mode, DNC mode
	External feed hold key	Program, MDI code run pause	Auto mode, MDI mode, DNC mode

Button	Name	Function	Operation mode when the function be valid
	External cycle start key	Program, MDI code run start	Auto mode, MDI mode, DNC mode
	MPG	Control the machine run(note: with it for MPU08，without it for MPU09）	MPG mode

Chapter 2 Power on/off and Safety Protection

2.1 Power-on

Before GSK988TA power on, please confirm:

1. CNC and the machine are normal in the appearance;
2. The power supply voltage complies to the requirements of the manufacturer;
3. The connection is correct and firm.

After GSK988TA is powered on, the page is shown in Fig. 2-1:

Fig. 2-1

Then, GSK988TA should be self-checked and initialized. After self-checking and initializing, the page is displayed the current position (dual-channel), which is shown in Fig. 2-2:

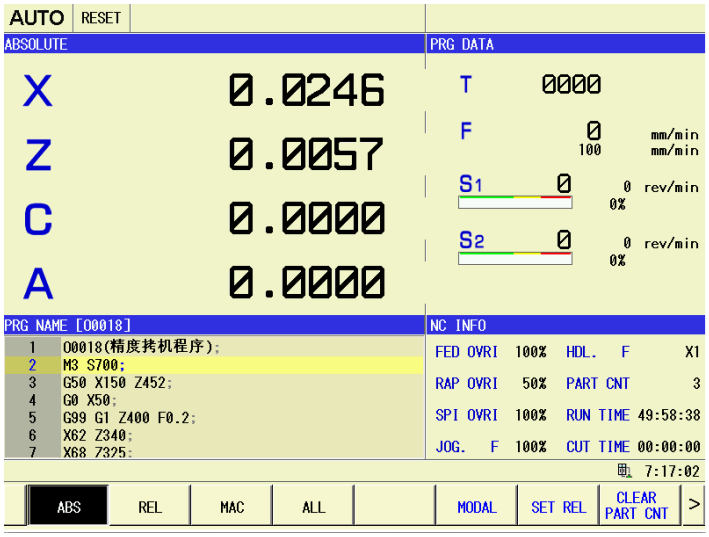


Fig.2-2

2.2 Power-off

Confirm the followings before power-off:

1. All movable parts of the machine should be stopped;
2. The miscellaneous function (such as the spindle, the water pump) is off;

3. CNC power supply should be cut off firstly, and then the machine power supply is cut off.

Note: 1. Turn on the power, again after power off for 20 seconds.

2. About the operation of cutting the machine power supply, please refer to the user manual from the machine manufacturer.

2.3 Overtravel Protection

To avoid the machine damage due to the each axis overtravel, the machine must be adopted the overtravel protection measures.

The limit switches should be respectively installed on the maximum stroke position in the positive and negative directions of each axis on the machine. When the overtravel occurs, the limit switch is operated, the system decelerates till stops, and the overtravel alarm occurs.

During automatic running, when one axis is touched the limit switch, all axes along which the tools are being moved should be decelerated and stopped, and the overtravel alarm occurs.

During manual operation, just the axis in which the tool touching the limit switch is decelerated and stopped, the tool is still moved along other axis.

The method of clearing “the overtravel” is: In manual mode, the worktable is moved reversely (like the positive overtravel, it is moved negatively, vise versa), and moved away from the limit switch. The system resets and issues the alarm.

Note: The method of the releasing the machine overtravel is different with that introduced in the manual; about the detailed operation, please refer to the user manual of the machine manufacturer.

2.4 Overtravel Protection of Stored Stroke

GSK988TA system provides three stored stroke check areas: The areas in which the tools can’t enter can be specified as the stored stroke check 1, the stored stroke check 2 and the stored stroke check 3, which is shown in Fig.2-3:

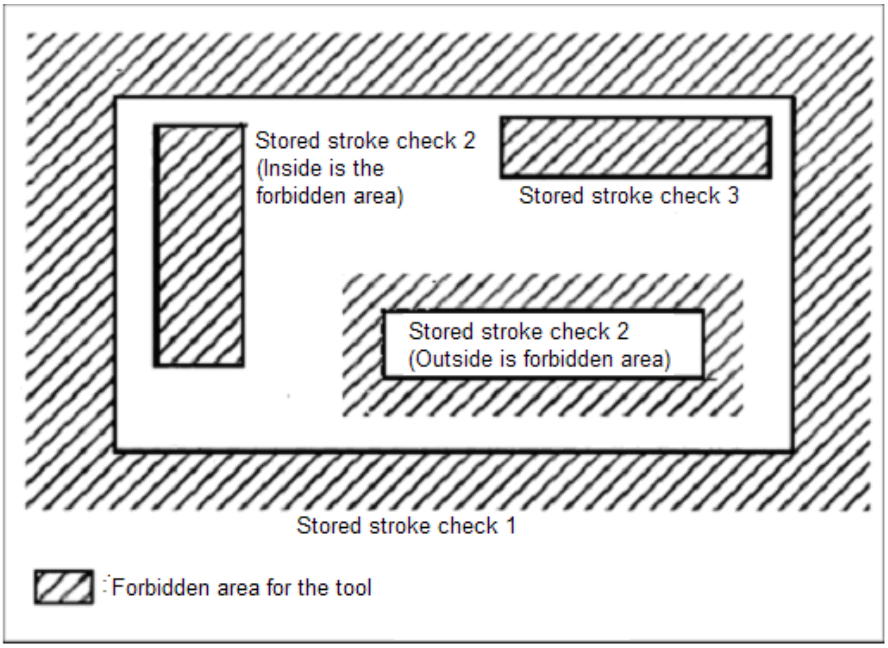


Fig.2-3

When the tool exceeds a stored stroke limit, an alarm is displayed and the tool is decelerated and stopped. When the tool enters a forbidden area and an alarm is generated, the tool can be

moved in the reverse direction from which the tool comes.

Stored stroke check 1: Parameters (Nos. 1320, 1321 or Nos. 1326, 1327) set the boundary. Outside the area of the set limits is a forbidden area. The machine builder sets this area as the maximum stroke.

Stored stroke check 2 (G22 G23): Parameters (Nos. 1322, 1323) or codes set the boundaries. In case of programming, a G22 code forbids the tool to enter the forbidden area, and a G23 code permits the tool to enter the forbidden area. Each of G22 and G23 should be executed independently of another codes in a block. About the details, refer to G codes introduction.

Stored stroke check 3: Set the boundary with parameters No.1324 and 1325. The area inside the boundary becomes the forbidden area.

Forbidden area overlapping: Area can be set in piles (refer to the following Fig. 2-4). Unnecessary limits should be set beyond the machine stroke.

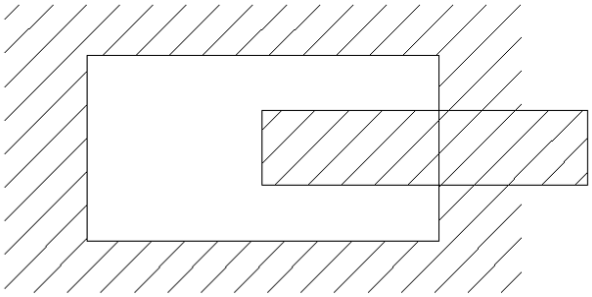


Fig.2-4

Effective time for a forbidden area: Each limit becomes effective after the power is turned on and manual reference position return or automatic reference position return by G28 has been performed. After the power is turned on, if the reference position is in the forbidden area of each limit, an alarm is generated immediately.

Time for displaying an alarm: Parameter BFA (bit 7 of No. 1300) selects whether an alarm is displayed immediately before the tool enters the forbidden area or immediately after the tool has entered the forbidden area. The parameter is only valid to the stored stroke 1 rather than “the software limits 2 and 3”.

Releasing the alarms: When the tool has become unmovable in the forbidden area, the tool is moved reversely (like the positive overtravel, it is moved negatively, vise versa) out of the forbidden area in manual mode, and the reset key is pressed to clear the alarm; if the setting is wrong, correct it and perform the reference position return, again.

Note: In setting a forbidden area, if the two points to be set are same, the area is as follows:

1. **When the forbidden area is stored stroke check 1, all areas are forbidden areas.**
2. **When the forbidden area is stored stroke check 2 or stored stroke check 3, all areas are movable areas.**
3. **When 1300.7=1 and the alarm occurs, the machine coordinate is outside the forbidden area, the alarm can be directly cancelled by pressing the resetting key.**
4. **When 1300.7=0, and the alarm occurs, the machine coordinate is inside the forbidden area, the tool is moved outside of the alarm area, and then the alarm can be cancelled by pressing the resetting key.**

The software limit function is invalid when the stored stroke check 1 is set and the coordinate set value of the positive boundary is less than that of the negative boundary.

2.5 Emergency Operation

During machining, some unexpected consequence may occur due to the user programming, operation and the product fault, GSK988TA operation must stop immediately. In this chapter, the measures are taken by GSK988TA in emergency; about the measures taken by the CNC machine tool, please refer to the user manual from the machine builder.

2.5.1 Reset

When GSK988TA is abnormally output and the coordinate axis is abnormally operated, pressing 

key makSe GSK988TA in resetting status;

1. All axis movement is decelerated and stopped;
2. M and S function output is invalid;
3. Automatic running stops and modal function remains.

Note: After 

is pressed, whether automatically switch off the signals of the spindle CW/CCW rotation, the lubrication and the cooling, etc. is set by the parameters.

2.5.2 Emergency Stop

During the machine running, the emergency stop button (the external emergency stop signal is valid) is pressed in the dangerous or in the emergency situation, CNC takes the emergency stop measures, and the machine movement is stopped immediately, all output (such as the spindle revolution and the coolant, etc) are all switched off. The emergency stop alarm is released after releasing the emergency stop button, CNC enters the resetting status.

Note 1: Please confirm the trouble has been removed before the emergency stop alarm is released;
Note 2: The electric shock can be reduced when the emergency stop button is pressed before power-on/off.
Note 3: After the emergency stop alarm is released, the reference position return should be executed, again to guarantee the correctness of the coordinate position.

2.5.3 Feed hold

During the machine running, the feeding can be hold after pressing 

key. Please pay attention that the function can't stop the feeding immediately during thread cutting and the cycle code executing.

2.5.4 Cutting off the Power Supply

During the machine running, in danger or in the emergency situation, the machine power supply must be cut off immediately to avoid the accident. Please pay attention that the reference position return should be executed, again if the coordinate displayed by CNC doesn't comply with the actual position after power off.

Chapter 3 Display Page

This chapter will introduce the switch of the subpages and the relation between the operation input and the software keys and the detailed operation method.

GSK988TA system MDI panel includes 8 function keys, like position, program and setting, etc and each function key is corresponded to one page set, and each page set also includes many subpages and the operation software keys. The following content explains how the operation software is changed after pressing each function keys.

3.1 Position Display Page Set

After the system is powered on, the initial display page is the position one, the position page without loading the program in Auto mode and during resetting status is shown in Fig.3-1:

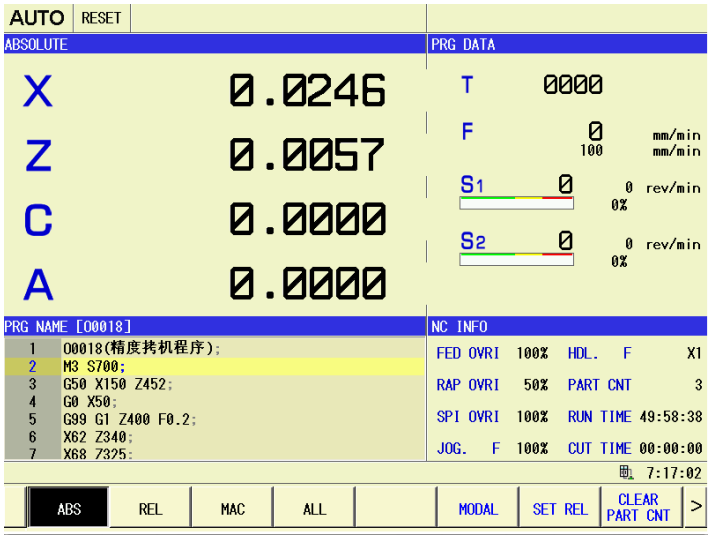


Fig.3-1

Note: The content in the page may be different based on the different configurations, the figure and all contents introduced below are according to the standard two-axis turning machine configuration.

Press POSITION function key to enter the position page set; it includes the subpages of the absolute coordinate, the relative coordinate and the machine coordinate, and the content displayed in each page can be checked by pressing the corresponding software key, which is shown in Fig.3-2.

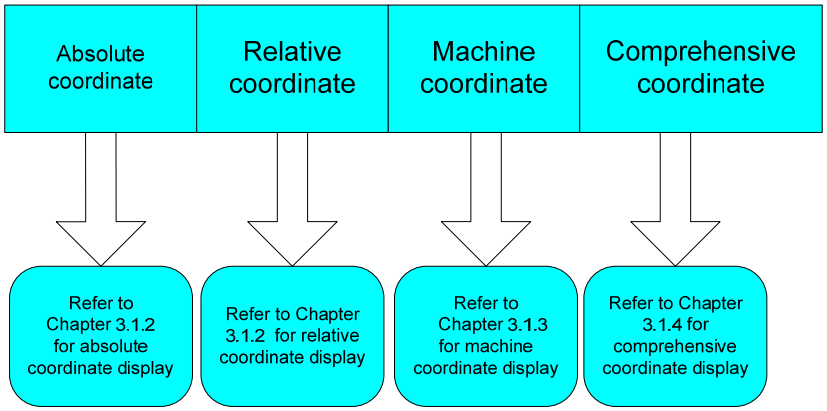


Fig.3-2

3.1.1 Absolute Coordinate Display

On the position page set, press ABS software key to switch into the absolute coordinate display page. In Auto mode, the running status is shown in Fig.3-3:

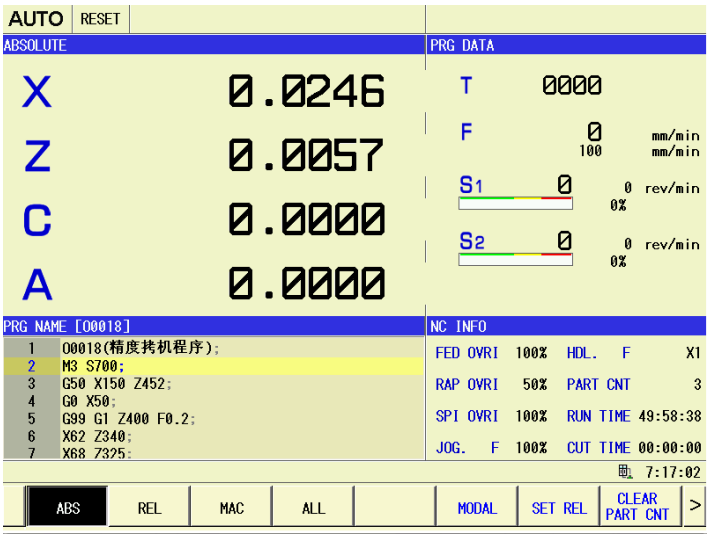


Fig.3-3

On the left top corner, the current operation mode is Auto mode. During running, linear cutting…… is popped up.

The coordinate display area is displayed that each axis coordinate value of X and Z, etc is the absolute position of the current workpiece coordinate system on which the tool is.

Processing data:

T: Current tool number and the tool offset number.

Actual rate F: During actual machining, the actual machining rate after the feedrate override calculation;

Programmed rate: In the program, the rate is specified by F code;

The spindle actual rate S: The spindle speed is the feedback of the spindle encoder, and the spindle actual speed can be displayed only after the spindle encoder is installed;

The programmed spindle speed S: In the program, the spindle speed is specified by S code;

Processing data:

T: Current tool number and the tool offset number.

Actual rate F: During actual machining, the actual machining rate after the feedrate override calculation;

Programmed rate: In the program, the rate is specified by F code;

The spindle actual rate S: The spindle speed is the feedback of the spindle encoder, and the spindle actual speed can be displayed only after the spindle encoder is installed;

The programmed spindle speed S: In the program, the spindle speed is specified by S code;

Comprehensive message:

Feedrate override: The override selected by the feedrate switches;

Rapid override: The override selected by the rapid override switches;

Spindle override: The override selected by the spindle override switches;

Manual override: The override selected by the manual override switches;

MPG override: The current MPG override;

Number of the machined work pieces: The machined workpiece number is added one after M code set by M02, M30 or parameter 6710 is executed by the program and the execution is completed.

Cutting time: The execution time of automatic running for one time doesn't include the stop and feed hold time; Timing starts from 0 after automatic running starts each time, the time unit is hour, minute, second in turn;

Running time: All execution time of the system in Auto mode doesn't include the stop and feed hold time and it is the accumulation of the cutting time;

G function code: The modal value of G code of each group;

Press modal and Comprehensive message software keys to switch between the modal and the comprehensive message.

The program display area: Display the currently being executed program. And the block displayed in green color is the currently being executed one.

3.1.2 Relative Coordinate Display

On the position page set, press REL software key to switch into the relative coordinate display page, which is shown in Fig.3-4. Then, the relative coordinate value is displayed in the coordinate display area. The displayed U and W coordinate values are the current position relative coordinate ones. U and W coordinates can be cleared during stopping and resetting status.

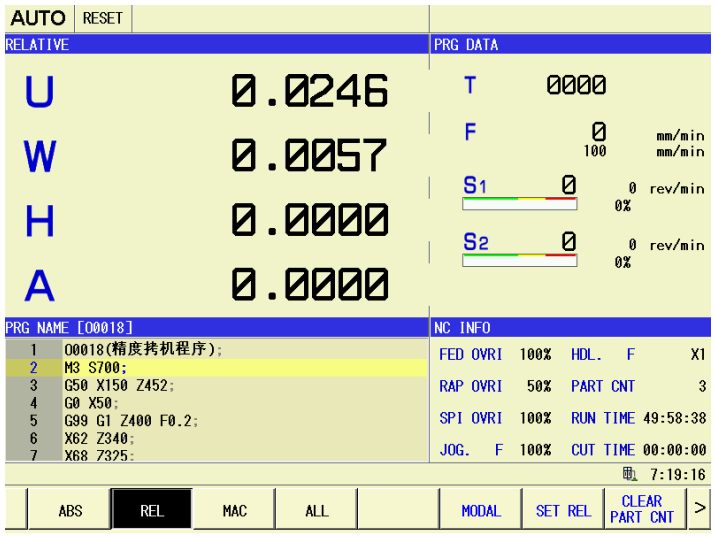


Fig.3-4

3.1.3 Machine Coordinate Display

On the position page set, press MAC to switch into the machine coordinate display page. The machine coordinate system is set by reference position return, and the page is shown in Fig.3-5:

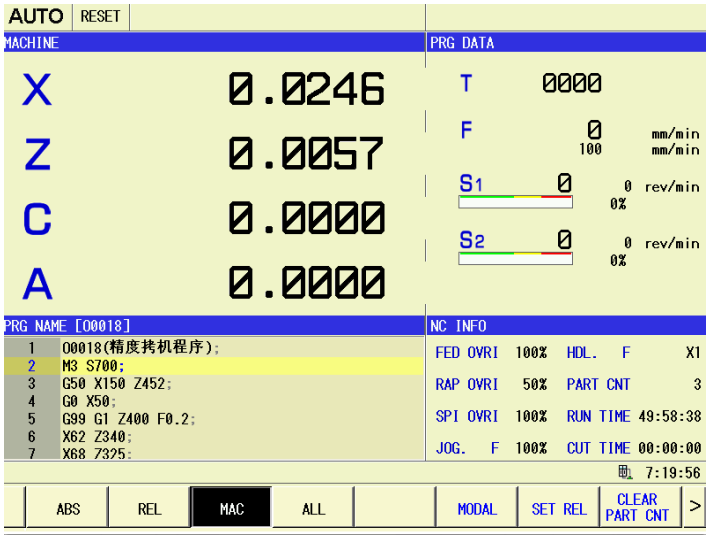


Fig.3-5

3.1.4 Comprehensive Coordinate Display

On the position page set, press ALL software key to switch into the comprehensive coordinate display page. Then, the comprehensive coordinate values displayed on the interface top include the absolute coordinate, the relative coordinate, the machine coordinate and the distance to go, and the page is shown in Fig.3-6:

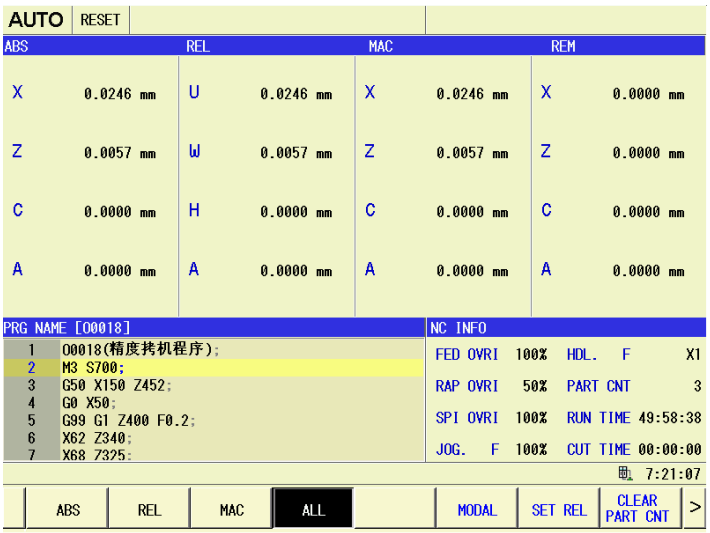


Fig.3-6

3.1.5 Relative Coordinate Setting

On the position display page set, press SET REL software key to set the relative coordinate, which is shown in Fig.3-7:

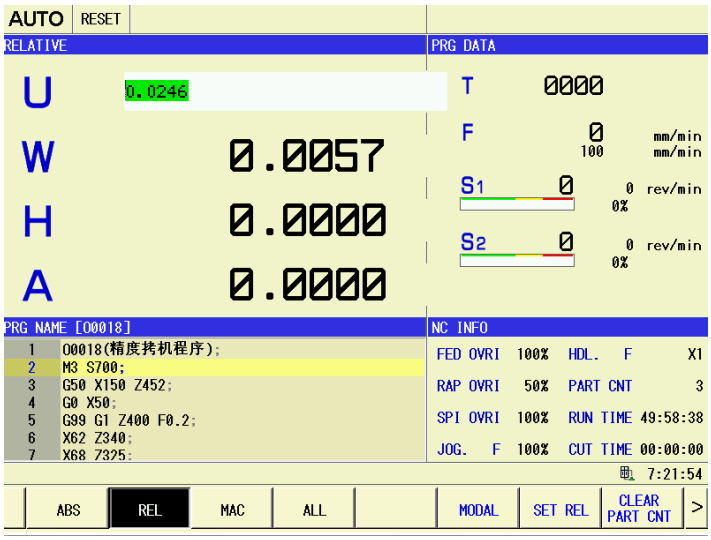


Fig.3-7

Then, the relative coordinate value of each coordinate axis can be set. The setting steps are:

（1）During resetting, press SET REL software key to input the relative coordinate axis, like the relative coordinate value U in the above figure;

（2）Press ⇧ or ⇩ to select the coordinate axis to be set, and the axis can be input;

（3）Input the relative coordinate values and press INPUT key to complete setting; the cursor is moved into the next axis and the relative coordinate of the next axis can be set. If there is no axis after the current one, setting the relative coordinate is completed.

3.1.6 Switch between the Modal and the Comprehensive Message

On the position page set, press MODAL and NC INFO to switch between the modal the comprehensive message, the modal display page is shown in Fig.3-8:

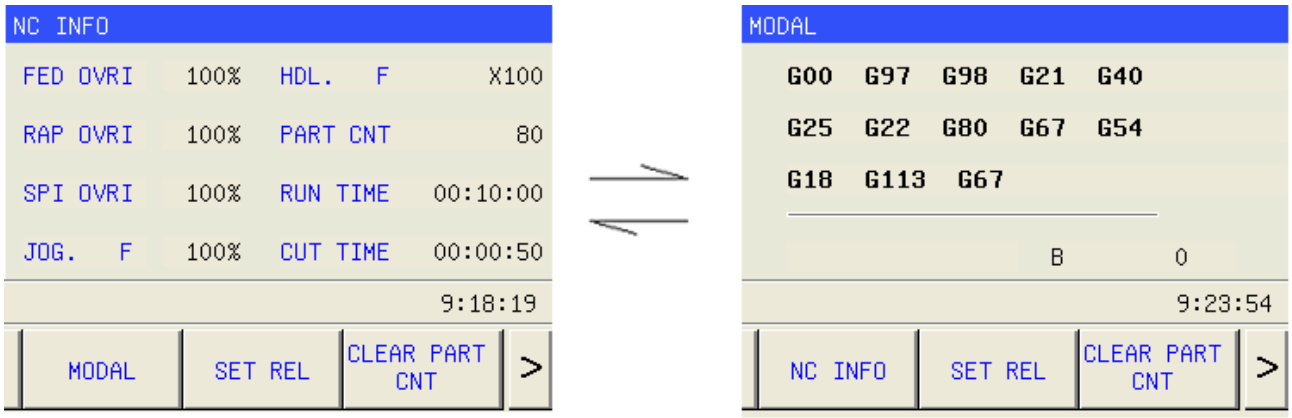


Fig.3-8

3.1.7 Clearing the Machining Workpiece Number

On the position page set, press CLEAR PART CNT to clear the current number of the machined workpiece.

3.2 Program Page Set

Press PROGRAM function key to enter the program page set. It mainly includes the local content, MDI program, the current/next display; if the U disc is inserted, the U disc content is also displayed. Search the displayed content of each page by pressing the corresponding key as Fig.3-9.

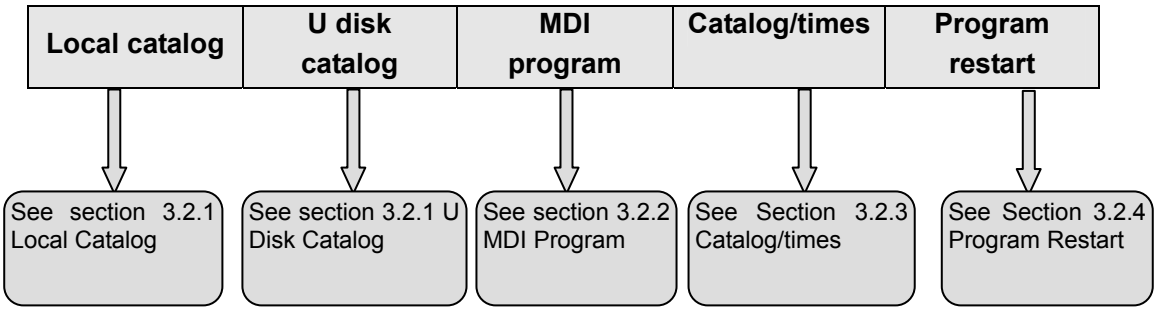


Fig.3-9

3.2.1 Local Content and U Disc Content

● **Local content**

Press LOCAL software key to display the program content in the system. The programs in the local content can be loaded, opened, copied, pasted, created, saved as, deleted, renamed and searched, etc.

The local content in the program is shown in Fig.3-10:

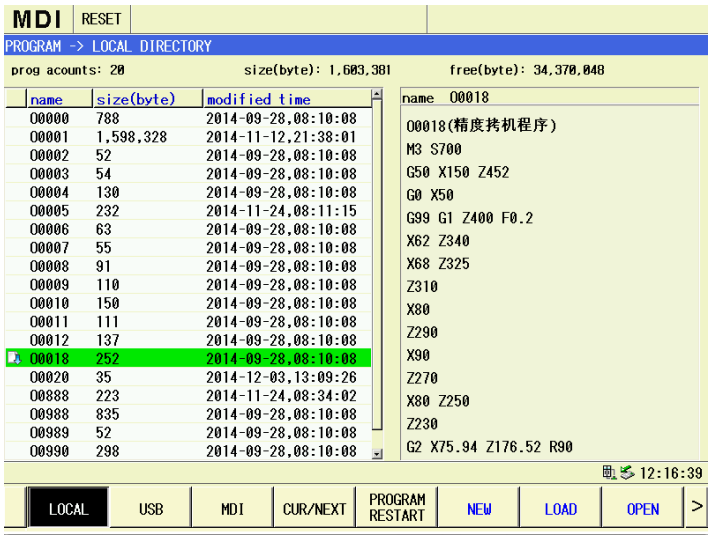


Fig.3-10

On the page top status message display area, the running mode and status, on which the system is, are displayed on the display area; the total number of the program, the total used space and the remaining space of all the programs in the current system are displayed below.

In the list, the program list of the current program and the size of each program and the latest rewritten date are displayed. Among them, the program with the green background is the selected one, like O0043 program in the above figure. While the program of the red font and marked with the channel is the one which should be loaded into the position display page of the channel and can be executed, like O8888 and O0043 programs in the above figure.

- **Content in the U disc**

When the system is with the U disc, “the content of U disc” software key is also displayed in the page meanwhile, which is shown in Fig.3-11. Press USB software key, CNC program content in “NCPROG” file of the U disc is displayed in the window. The files in the U disc content can be input and output.

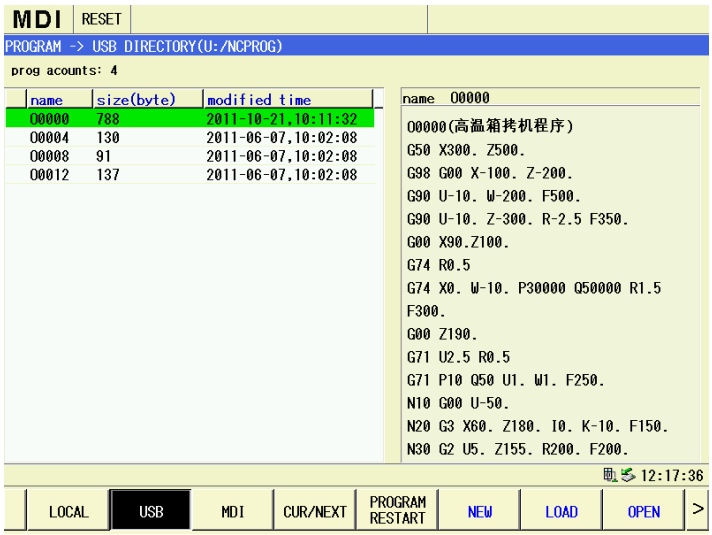


Fig.3-11

3.2.2 MDI Program

Press PROGRAM function key to enter the program page set. Press MDI soft key to enter the MDI program display page. In MDI mode, input up to 10-line programs in the MDI program input box, as shown in Fig.3-12:

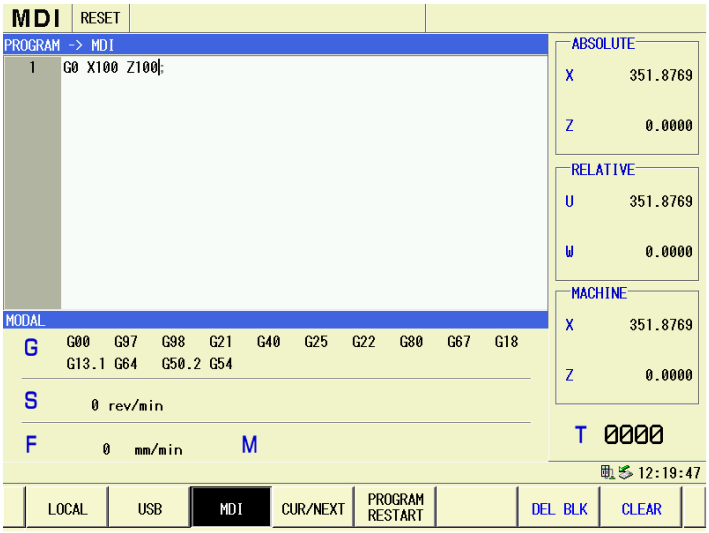


Fig.3-12

In MDI mode, the system displays the soft key DEL BLK and CLEAR. Press DEL BLK todelete the NC code which line the cursor is in. Press CLEAR to clear all NC codes in the MDI program input box.

3.2.3 Current/Next Block

Press PROGRAM function key to enter the program page set and press CUR/NEXT software key to display the current being executed block and NC code of the next block, which is shown in Fig.3-13:

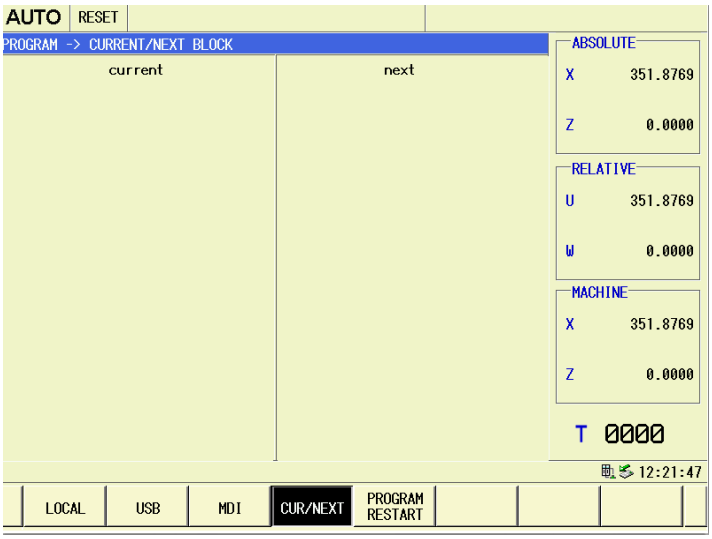


Fig.3-13

3.2.4 Program Restart

Press PROGRAM to enter the program page set and then press the soft key PROGRAM RESTART to enter the program restart state display page.

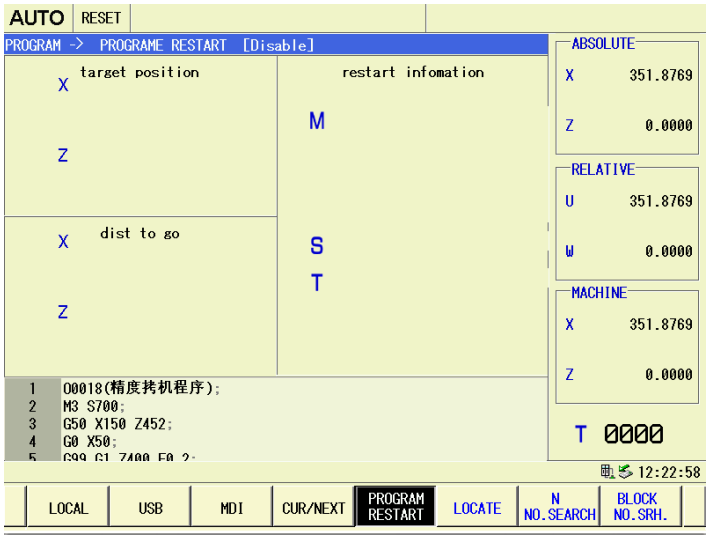


Fig.3-14

3.3 System Page Setting

Press SYSTEM function key to enter the system page set, and it mainly includes the subpages of the parameter, the pitch error compensation, the system message, the file management and the ladder diagram, etc. The content displayed on each page can be checked by pressing the corresponding software key and its software layer structure is shown below:

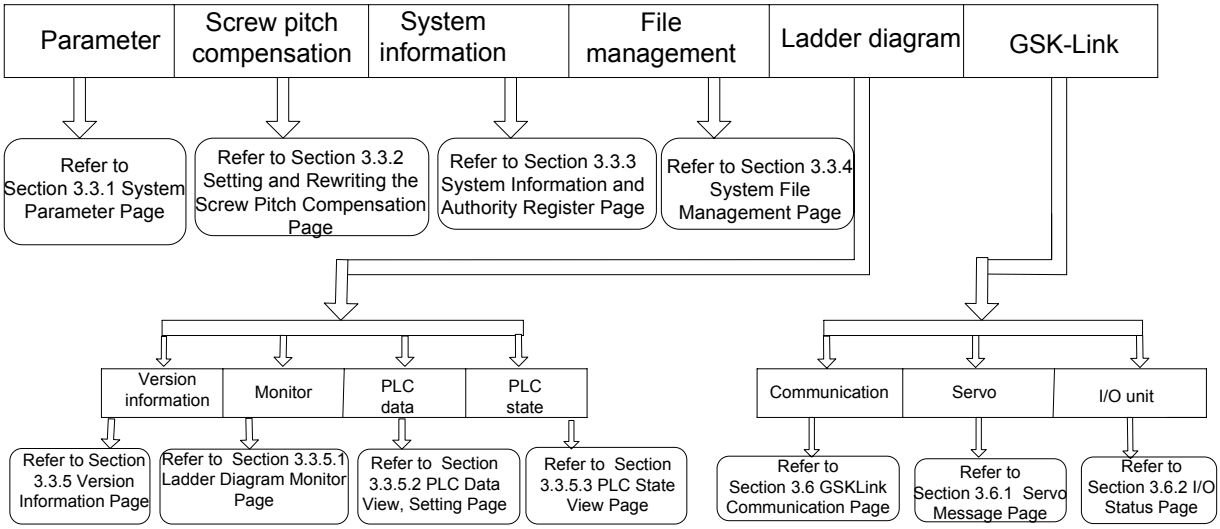


Fig.3-15

3.3.1 Parameter Setting

On the system page set, press PARAM software key to enter the parameter setting interface, which is shown in Fig.3-16:

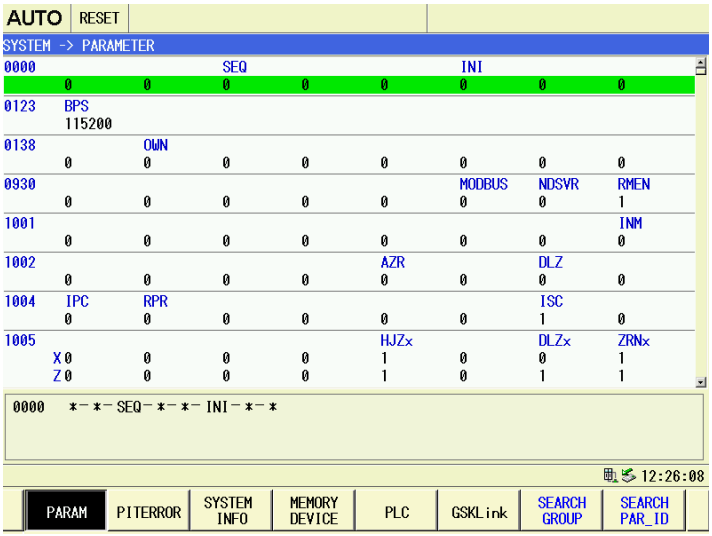


Fig.3-16

The page is displayed the detailed message of the user parameters. On the page, the system parameters can be set and rewritten, the parameters currently set by the user can be operated the backup and the parameter can be restored into the system default one or the user backup one.

The parameter can be set in MDI mode and the parameter switch is ON and the operation authority level is equal to or higher than level [3]. The parameter to be rewritten can be set by , , and keys; or press SEARCH PAR_ID or SEARCH PAR_NAME software keys to input the parameter sequence number to be selected, and then press OK software key, finally move the cursor to the parameter position. Like #0000 parameter in the above figure. Press INPUT key to make the selected parameter can be rewritten, like #0000 parameter in the following Fig.3-17:

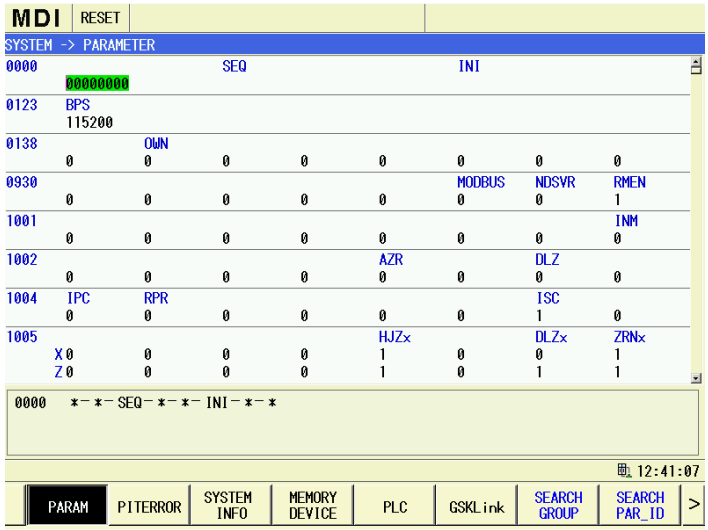


Fig.3-17

Press the numerical value key to input the digits of 8 bits in binary system, and then press INPUT key to confirm the setting is completed; when the length of the input value isn't 8 bits, 0 is supplied in high bit;

Moreover, the bit parameter can be set based on the bits:

（1）On the parameter setting page, the parameter to be set is selected by the keys , , and .

（2）The parameter bit to be rewritten can be selected by and keys.

（3） INPUT key is repeatedly pressed, the parameter bit is switched between 0 and 1, and the value of the parameter bit is rewritten.

（4）Move the cursor to complete setting.

The other parameters to be set are selected by , , and .

Setting method of the parameters of the numerical value is similar with that of the parameters of bit type:

(1) The parameter to be rewritten can be set by , , and keys; or press SEARCH PAR_ID or SEARCH PAR_NAME software keys to input the parameter sequence number or the parameter name to be selected, and then press OK software key, finally move the cursor to the parameter position.

(2) Press INPUT key to make the selected parameters modifiable.

(3) Press the numerical value key to set the numerical values and then press INPUT key to confirm the setting completed.

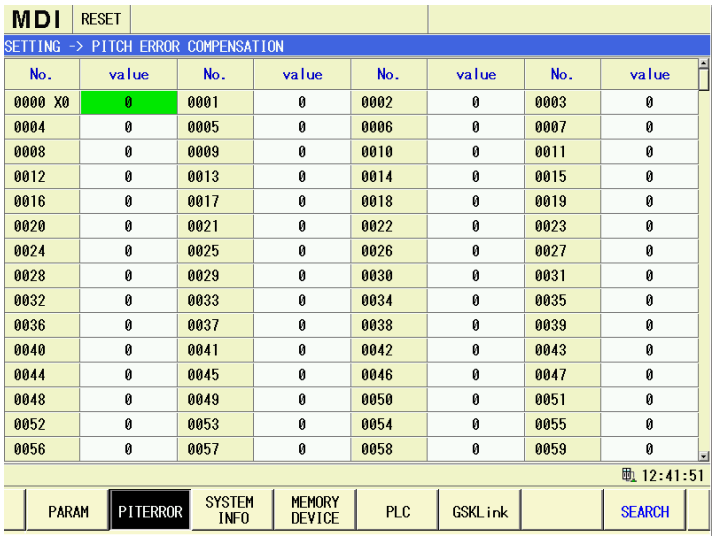
(4) Select the other parameters to be set by , , and keys.

Note 1: After the system parameters are rewritten, some parameters become valid immediately, while some parameters become valid after the system is powered on, again. About the details, refer to GSK988TA parameter introduction.

Note 2: The parameter setting can be rewritten in MDI mode and the parameter switch is ON and the operation authority level is equal to or higher than level [3].

3.3.2 Pitch Compensation Page

On the system page set, press PITERROR software key to enter the pitch error compensation page, which is shown in Fig.3-18:



No.	value	No.	value	No.	value	No.	value
0000 X0	0	0001	0	0002	0	0003	0
0004	0	0005	0	0006	0	0007	0
0008	0	0009	0	0010	0	0011	0
0012	0	0013	0	0014	0	0015	0
0016	0	0017	0	0018	0	0019	0
0020	0	0021	0	0022	0	0023	0
0024	0	0025	0	0026	0	0027	0
0028	0	0029	0	0030	0	0031	0
0032	0	0033	0	0034	0	0035	0
0036	0	0037	0	0038	0	0039	0
0040	0	0041	0	0042	0	0043	0
0044	0	0045	0	0046	0	0047	0
0048	0	0049	0	0050	0	0051	0
0052	0	0053	0	0054	0	0055	0
0056	0	0057	0	0058	0	0059	0

Fig.3-18

On the page, the user can check and set the pitch error compensation value corresponding to each screw pitch number.

On the pitch error compensation page, the compensation value of the pitch error compensation number can be selected by the page keys , and the cursor movement keys , , , ; or the pitch compensation number can be searched by SEARCH software key, and the cursor can be positioned into the compensation value of the pitch compensation number to be rewritten.

When the operation authority is equal to or higher than level [2], INPUT key is input to make the selected pitch error compensation value modifiable. The compensation value is input by the numerical keys and rewriting is completed after INPUT is pressed.

3.3.3 System Message Page

On the system page set, press SYSTEM INFO software key to enter the system message display page, and the page is shown in Fig.3-19:

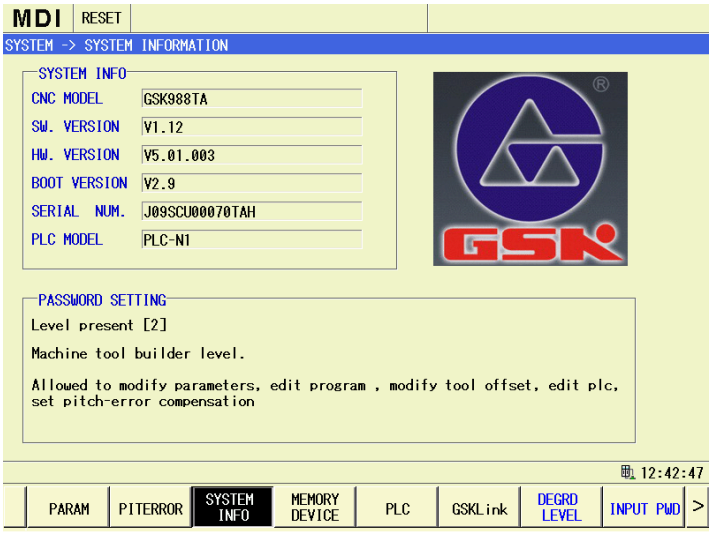


Fig.3-19

The system message display page is mainly displayed the product type, the software, the hardware and BOOT versions, the system code number, PLC type and operation authority level. The operation authority level password can be rewritten and the operation authority can be set on the page.

To realize the multi-level operation authority management, like the development maintenance, the machine design and the equipment management, etc, GSK988TA/TB CNC system sets 5 level operation authority level, 1 is the highest, 5 is the lowest:

- Level 1: The development level with the system software maintenance authority;
- Level 2: The machine manufacturer level includes PLC program editing, the screw pitch error compensation data input and power off in the set time, etc.
- Level 3: (The user) equipment management level includes rewriting the parameter, editing the part program and editing the tool compensation data, etc;
- Level 4: The machine operation level includes editing the tool compensation data and selecting the part program (that is to say: the tool setting can be operated the part program for automatic running can be selected), the parameters can't be rewritten and the part program can't be edited;
- Level 5: Operation limit level, without the operation password (the operation password cancel status), the parameters can't be rewritten, the part program can't be edit, the cutter compensation data can't be edit, the part program can't be selected (that is to say: the

tool setting operation is invalid, only the current program of the system can run). The manual, MPG, zero return, MDI running and automatic running can be operated, and some files of the system can get backup, while they can't be downloaded.

Note: CNC files is uploaded into PC rather than downloaded into CNC internal.

The operation list of operation functions about the operation authority level.

Operation authority level / Operation function	Level 1 (Development)	Level 2 (Machine tool manufacturer)	Level 3 (Equipment management)	Level 4 (Machine operation)	Level 5 (Limit operation)
System software upgrade	Yes	Yes	Yes	Yes	Yes
Setting power off in the set time	Yes	Yes	No	No	No
Editing, downloading and uploading PLC program	Yes	Yes	No	No	No
Inputting the pitch error compensation data and downloading the pitch error compensation files	Yes	Yes	No	No	No
Parameter switch ON (Allowable to rewrite the parameters)	Yes	Yes	Yes	No	No
Program switch ON (Allowable to edit the program)	Yes	Yes	Yes	Yes	No
Uploading and downloading the part programs	Yes	Yes	Yes	Yes	No
Setting the tool lifetime and downloading the tool lifetime file	Yes	Yes	Yes	Yes	No
Inputting the macro variables	Yes	Yes	Yes	Yes	No
Inputting the cutter compensation data (The tool setting allowable) and downloading the cutter compensation and tool offset files	Yes	Yes	Yes	Yes	No
Uploading the pitch error compensation files	Yes	Yes	Yes	Yes	Yes
Uploading the tool lifetime file	Yes	Yes	Yes	Yes	Yes
Uploading the cutter compensation and tool offset files	Yes	Yes	Yes	Yes	Yes

If the operation limited by the authority level should be executed, the corresponding operation authority level should be obtained. Press SYSTEM key to enter the system interface, and then press SYSTEM INFO to enter the password display interface, and press DEGRD LEVEL, ALTER PWD or

INPUT PWD to enter the corresponding setting, and input the password corresponding to the operation level, so the operation authority corresponding to the level is obtained. On the password setting page, the operation password of the level or that lower than the level, the current password level can also be degraded.

The authority level doesn't remain after power off at operation authority level 1 or 2, and the operation level 3 is entered after power on, again. The operation authority of 3～5 levels remain after power off, and the operation authority level before power off is restored after power off, again.

When the authority isn't enough, the reminder is popped up to remind that the corresponding operation authority isn't enough. In Auto mode, the operation authority executed by the program isn't enough, the machine movement should be stopped and the alarm occurs.

（1）Entering the authority level

Press DEGRD LEVEL software key to reduce the operation level and the current operation authority level is displayed in the operation authority column of the window.

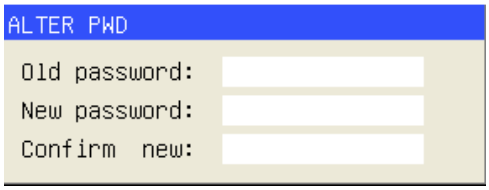
Press INPUT PWD software key to input the password of the corresponding level to enter the corresponding operation level.

Note: The initial password relation corresponding each authority level is shown in the following list.

Operation level	Initial password
Level 1	***
Level 2	***
Level 3	333333
Level 4	444444
Level 5	Without operation password

（2）Rewriting the password

Firstly, the operation authority level for rewriting the password should be entered; press ALTER PWD software key to rewrite the system current authority password. Input the new and old passwords in



edit box, and the cursor can be switched between the new and old password edit boxes by pressing CHANGE, and finally press OK to complete rewriting the password.

3.3.4 System File Management

In the system page set, press MEMORY DEVICE software key to enter the file management display page and the page is shown in Fig.3-20:

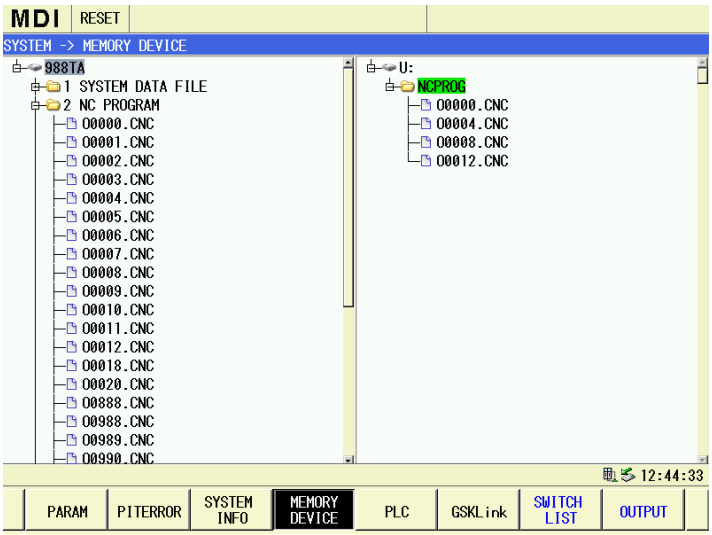


Fig.3-20

The window is divided into the left and right columns. The left display column is displayed the system files and the content of the part program files; when the system is with the U disc, the right display column is displayed the file content in the U disc, which is shown in the above figure. Then, the system files can be input and output, and the files in the system can be output to the U disc, or the files in the U disc can be input into the system.

(1) Press SWITCH LIST software key to switch the cursor between the left file content column of the system and the right content column in U disc.

(2) When the cursor is on the file folder, press ⇦ and ⇨ to fold and unfold the file.

(3) Press ⇧, ⇩ keys to move the cursor to the file to be operated, and press INPUT key to select the file and the selected file is ticked off, like the part programs O0098，O0003 and O0777 in the system file content. When the cursor is on the selected file, then, INPUT key is pressed to select all files in the folder.

(4) Then, after the system files are selected, OUTPUT software key is pressed to output all the selected files into the U disc; same, after the files in the U disc are selected, OUTPUT software key is pressed to input all the selected files into the system file content.

3.3.5 The Ladder Diagram

Press SYSTEM function key and then press PLC software key to enter the current PLC display page and real-time check PLC execution situation; the ladder diagram page mainly includes the subpages of the version message, the monitor, PLC data and PLC status, etc, and the content displayed in each page can be checked by pressing the corresponding software keys. And the page is displayed in Fig.3-21.

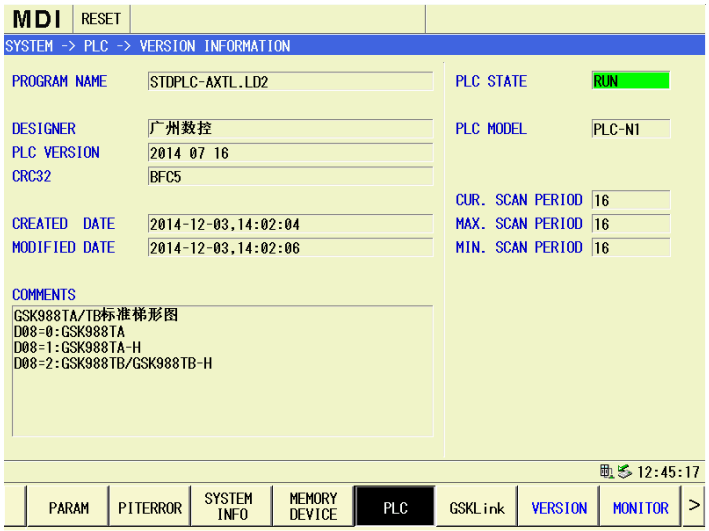


Fig.3-21

The system current running mode and status are displayed on the top of the page and the version message of the ladder diagram, the currently running ladder diagram program and its running status, etc are displayed.

3.3.5.1 The Ladder Diagram Monitor Display

Press SYSTEM function key, and then press the ladder diagram software key to enter the ladder diagram page set and press MONITOR software key to enter the operation monitor display page of the ladder diagram program, which is shown Fig.3-22:

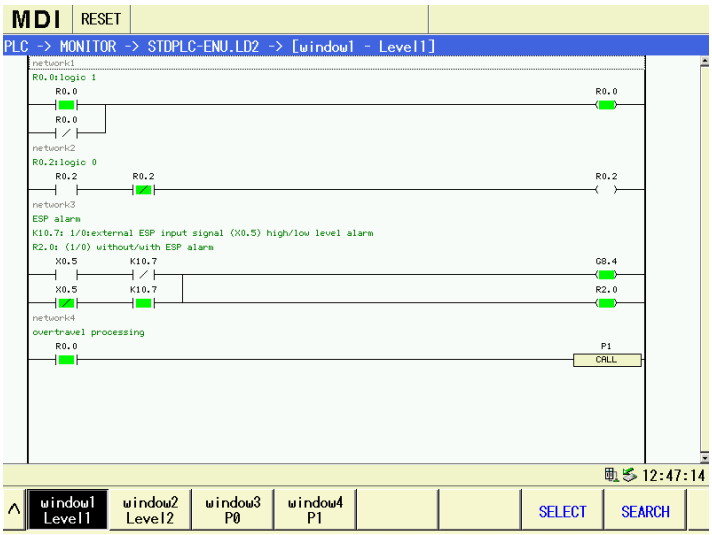


Fig.3-22

The monitor page can be checked the connect/disconnect status of the current contacts and the coils, and the current value of the timer and the counter. When the contact and the coil are connected, the base color is green; otherwise, the base color is same as the window one. For example, X0.5 means the contact X0.5 is connected, Y25.2 means the coil Y25.2 is not connected.

1. Checking the window program

In the monitor page, four window programs should be monitored meanwhile, and window1 Level1, window2 Level2, window3 P0 and window4 P1 can be respectively pressed to check the ladder diagram block corresponding each window, and then, the ladder diagram of the block corresponding to the selected window is displayed on the screen.

2. Selecting the window block

（1）The window to choose the block is selected by pressing window1 Level1, window2 Level2, window3 P0 or window4 P1 to select the window.

（2）Press SELECT software key to select the window program, and then the page is shown in Fig.3-23:

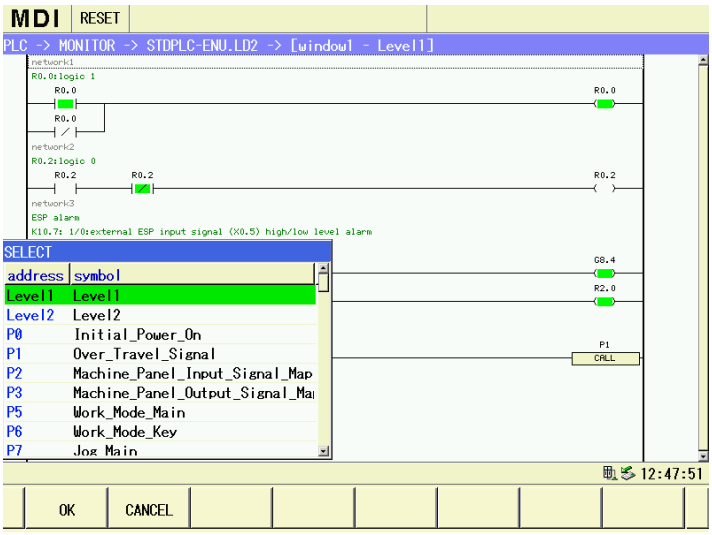


Fig.3-23

（3）Press  ,  ,  ,  to select the ladder diagram corresponding the window.

（4）Press OK software key to confirm the selection and to return the previous menu and press CANCEL software key to cancel the selection and to return to the previous menu.

3. Searching for the parameter, the code and the internet

(1) Select the block window to search for the code, the parameter and the internet; that is to say, press window1 Level1, window2 Level2, window3 P0 or window4 P1 software key to select the window and the corresponding block ladder diagram is displayed in the window and the code, the parameter and the internet can be searched in the window.

(2) Press SEARCH software key to enter the searching page, which is shown in Fig.3-24:

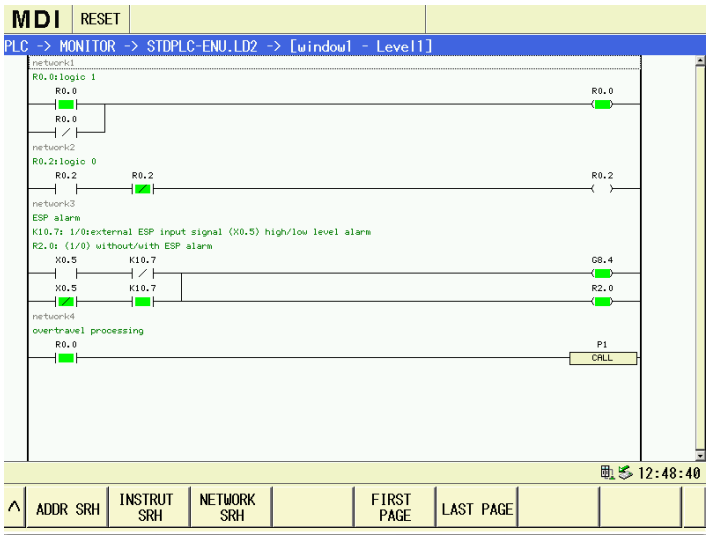


Fig.3-24

（3）Separately press [ADDR SRH], [INSTRUT SRH], [NETWORK SRH] software key to search for the parameter, the code and the internet in the block corresponding to the window and move the cursor to position in the corresponding location.

（4）Press [FIRST PAGE], [LAST PAGE] to move the cursor to position at the initial and the last lines of the block corresponding the window.

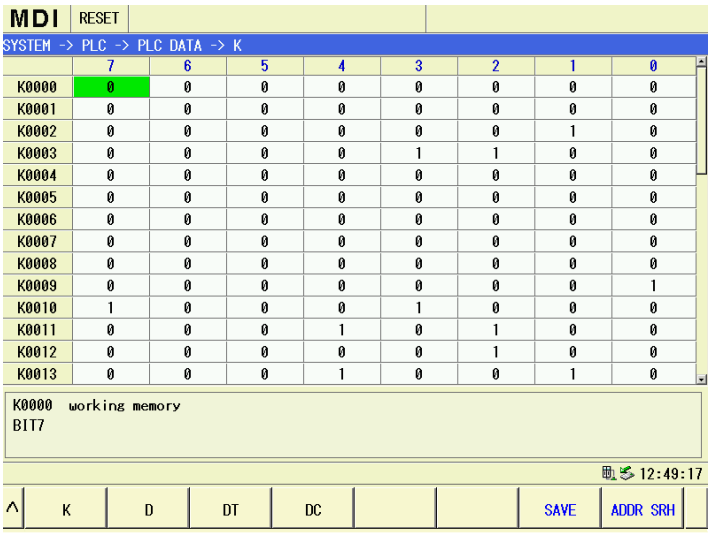
（5）Press [∧] software key and the screen is displayed returning to the previous menu.

4. Return

In the figure, press [∧] software key and the screen window is displayed returning to the previous menu.

3.3.5.2 PLC Data

In the ladder diagram page set, press the soft key [>], and then press [PLC DATA] to enter PLC data status display page and it includes setting K, D, DT and DC parameters, which is shown in Fig.3-25:



	7	6	5	4	3	2	1	0
K0000	0	0	0	0	0	0	0	0
K0001	0	0	0	0	0	0	0	0
K0002	0	0	0	0	0	0	1	0
K0003	0	0	0	0	1	1	0	0
K0004	0	0	0	0	0	0	0	0
K0005	0	0	0	0	0	0	0	0
K0006	0	0	0	0	0	0	0	0
K0007	0	0	0	0	0	0	0	0
K0008	0	0	0	0	0	0	0	0
K0009	0	0	0	0	0	0	0	1
K0010	1	0	0	0	1	0	0	0
K0011	0	0	0	1	0	1	0	0
K0012	0	0	0	0	0	1	0	0
K0013	0	0	0	1	0	0	1	0

Fig.3-25

1. Setting K parameter

(1) On PLC data display page, press [K] software key to enter setting K parameter display page.

(2) Press [ ], [ ], [⇧], [⇩], [⇦], [⇨] key to select the parameter status bit to

be rewritten; or press ADDR SRH software key to input K variable to be selected, and then press OK software key to move the cursor to position in the parameter. The meaning of the status bit is displayed at the bottom of the screen.

(3) INPUT key is pressed repeatedly and the status bit can be switched between 0 and 1, and the status of the selected K parameter status bit can be rewritten.

(4) Press ⇧, ⇩, ⇦, ⇨ to move the cursor to complete rewriting.

Press ADDR SRH software key to input K parameter address to be searched and the cursor is positioned to K parameter address to be searched.

2. Setting D parameter

（1）In PLC data status display page, press D software key to enter setting parameter D display page, which is shown in Fig.3-26:

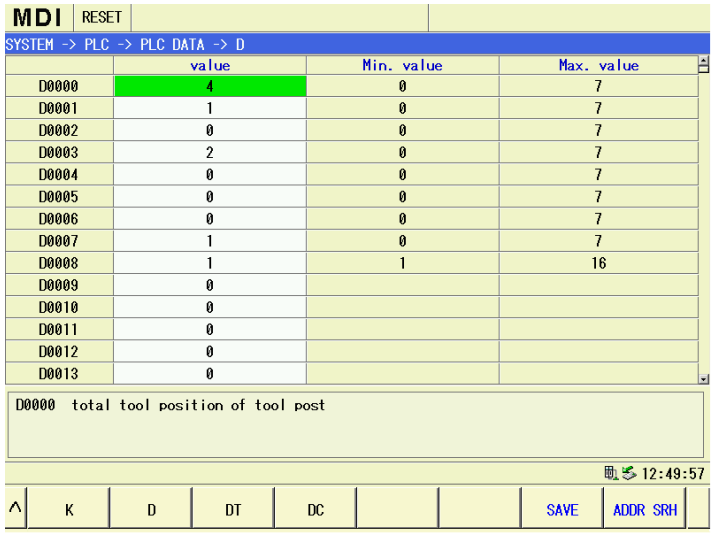


Fig.3-26

（2）Press ⊟, ⊟, ⇧, ⇩ key to select parameter D to be rewritten; or press ADDR SRH software key to input parameter D to be selected, and then press OK software key to move the cursor to position in the parameter. The meaning of the parameter is displayed at the bottom of the screen;

（3）Press INPUT key to make the selected parameter D modifiable.

（4）Input the numerical value to be rewritten and then press INPUT to complete rewriting.

3. Setting parameter DT

（1）On PLC data status display page, press DT software key to enter DT parameter setting display page.

The method of setting parameter DT is same that of setting parameter D.

4. Setting parameter DC

On PLC data status display page, press DC software key to enter DC parameter setting display page.

The method of setting parameter DC is same that of setting parameter D.

3.3.5.3 PLC Status

On the ladder diagram page set, press > software key and then press PLC STATE to enter PLC status display page, and the page is shown in Fig.3-27:

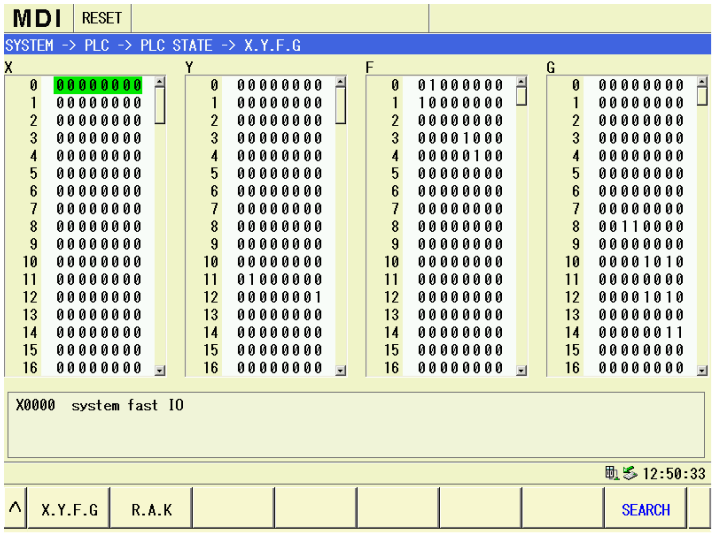


Fig.3-27

On the page, press X.Y.F.G software key, the window is displayed the status message of each parameter X, Y, F and G. Press ⇦, ⇨ keys to check among parameters X, Y, F and G.

Press R.A.K software key to check the status message of each parameter R, A and K. Then, press ⇦, ⇨ keys to switch among parameters R, A, K, and press ⇧, ⇩ keys to check among parameters P, A and K.

During checking the parameters, the note of each parameter is displayed at the bottom of the screen meanwhile; press CHANGE to switch and to check the detailed note of each data in each parameter.

Press SEARCH software key to move the cursor to position the parameter to be searched. Searching is operated in the whole page, and only inputting the parameter name and parameter number can be found exactly; while only inputting the parameter number can't find the parameter.

Press ∧ software key and the screen is returned to the previous menu.

3.3.6 GSK-Link Communication Setting Page

Press the function key SYSTEM and then the soft key GSKLink to enter the GSKLink display page to view the current communication message. GSKLink page includes the communication, servo, I/O subpage. The user can view the displayed content in subpages by corresponding soft keys. The displayed page is shown in Fig.3-28.

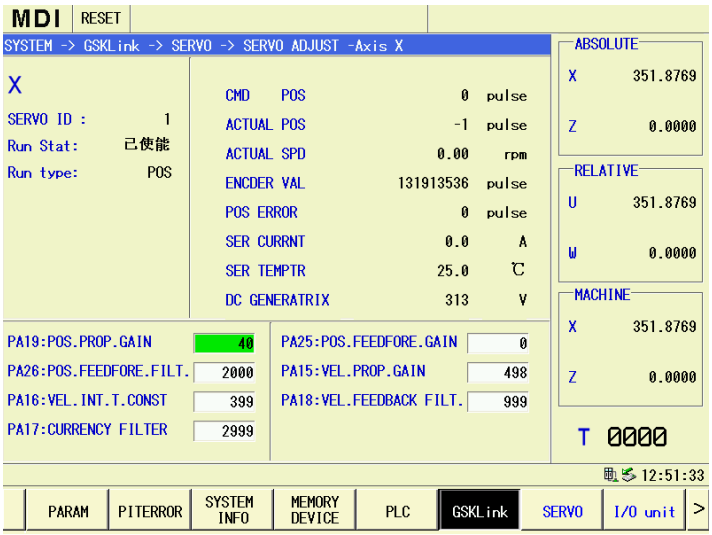


Fig.3-28

The upper in the page displays the devices' parameters, corresponding physical addresses and logic ID required by the CNC.

3.3.6.1 Servo Message Page

Press the function key SYSTEM, and then GSKLink to enter the GSKLink page set. Press SERVO to enter the servo message display page. GSKLink page includes the servo adjustment, servo parameter, servo I/O, servo debugging and oscilloscope subpage. The user can view the displayed content in subpages by corresponding soft keys. The displayed page is shown in Fig.3-29.

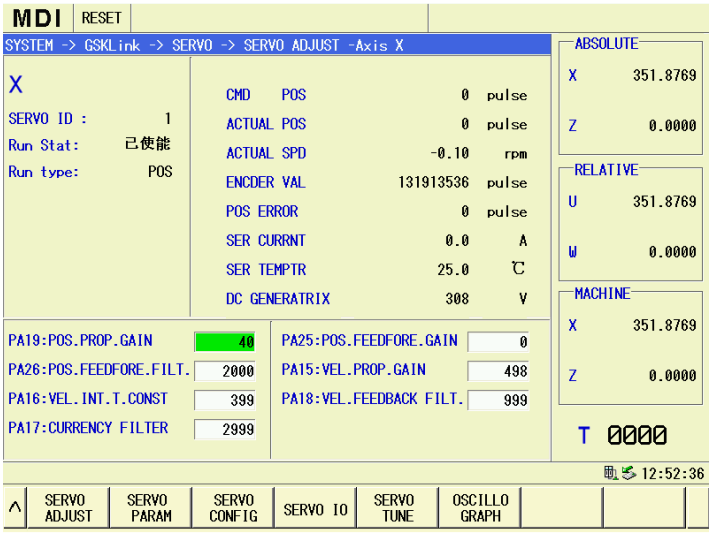


Fig.3-29

Servo adjustment

In the servo display page, press the soft key SERVO ADJUST to enter the servo adjustment displaypage. The display page is shown in Fig.3-30：

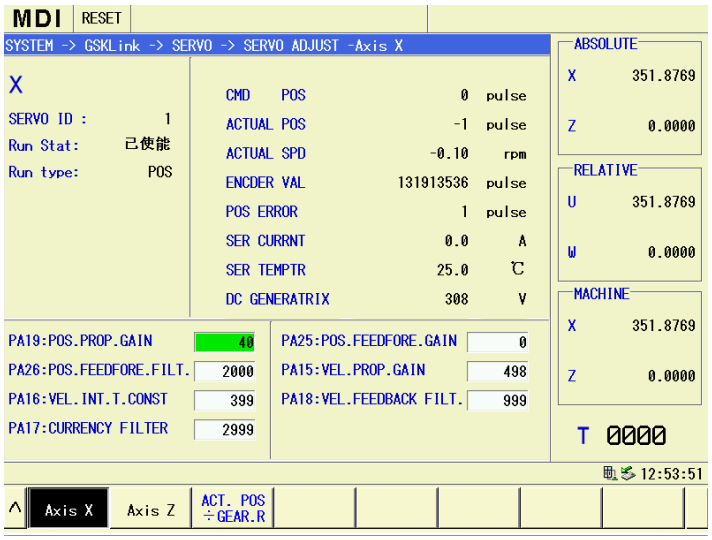


Fig.3-30

GSK988TA's servo adjustment module provides the following functions:

Monitor the system's controlled axis in real-time by the servo communication feedbacking data, which makes the operator understand the current working states of the servo, motor and so on, including:

（1）When the servo is in the position control mode, the system displays the servo receiving code pulse quantity, the servo's feedback pulse quantity from the motor's encoder, the motor's actual speed, the servo's internal current and the temperature checked by the servo;

（2）When the servo is in the speed control mode, the system displays the servo receiving the code speed, the servo's actual speed gained by the motor, the servo receiving code pulse quantity, the servo's internal current, and the servo checking temperature(the servo spindle increasing the spindle's encoder value display).

Data display area description of the servo diagnosis page:

X：the current selected axis' axis name.

SERVO ID：the slave machine's salve machine number connected the axis.

RUN STAT：the current servo drive's run state.

RUN TYPE：the servo control mode corresponded to the diagnosis data may be displayed to "Position" and "Speed".

CMD POS：the position pulse quantity received by the diagnosis servo from the system.

ACTUAL POS：the position pulse quantity feedback by the diagnosis servo.

The displayed in the bottom of the above is all kinds of gain adjustment of the servo run, and the page can be adjusted when the system switch is ON. The servo gain's adjustment options are: position proportional gain, position loop feedforward gain, position feedforward filter coefficient gain, speed loop proportional gain, speed loop gain, speed feedback filter coefficient and the analog filter coefficient. The corresponding adjustments are referred to their options.

Axis switch: press Axis X, Axis Z, Axis S to switch the displayed servo diagnosis and adjustment of X, Z, S.

Servo parameter

In the servo display page, press the soft key SERVO PARAM to enter the servo parameter display page. The display page is shown in Fig.3-31：

No.	data	comments	
000	315	0-9999	密码(315:用户参数 385:调电机默认参数)
001	150	1-1000	电机型号代码
002*	0	0-1	电机类型(0:同步机 1:异步机)
003	0	0-35	上电初始化显示内容
004	21	9-25	控制模式
005	0	0-2	
006	2	0-2	
007	2	0-2	
008	0	0-1000	
009	0	0-10	
010	0	0-30000	
011	2	0-11	
012	0	0-1	

Fig.3-31

The servo parameter page includes viewing the servo parameters, modifying from the CNC side and saving the servo parameters.

Axis switch: press Axis X, Axis Z, Axis S to switch the displayed servo parameters of X, Z, S.

Modifying parameters: press INPUT to input a parameter value or directly input a parameter value, and then press INPUT to complete the modification.

Saving a parameter: after a servo parameter is modified, SAVE is pressed, and the

modified parameter value remains unchanged after the servo is turned on again.

Reading a parameter: press RELOAD to recover the parameter from the servo EEPROM motor parameter area.

Backuping a parameter: after a parameter is modified without a mistake, BACKUP is pressed to save the parameter to EEPROM backup area.

Recovering a parameter: press RECOVER to recover the parameter backup in EEPROM backup area.

Searching a parameter: press NO.SRH to input a parameter number, and then press the confirmation key.

Note: when the system is in MDI mode, the parameter switch ON and the operation authorization is more than [3], the parameter modification can be executed, but some parameters cannot be modified in the CNC or be done at the more than [3] operation authorization.

Servo allocation

In the servo display page, press SERVO CONFIG to enter the servo allocation display page. The display page is shown in Fig.3-32:

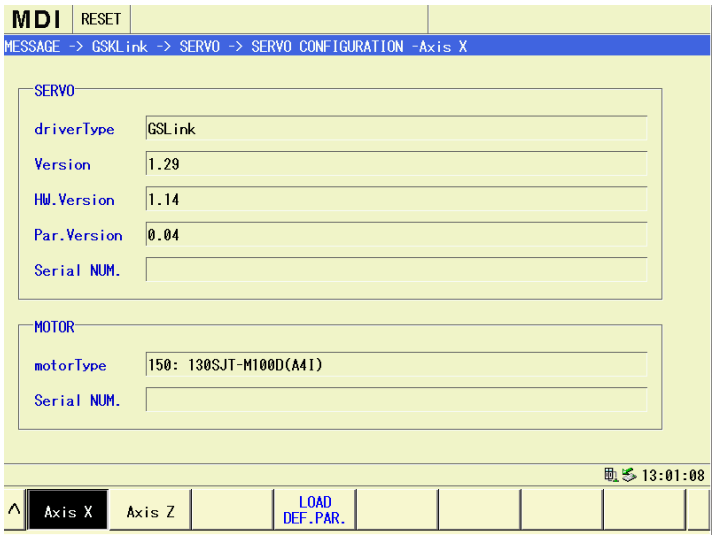


Fig.3-32

The servo allocation page can view the servo driver, the servo motor's basic message window and recover the motor's default parameters.

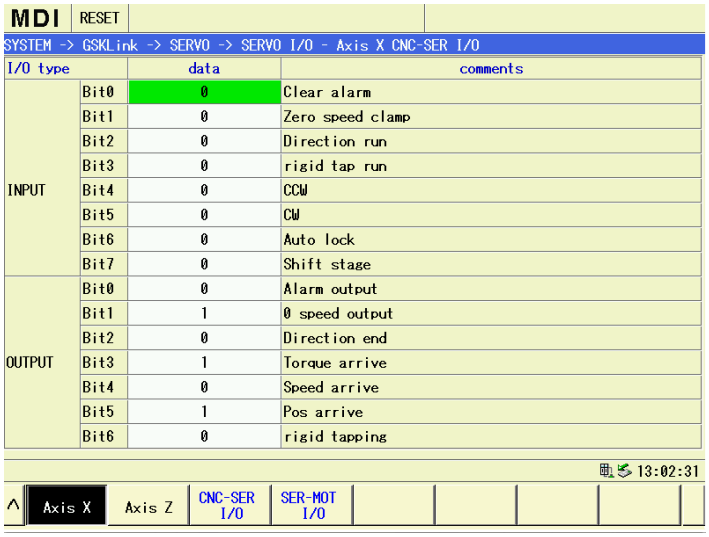
1. Basic messages of the servo and motor include the following contents
   Servo: servo type, software version, hardware version and servo serial number.
   Motor: motor type and motor serial number.
2. Recover the motor's default parameters

   After confirming the servo and motor's basic messages correct, press LOAD DEF.PAR. to debug the motor's default parameters.。

Note: the operator cannot recover the motor's default parameters at will. If necessary, contact with our technicians.

Servo I/O

In the servo display page, press SERVO IO to enter the servo I/O display page. The display page is shown in Fig.3-33：



I/O type		data	comments
INPUT	Bit0	0	Clear alarm
	Bit1	0	Zero speed clamp
	Bit2	0	Direction run
	Bit3	0	rigid tap run
	Bit4	0	CCW
	Bit5	0	CW
	Bit6	0	Auto lock
	Bit7	0	Shift stage
OUTPUT	Bit0	0	Alarm output
	Bit1	1	0 speed output
	Bit2	0	Direction end
	Bit3	1	Torque arrive
	Bit4	0	Speed arrive
	Bit5	1	Pos arrive
	Bit6	0	rigid tapping

Fig.3-33

The servo I/O page is to view the servo driver's internal I/O signal state, among which I/O is divided into hardware I/O and bus I/O:

1. Hardware I/O：

   Hardware I/O inputs/outputs I/O signals by the servo driver CN1 interface.

2. Bus I/O

   Bus I/O is defined that the servo receives or sends the CNC's I/O signals by the bus interfaces.

Servo debugging

In the servo display page, press SERVO TUNE to enter the servo debugging page. Observing the motor speed waveform or circular degree test chart by the servo debugging function in the page can ensure the servo parameter is the most reasonable and optimization.

Oscilloscope

In the servo display page, press OSCILLO GRAPH to enter the oscilloscope page. Before using the oscilloscope, set the monitoring servo data, oscilloscope monitor type, waveform zooming unit and the trigger sampling's sampling unit. The oscilloscope setting page is shown in Fig.3-34:

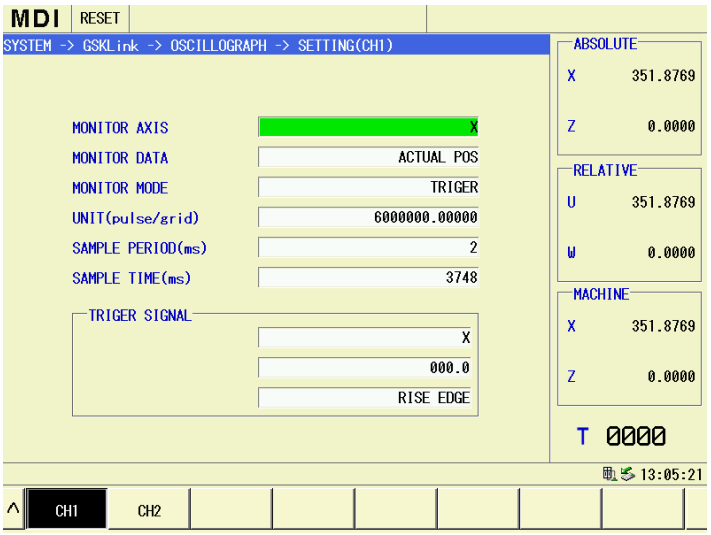


Fig.3-34

The set contents are described in the followings:

（1）“CH1”, “CH2”：select the required set channel.

（2）Monitor mode: set the oscilloscope to a trigger or memory type. The trigger type is to set a sampling described in the above to realize the arrival-time stop’s sampling mode, and the memory type is to stop sampling’s sampling mode after the servo alarm occurs.

Differences of the two monitor modes are described in the following:

Property type	Sampling start mode	Sampling end mode	Save the wave data or not
Trigger type	Press start/stop soft key	Automatic stop when the sampling time is arrival	No
Memory type	Press start/stop soft key	Automatic stop when the servo alarms	Automatic save

（3）Sampling period: GSKLink communication function’s sampling period is fixed at 60ms which can not be modified.

（4）Set other relevant data. The set items and contents are shown below:

Item	Explanation	Setting step
MONITOR AXIS	select the current wave monitor data’s axis	Press INPUT to open the option box, and then press “UP” or “DOWN” key on the MDI panel to select the option, at last press INPUT to complete the option setting
MONITOR DATA	Select the current waveform monitoring servo data according to the required	Ditto

	including: Code position Feedback position Code speed Feedback speed Servo temperature Servo current	
UNIT(pulse/grid)	Set the waveform's unit in the vertical axis display. Taking an example of code position: Setting to 5000 means the height in the oscilloscope's background gridding means 5000 pulses	Directly input the digit and then press INPUT to complete the modification
SAMPLE PERIOD(ms)	Set the sampling time limit of the trigger oscilloscope.	Cannot modify it

After setting the oscilloscope data in the setting page, press MONITOR to enter the oscilloscope monitor page as Fig.3-35:

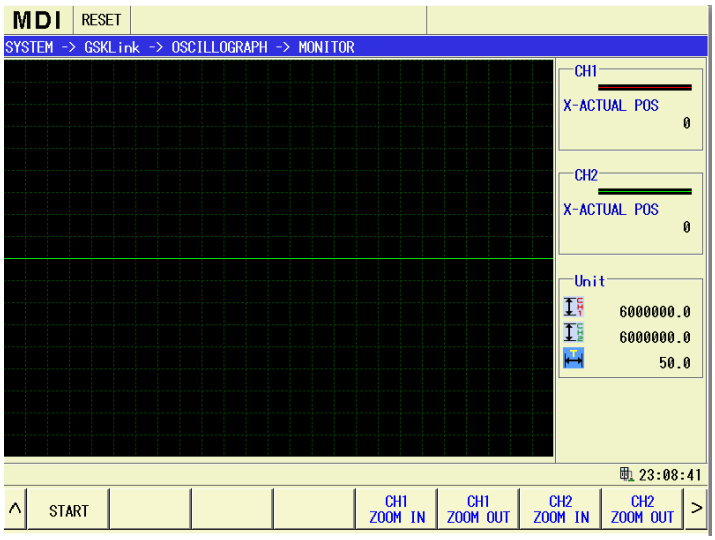


Fig.3-35

Press the soft key "Start" to start sampling the servo data in the oscilloscope page. When the sampling data is 0, the waveform depicts from the left edge's center point to the right edge point-by-point, which path means the monitor data's 0 value separate position. The monitor data's + value is on the upward side of 0 value separate position, - value is in the below of 0 value separate position. The condition is the same with the real oscilloscope display.

Press "Stop" to end the sampling when sampling. For the memory oscilloscope, manually pressing automatically save the last 1500 points' sampling data.

Chart explanation:

: CH1（channel 1）wave

: CH2（channel 2）wave

: CH1 data unit

 : CH2 data unit

 : time axis unit

3.3.6.2 I/O Unit Page

Press SYSTEM, and then GSKLink to enter GSKLink page set. Press ▶ and then press I/O unit to enter I/O unit display page. I/O unit display page mainly includes I/O allocation, I/O parameter subpage. The user can view the displayed content in subpages by corresponding soft keys. The displayed page is shown in Fig.3-36.

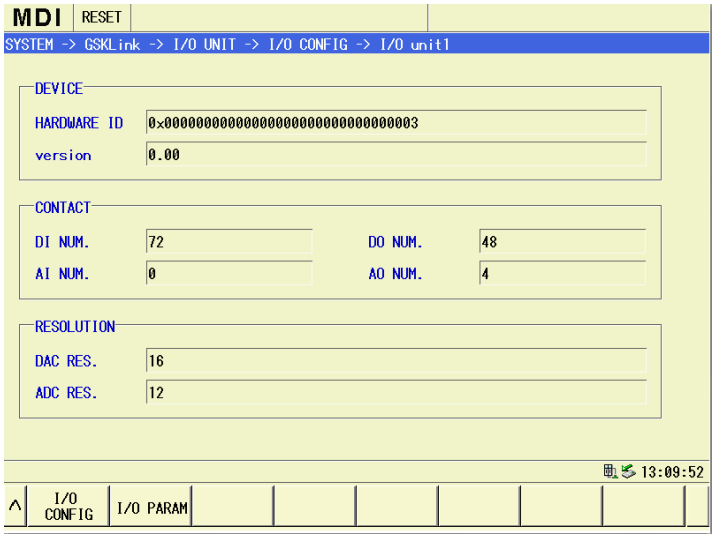


Fig.3-36

As the above figure, the user can view I/O hardware device message, port message and precision message in I/O allocation page.

Hardware device message includes: hardware ID and version number；

Port message includes: DI quantity, DO quantity, AI quantity and AO quantity；

Precision message includes: DAC precision and ADC precision.

In I/O display page, press I/O PARAM to enter I/O unit parameter setting page as Fig.3-37：

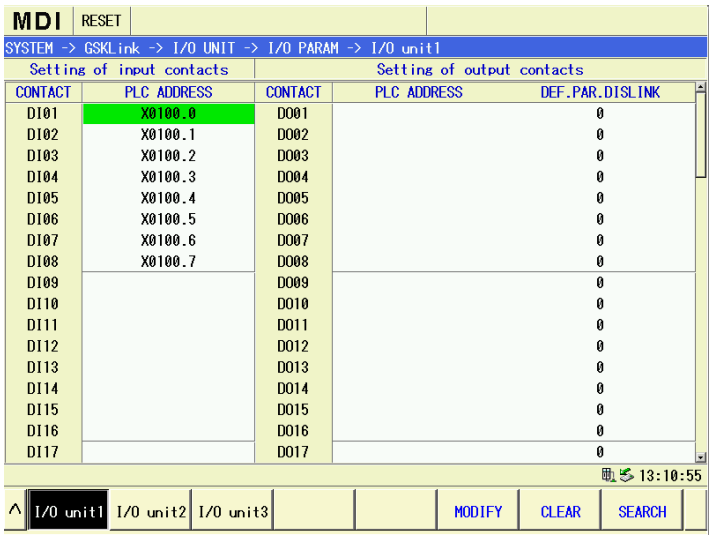


Fig.3-37

In the page, the user can view or set I/O unit's each input/output mapping address signals, special I/O, valid Level, special output, default state during emergency stop and default state's message during the loop OFF.

In I/O parameter page, press , and the cursor movement key , , , , to select the required set I/O parameter; pressing SEARCH appears the 4 soft keys SEARCH DI, SEARCH DO, SEARCH AI, SEARCH AO, and press the corresponding key to view the corresponding parameter.

Note: the concretely modifying I/O steps are referred to GSK988TA Installation and Connection.

3.4 Setting Page Set

Press SETTING function key to enter the setting page set, and the setting page set includes the subpages of setting the tool offset, setting CNC and the macro variables, and the contend displayed in each page by pressing the corresponding software key. The software key layer structure is shown in Fig.3-38:

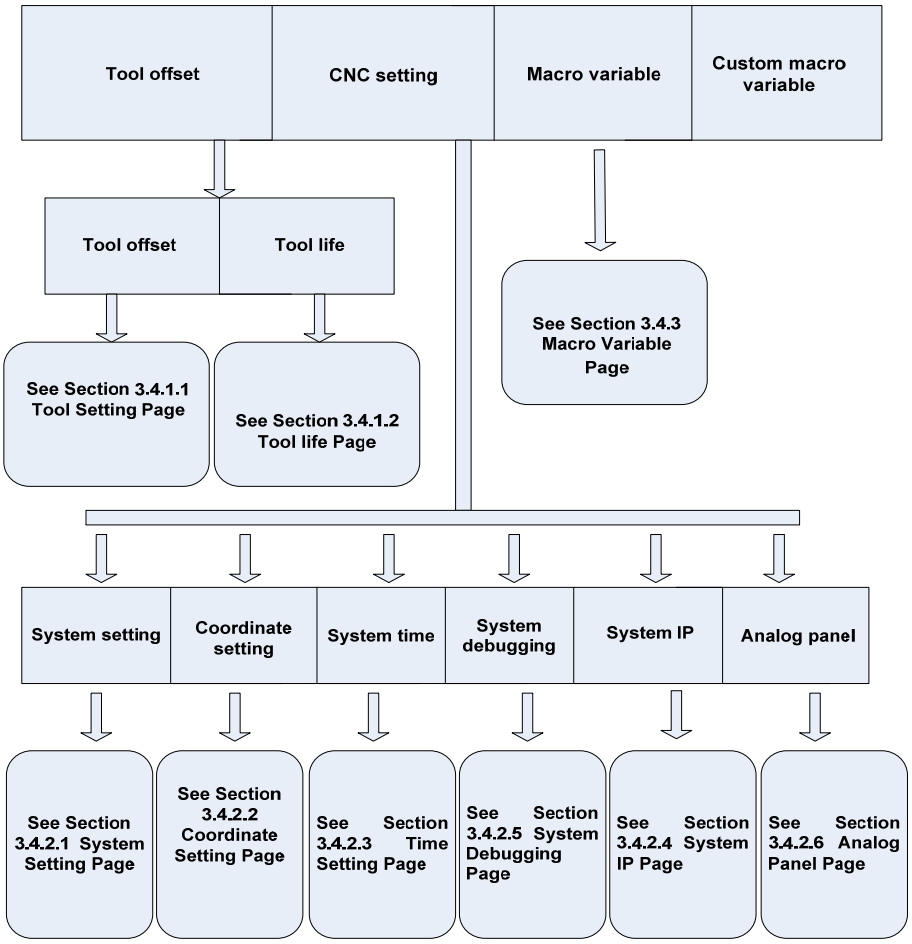


Fig.3-38

3.4.1 Tool Offset Setting

3.4.1.1 Tool Offset Setting

Press TOOL SETTING software key to enter the tool offset setting page, which is shown in Fig. 3-39:

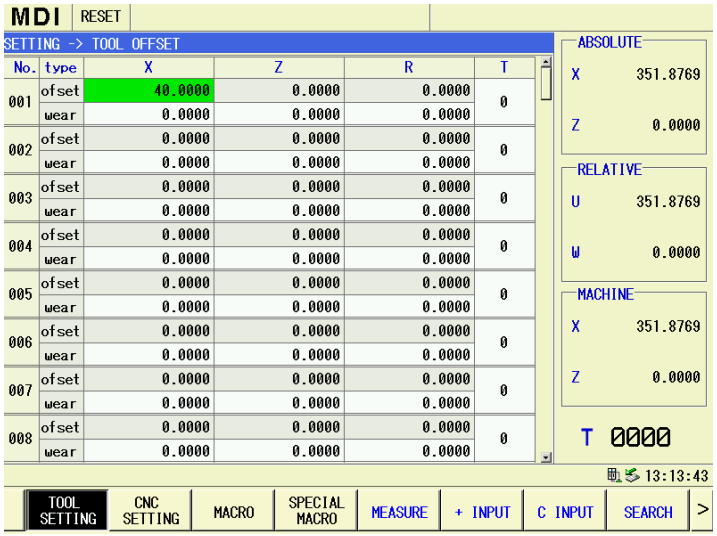


Fig.3-39

On the page, the tool offset value and the wearing value of each axis corresponding to each tool offset number should be checked and set; About the detailed setting method, refer to Chapter 7.

On the right column of the page of setting the tool offset, the message like the current absolute coordinate, the relative coordinate value and the tool number operated by the current program are displayed meanwhile.

Note 1: The axis number displayed in the page is set by the parameters #1010 and #8130.

Note 2: The linear axis or the rotary axis is specified by the axis property of each axis set by the parameter #1022 (It can't be 0).

Note 3: Each axis name is set by parameter 1020.

Note 4: Whether the tool offset value is specified by the diameter value or the radius value is set by the 1st bit of parameter #5004; Each axis movement amount is specified by the diameter or the radius is set by the 3rd bit of parameter #1006.

Note 5: Only when the operation authority is equal to or high than level [4], the tool offset and the wearing value can be set.

Note 6: The system maximum supports the tool offset of four axes; if more than four axes are displayed, only the previous four axes are displayed.

3.4.1.2 Tool Life

Press LIFE to enter the tool life set page in the tool offset page as Fig.3-40:

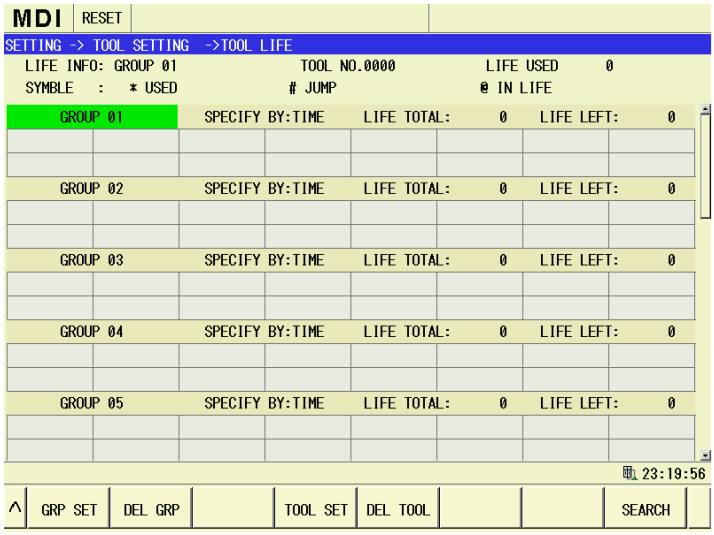


Fig.3-40

Tool tooltip: the fist line displays the tool message where the cursor is. The second line displays the tool message symbolic definition, which makes the user visually understand different symbolic tooltip's definition.

Tool group message field: the first line displays the current too message, including the current tool group, tool group's count type, tool group's preset life value, the selected tool's life used in the tool group.

Tool message field: display the tool message in the tool group.

1. Modifying the tool group data

In MDI mode, press GRP SET to pop-up a dialog box to set a tool group, press INPUT to select the tool group setting's count mode (time or times), press CHANGE to switch to the next line to set

the tool group life value as Fig.3-41:

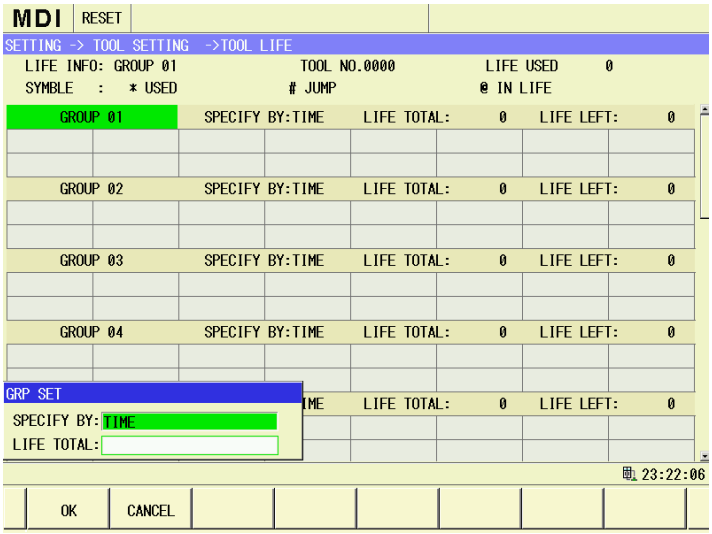


Fig.3-41

2. Tool state setting

In MDI mode, move the cursor to the tool number in the current tool group, press TOOL SET to set the current tool to the skip state or cancel the skip state.

3. Modifying the tool number

In MDI mode, move the cursor to the tool number grid, directly input the tool number modification, and then clear the current tool's tool life.

4. Deleting a tool

In MDI mode, move the cursor to the tool number grid,, press DEL TOOL to clear the current tool number, and then clear the tool number' tool life.

5. Deleting a tool group

In MDI mode, press DEL GRP to delete the whole tool group's tools, i.e., clear the whole tool group's tool numbers, and then clear the tool group's preset life.

6. Searching a tool

Press SEARCH to input a tool number and then search the tool number.

Note 1: The tool group's quantity is together determined by No. 6813 and No. 6800#0，#1. When No.6813 setting is less than 8，the total is defaulted to 128 groups.

Note 2: The set tool group life and tool group's count type cannot be reset, their life can be reset after the tools are deleted.

Note 3: No. 8132#0 must be set to 1 in the tool life page: it is displayed when the tool life management function is used.

3.4.2 CNC Setting Page

On setting page set, press CNC SETTING software key to enter CNC system setting page and it mainly includes the system setting, the coordinate setting, the system time and the system IP.

3.4.2.1 System Setting Page

On CNC setting page set, press SYSTEM to enter the system setting page and it includes setting the program switch, the parameter switch, the automatic sequence number and the inputting unit, etc, which is shown in Fig.3-42:

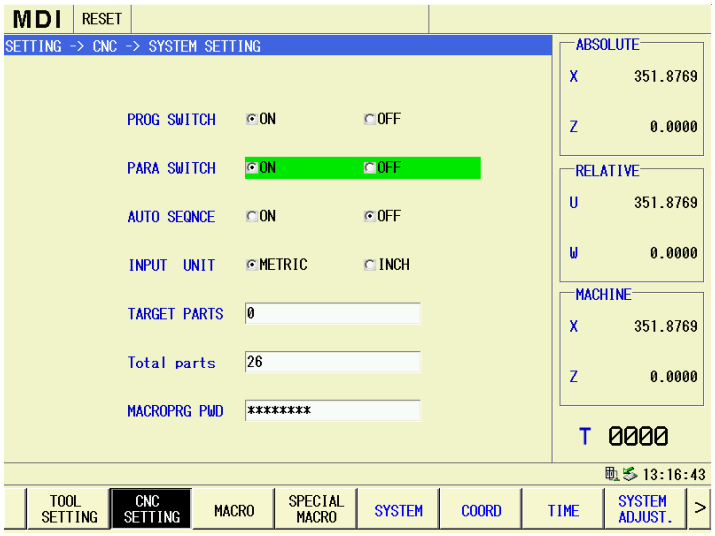


Fig.3-42

The page is mainly for setting ON/OFF of the program and the parameter switches, etc.

On the page, the program switch, the parameter switch, the automatic sequence number and the input unit can be switched by ⇧ and ⇩ key. In MDI status, when the operation authority level is equal to or higher than level [3], ON/OFF of the switch, inch/metric system can be selected by pressing ⇦, ⇨ or INPUT keys.

On the right column of the page, the current absolute position coordinate, the relative coordinate position value and the tool number used in the current program are displayed meanwhile.

Note 1: Only when the program switch or the parameter switch is ON, the program or the parameter can be edited, rewritten or set.

Note 2: Only when the operation authority is equal to or higher than level [3], CNC system can be set.

Note 3: Only when the program protection switch on the machine panel is ON, the program and the parameter switches ON/OFF can be set.

Note 4: Only when the operation authority is equal to or higher than level [2], the password of macro programs can be modified.

3.4.2.2 Coordinate Setting

Press SETTING function key to enter the setting page set; On CNC setting page, press COORD software key to enter the coordinate setting page, which is shown in Fig.3-43:

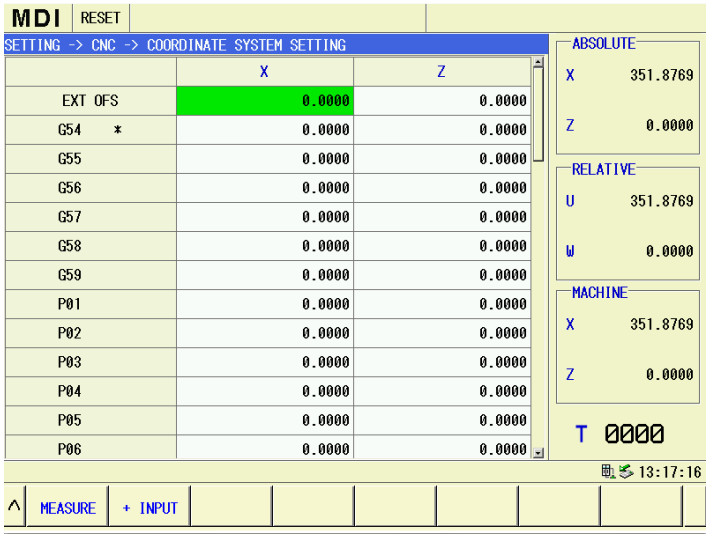


Fig.3-43

On the coordinate setting page, the origin offset amount of each axis and the offset value of each coordinate axis in each coordinate system are displayed, the origin offset amount corresponding to each axis and the offset value of each coordinate axis of each coordinate system can be set.

On the coordinate setting page, the coordinate system to be set is selected by the up/down keys ⇧, ⇩, and the coordinate axis to set the offset is selected by left/right keys ⇦, ⇨, there are three methods of rewriting the tool offset: Direct inputting, measuring inputting and + inputting:

Direct inputting: Select the coordinate axis to be rewritten, press INPUT key, and then input the offset value, and then confirm.

Measuring inputting: Select the coordinate system to be rewritten, press MEASURE to input the measured value (if it is X axis, input X--; Z axis, input Z---), and then confirm.

+input: Rewrite the input offset value, and it is the incremental input. For example, in G54 coordinate system, X axis needs to be added up the offset value of –0.2mm, firstly, move the cursor to X axis position in G54 coordinate system, and then press + INPUT, and input –0.2, and –0.2mm is added into the current offset value after confirming.

On the right column of the page, the current absolute position coordinate value and the relative one, and the tool number used by the current program, etc are displayed meanwhile.

Note 1: The coordinate offset can be set or rewritten only when the parameter switch is ON and in MDI modal and the operation authority is equal to or higher than level [4].

Note 2: The axis number displayed on the page is set by the parameters #1010 and #8130.
Note 3: The name of each axis is set by the parameter #1020.

Note 4: The origin offset amount of each coordinate in each coordinate system can be set by a parameter, and their corresponding relationship is shown below:

No. 1220: offset amount of each axis' external workpiece origin
No. 1221: offset amount of each axis' origin workpiece coordinate system 1 (G54)
No. 1222: offset amount of each axis' origin workpiece coordinate system 2 (G55)
No. 1223: offset amount of each axis' origin workpiece coordinate system 3 (G56)
No. 1224: offset amount of each axis' origin workpiece coordinate system 4 (G57)
No. 1225: offset amount of each axis' origin workpiece coordinate system 5 (G58)
No. 1226: offset amount of each axis' origin workpiece coordinate system 6 (G59)

3.4.2.3 Setting the System Time

Press SETTING function key to enter the setting page; on CNC setting page, press TIME software key to enter the system time setting page, which is shown in Fig.3-44:

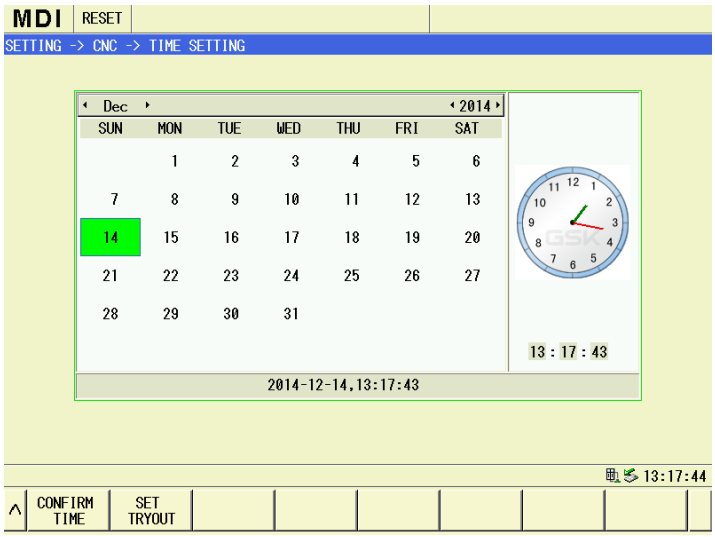


Fig.3-44

On the time setting page, it can be switched among the date, month, year and time columns by pressing CHANGE key.

Setting the month: Press CHANGE to switch the cursor into the month column, and then the month column is changed into the green one, press the cursor movement keys ⇧, ⇩, ⇦, ⇨ to change the month and press CHANGE to move the cursor into the other columns to complete setting the month.

Setting the year: Press CHANGE to switch the cursor into the year column, and then the year column is changed into the green one, press the cursor movement keys ⇧, ⇩, ⇦, ⇨ to change the year and press CHANGE to move the cursor into the other columns to complete setting the year.

Setting the date: Press CHANGE to switch the cursor into the date column, and then the date column is changed into the green one, press the cursor movement keys ⇧, ⇩, ⇦, ⇨ to change the date, finally press CONFIRM TIME to complete setting the date.

Setting the time: Press CHANGE to switch the cursor into the time column, and then the time column is changed into the green one, input the time to be set in the edit keypad, and then press the cursor movement keys ⇦, ⇨ to set the hour, minute and seconds, finally press CONFIRM TIME software key to complete setting the date.

3.4.2.4 System IP Setting

Press SETTING function key to enter the setting page set; on CNC setting page, press ETHENET software key to enter the system IP setting page, which is shown in Fig.3-45:

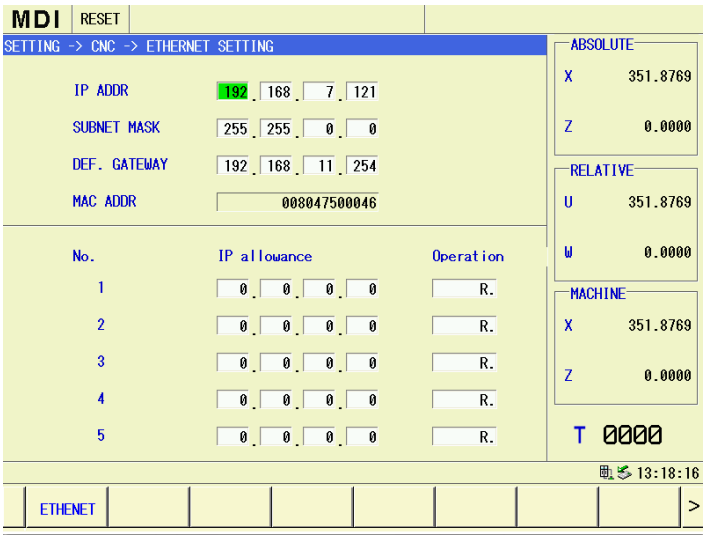


Fig.3-45

(1) Press the up/down keys ⇧, ⇩ to switch among IP address, subnet mask and the default gateway bar.

(2) Press the left/right keys ⇦, ⇨ to switch in each address bar of each address and then input the address to be set.

3.4.2.5 System Debugging Function

Press SETTING to enter the setting page set; press SYSTEM ADJUST. in the setting page set to enter the system debugging page.

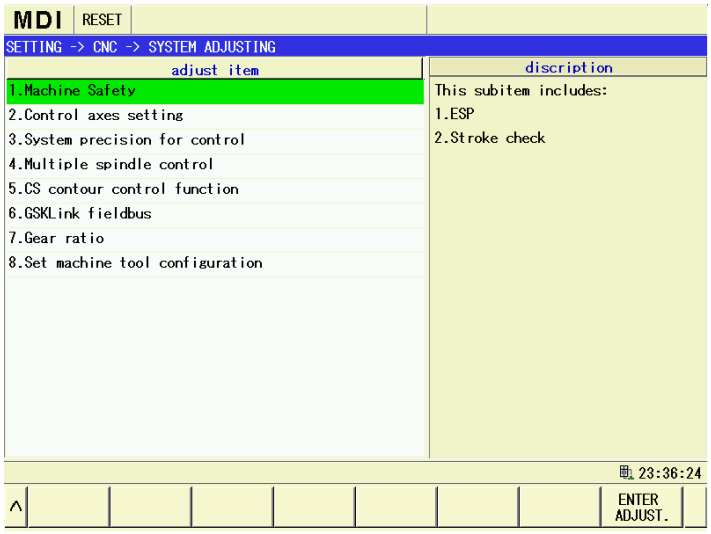


Fig.3-46

Machine safety protection and external switch

Move the cursor to the machine safety protection and the external switch debugging, and then press ENTER ADJUST. to enter the debugging page as Fig.3-47：

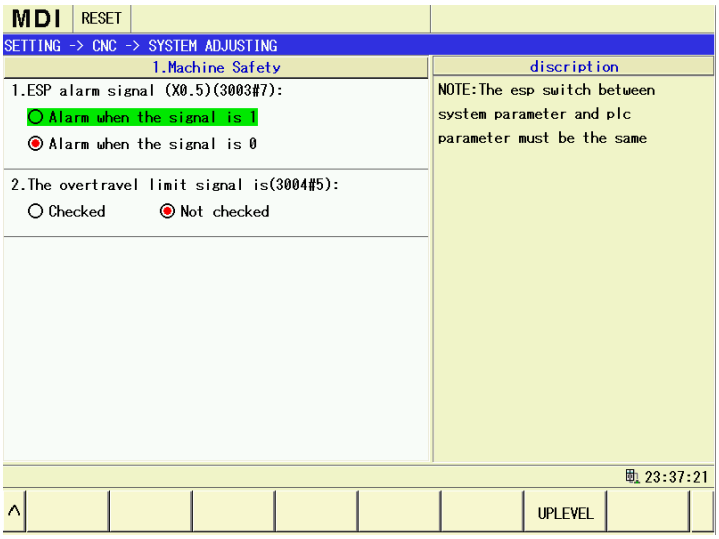


Fig.3-47

Setting method:

Function optional type: move the cursor to the required option, and then press INPUT in the edit keyboard.

Numerical value type: move the cursor the required option, press INPUT to input a numerical value, and then press INPUT to complete the setting.

Press {return} or [cancel] on the edit keyboard to return the previous page.

Electronic gear ratio setting

Move the cursor to the electronic gear ratio setting, and press ENTER ADJUST. to enter the electronic gear ratio setting page as Fig.3-48:

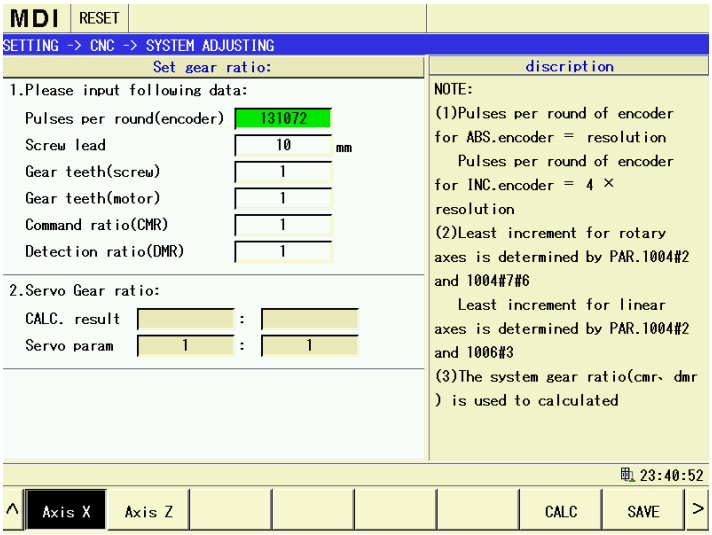


Fig.3-48

View and set the axis' gear ratio in the page, and the set gear ratio is automatically saved to the corresponding driver.

（1）Select the required set axis as X axis in the above figure；

（2）Ensure the one-rotation pulse quantity of the encoder in data bar is correct (after the bus connection is normally, one-rotation pulse quantity of the encoder can be read from the driver);

（3）Input the roll screw's lead in the lead bar. Example: when the lead is 10mm, 10 is input as the above figure;

（4）When the lead screw: the motor is not 1:1, a gear quantity between the lead screw and motor. When the ratio is 1:1, it is not input because the system defaults the ratio value;

（5）The defaulted gear ratio is 1:1，and the ratio between the system checking multification ratio and code multification ratio should be set if necessary;

（6）After the above data is set. CALC is pressed，and the system automatically counts the rear ratio and is displayed in the result. Then press SAVE and the system automatically save the result to the corresponding servo driver. SAVE is not pressed when only the

result not to be saved is needed.

3.4.3 Macro Variable Page

On the setting page set, press MACRO software key to enter the macro variable setting page, which is shown in Fig.3-49.

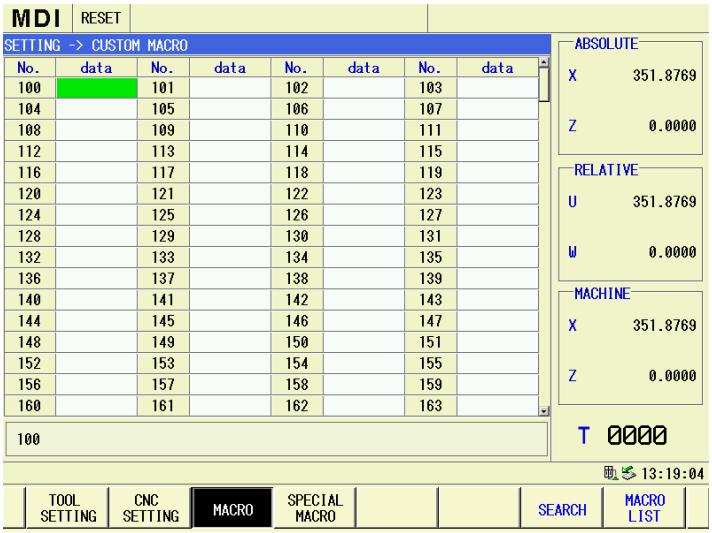


Fig.3-49

On the macro variable page, the value corresponding to each macro variable can be checked and set.

On macro variable page, press LOCAL MACRO, PUBLIC MACRO, SYSTEM MACRO to select the type of the variable, and press the page keys, the direction keys, to select the macro variable to be rewritten, and the base color of the selected macro variable is changed into green; or press SEARCH software key to input the macro variable sequence number to be selected, press OK and then the cursor is positioned into the macro variable data.

In MDI mode, when the operation authority level is equal to or high than level [4], the macro variable data can be rewritten with the number keys and the backspace key; or the macro data can be rewritten by pressing INPUT key, and the macro data can be rewritten by the number keys and the backspace key, and the rewriting is done after pressing INPUT key.

3.5 Message Display Page Set

Press MESSAGE function key to enter the message page set, and the message interface is with three pages: the alarm message, the previous record, and the diagnosis, and the content displayed in each page can be checked by pressing the corresponding software keys. Its software layer structure is shown in Fig.3-50:

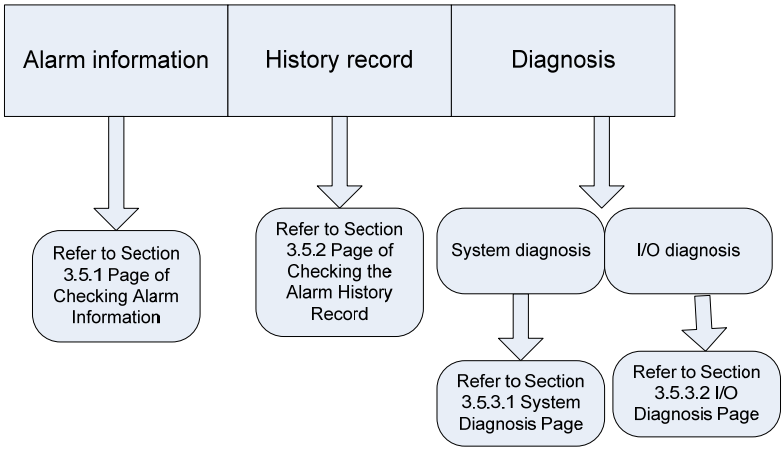


Fig.3-50

3.5.1 Alarm Message

On the message page set, press ALARM MESSAGE software key to enter the alarm message display page to display the number of CNC alarms and PLC alarms and the detailed message, and the page is shown in Fig.3-51:

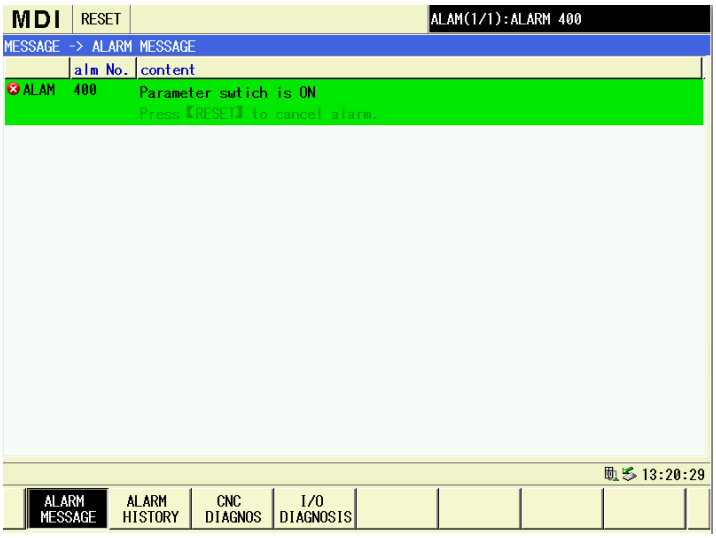


Fig.3-51

On the alarm message display interface, the alarm and the reminder of CNC and PLC message are listed in one window list and it is divided by the alarm number. Press up/down keys ⇧, ⇩

to scroll the list line-by-line, and the page keys , to scroll the list page-by-page.

When PLC alarm or reminder occurs, the address A message below the message line is displayed; when CNC alarm or reminder occurs, the reason and the troubleshooting are displayed below the message line.

Clearing the alarm: Press RESET to cancel the alarm. About some alarms, refer to the reason and troubleshooting.

Note 1: When PLC alarm or reminder occurs, the address A message is displayed in black color below the message line;

Note 2: When CNC alarm or reminder occurs, the reason and the troubleshooting is displayed in black color below the message line.

Note 3: The alarms of #0——#1000 are CNC ones; the alarms of #1000——#2000 are PLC ones, those after #2000 are reminder message.

Note 4: When the parameter becoming valid after power-on is rewritten, the alarm can be cleared only after power on, again.

Note 5: Please refer to Appendix One Alarm Message List and Appendix Two PLC Alarm Explanation about the detailed alarm message and PLC alarm.

3.5.2 History Record

Press MESSAGE key to enter the message interface and then press ALARM HISTORY software key to enter the previous record interface. The interface records the latest previous alarm message, which includes the alarm date, the alarm time, the alarm number and the alarm content. The alarm note message can be checked by pressing , , , , the page is shown in Fig. 3-52:

Fig.3-52

Clearing the previous alarm record: On the page of the previous alarm record, press CLEAR ALARM software key to clear the previous records of all alarms and the reminder message,

and the page is blank after clearing.

Note: Whether clear the alarm record is set by parameter 3110.2.

3.5.3 System Diagnosis

Press MESSAGE key to enter the message interface and press CNC DIAGNOS software key to enter the diagnosis page. And then, press System diagnosis button to select the pages of different diagnosis classification data, which is shown in Fig.3-53:

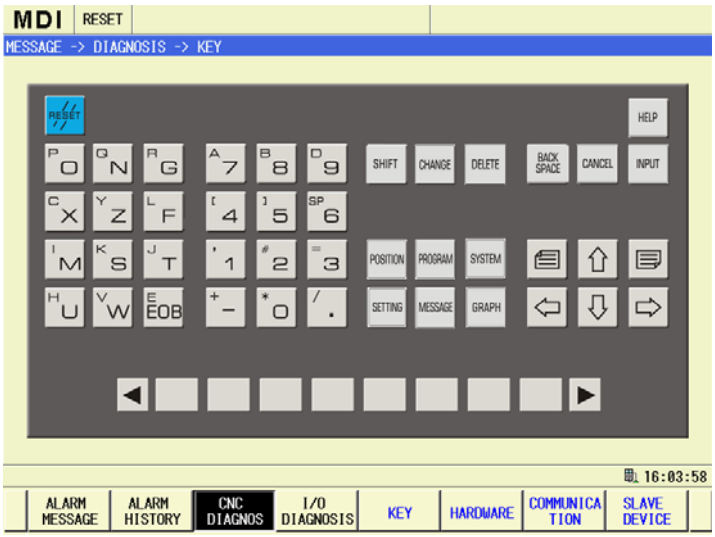


Fig.3-53

The system diagnosis display page includes the CNC's edit keyboard, hardware interface, bus communication and servo data. Press the corresponding soft keys to switch to the corresponding page.

Edit Keypad Diagnosis

After entering CNC diagnosis page, press CNC DIAGNOS software key to enter the system diagnosis page, and then press KEY to enter the edit keyboard diagnosis page, which is shown in Fig.3-54

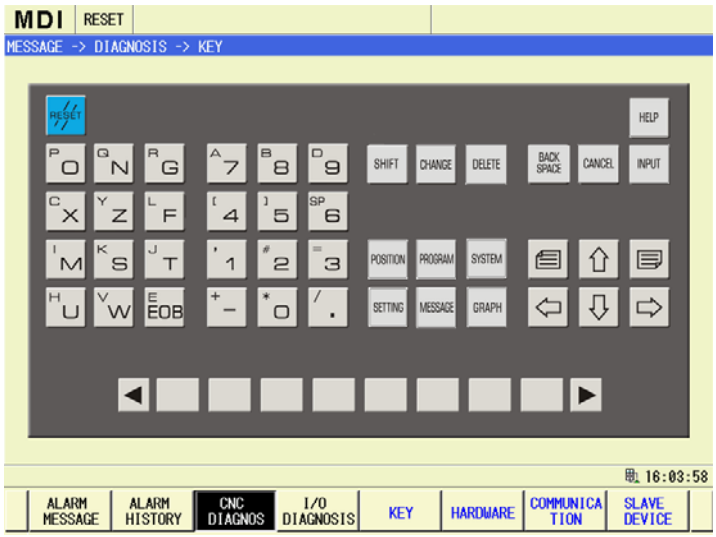


Fig.3-54

The edit keyboard diagnosis page mainly disgnosizes CNC edit keyboard is normal or not. Press a key on the edit keyboard, and the corresponding diagnosis message becoming 1 from 0 on the screen means the key is normal, otherwise it is not normal. Pressing , , , , , , in the page can view the corresponding key's message. To avoid of creating a corresponding function operation during viewing some key (such as direction key and page up/down key)'s diagnosis message, press LOCK SCREEN to lock the current screen.

In the edit keyboard diagnosis display page, there are two line of diagnosis number's concrete content display line at the bottom of the page, the first line displays the diagnosis number; the second line displays some bit's definition of the diagnosis number where the current cursor is.

Hardware Interface Diagnosis Page

In the system diagnosis page, press HARDWARE to enter the CNC hardware interface's diagnosis page as Fig.3-55:

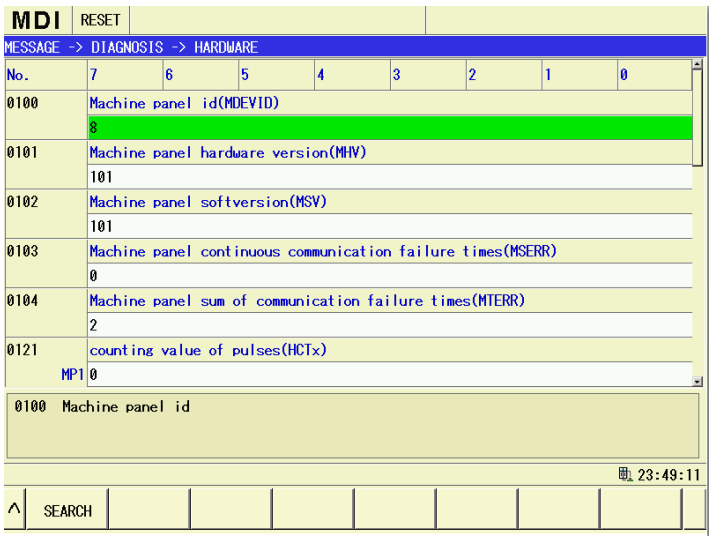


Fig.3-55

The hardware interface diagnosis display page mainly disgnosizes CNC's each hardware's version number, mistaken messages and hardware count. Press , , , or perform the search function to view the corresponding message.

There is a detailed diagnosis message content display at the bottom of the hardware interface diagnosis display page.

Bus Communication Page

In CNC diagnosis page, press COMMUNICATION software key to enter the CNC bus communication diagnosis page, which is shown in Fig.3-56:

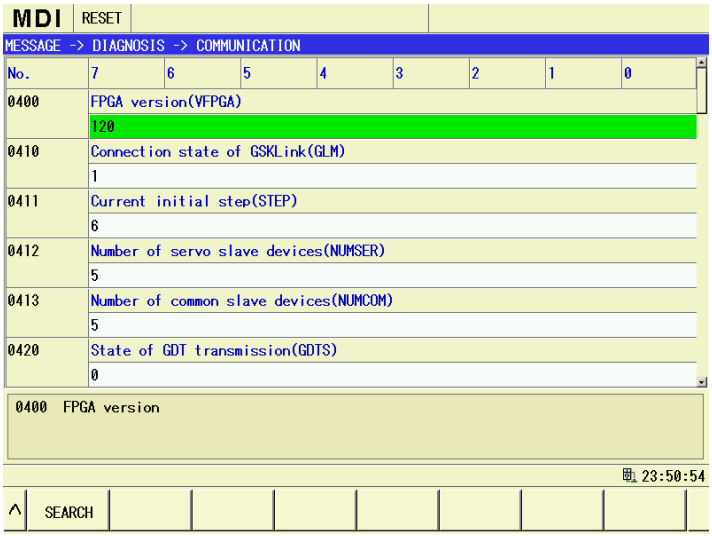


Fig.3-56

The bus communication diagnosis display page mainly diagnosizes connection states between the CNC and each bus communication device, and the bus sending data's messages. Press , , , or perform the search function to view the corresponding message.

There is a detailed diagnosis message content display at the bottom of the bus communication diagnosis display page.

Servo data diagnosis

Press SLAVE DEVICE to enter the servo data's diagnosis page as Fig.3-57：

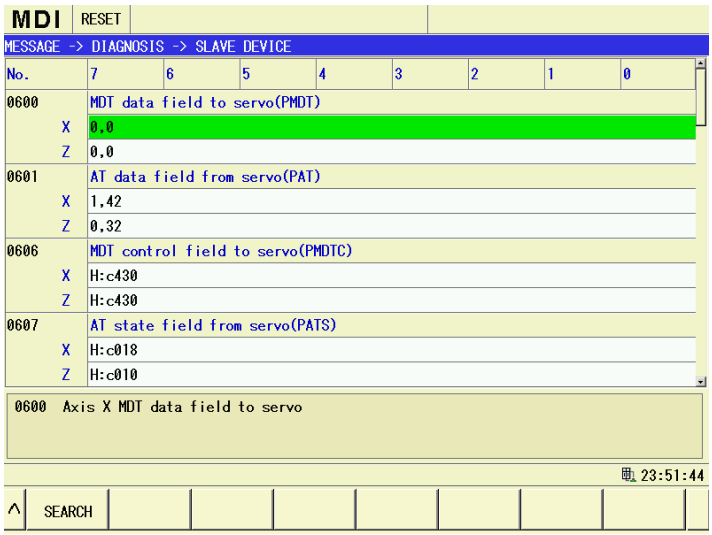


Fig.3-57

The servo data diagnosis display page mainly diagnosizes each servo slave station, and the bus communication's data message between the I/O unit's slave station and the CNC's bus communication. Press , , , or perform the search function to view the corresponding message.

There is a detailed diagnosis message content display at the bottom of the servo data diagnosis display page.

3.5.4 I/O Diagnosis

In CNC diagnosis page, press I/O diagnosis software key to enter I/O unit diagnosis page, which is shown in Fig.3-58:

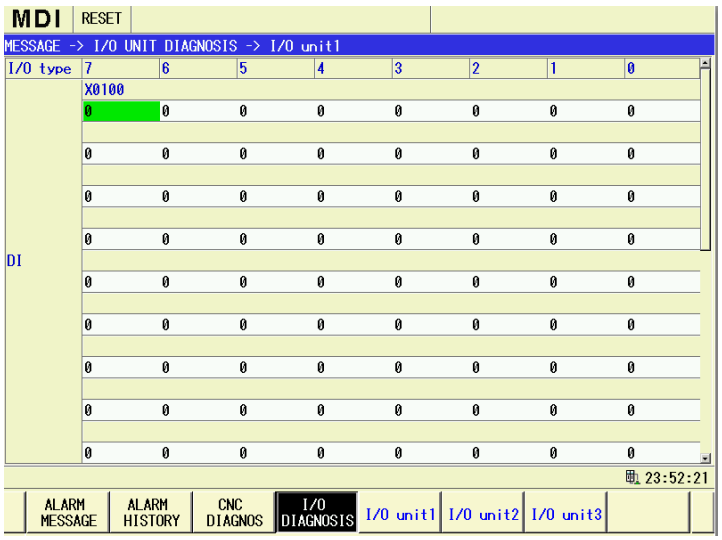


Fig.3-58

The page mainly diagnosizes the I/O unit's hardware I/O message and I/O unit's hardware's fault. When the PLC diagnosizes I/O output signal is valid and the machine output is invalid, the system can diagnosize I/O unit's hardware is fault or not. Press , , , , ,


to view the corresponding message.

3.6 Figure Display Page Set

Press GRAPH function key to enter the graph page set, and it mainly includes the subpages of setting the graph, the path display, and the graph simulation, and the content displayed in each page can be checked by pressing corresponding software key. Its software layer structure is shown in Fig.3-59:

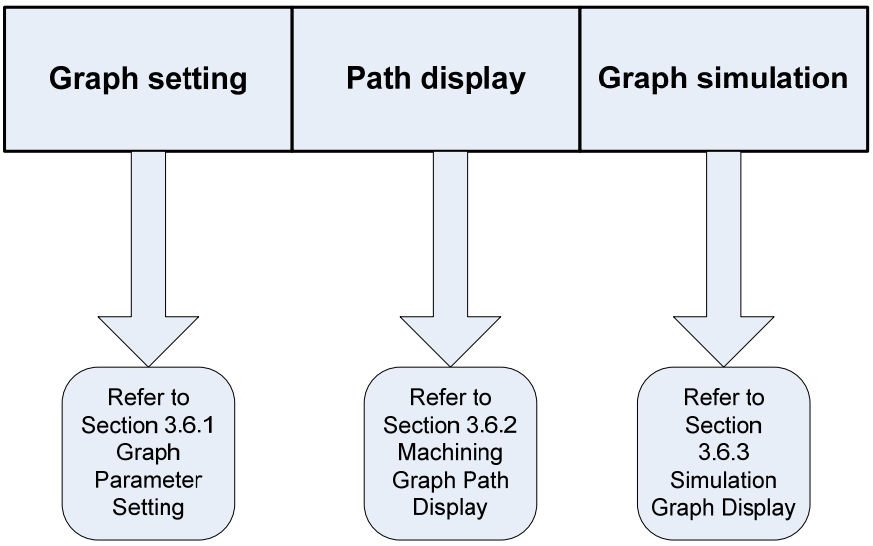


Fig.3-59

3.6.1 Setting the Graph Parameters

In the graph page set, press GRAPHSET software key to enter the graph setting page, the page is shown in Fig.3-60:

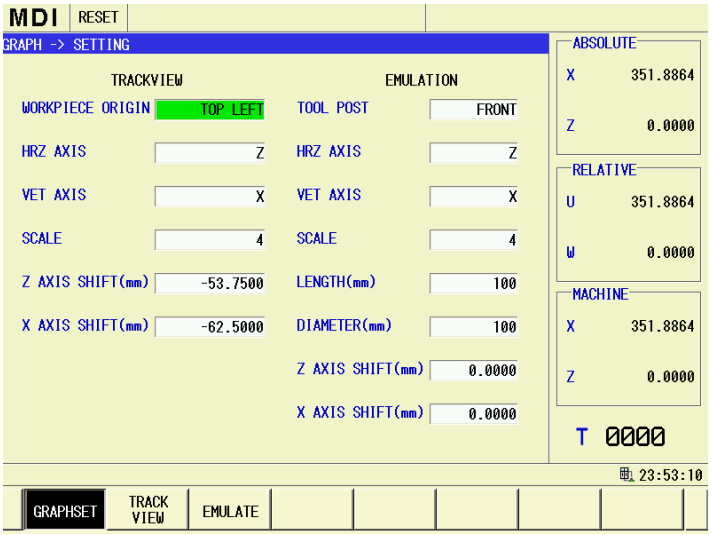


Fig.3-60

On the page, the graph path parameters can be set.

The origin position of the coordinate is set, the horizontal and the vertical axes of the graph are selected, and the offset of the coordinate axis and the magnification times of the graph are set; the horizontal and vertical axes, the graph simulation magnification scale and the coordinate axis offset should be set.

On the right column of the page, the current absolute position coordinate and the relative coordinate position values and the tool number used by the current running program are displayed meanwhile.

Press ⇧, ⇩ key to switch among each item and press ⇦, ⇨ to switch among the data of two channels. In MDI status, the graph parameters are rewritten with the digit key and the backspace key, the numerical value to be rewritten is input, and press INPUT key to confirm the setting is completed. About the detailed operation, refer to Chapter 8.1.

3.6.2 The Machined Graph Path Display

On the graph page set, press TRACK VIEW software key to enter the path display page, which is shown in Fig.3-61:

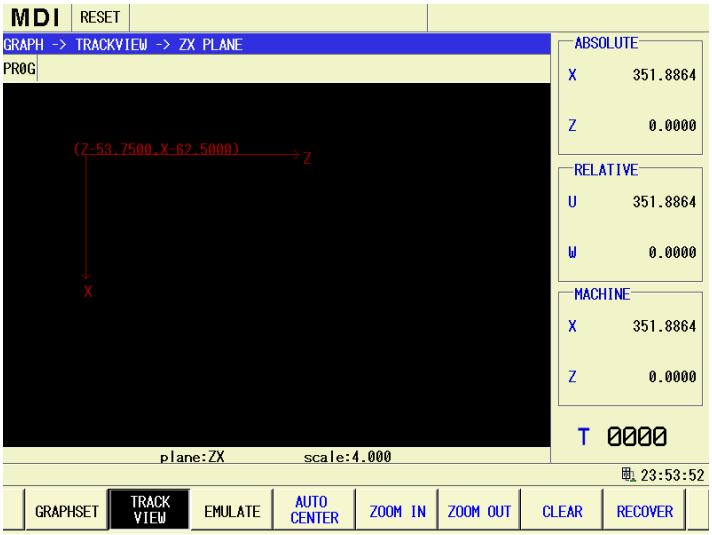


Fig.3-61

The coordinate plane displayed by the current path and the scaling of the path graph are displayed below the path display screen.

On the right column of the page, the current absolute position coordinate and the relative coordinate position values and the tool number used by the current running program are displayed meanwhile.

Then, the graph can be scaled up/down, the path can be cleared, and meanwhile, the graph can be moved upward/downward/leftward/rightward by pressing ⇧, ⇩, ⇦, ⇨ keys meanwhile.

Note: Each axis name is set by the parameter #1020 and the different letter names are set for each axis.

3.6.3 Simultaneous Graph Display

In the graph page set, press EMULATE software key to enter the simultaneous graph display page, which is shown in Fig.3-62:

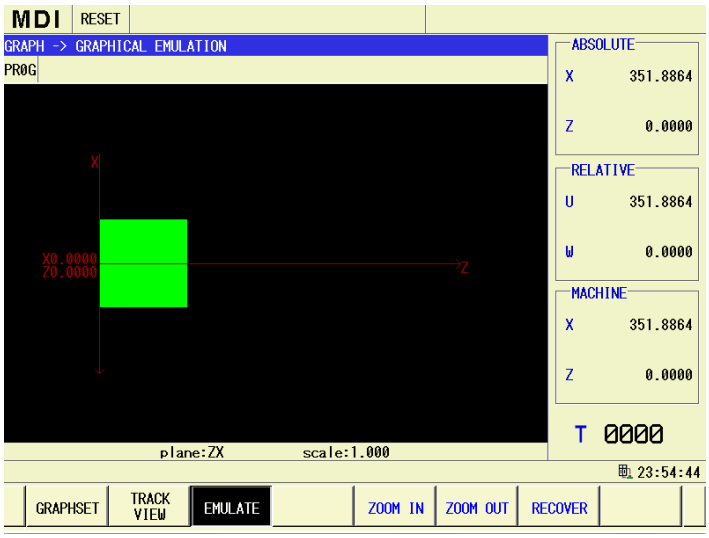


Fig.3-62

The coordinate plane displayed by the current simulation graph and the scaling of the simulation graph are displayed below the graph simulation screen.

The graph can be scaled up/down, the path can be cleared, and meanwhile, the graph can be moved upward/downward/leftward/rightward by pressing ⇧, ⇩, ⇦, ⇨ keys.

Note: Each axis name is set by the parameter #1020 and the different letter names are set for each axis.

3.7 Help Page Set

Press HELP function key to enter the help page set, which is shown in Fig.3-63. The help page set mainly includes the subpages of the operation help, the programming help, the alarm help and the parameter help, etc, and the content displayed by each page can be checked by pressing the corresponding software keys.

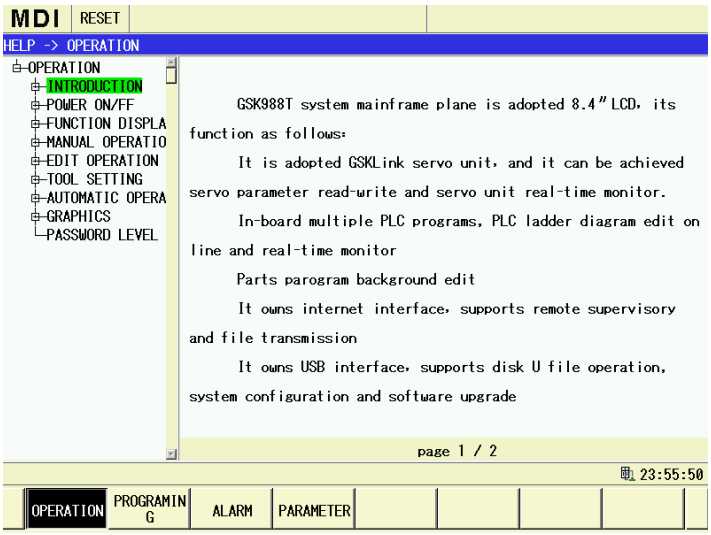


Fig.3-63

Each subpage can be divided into two parts: the left directory and the right corresponding content. The following shortcut keys can be operated:

Content: Page up key: Page up for one page in the content;
Page down key: Page down for one page in the content;

Directory: Upward key: Check the previous directory;
Downward key: Check the next directory
Leftward key: Return to the up one level directory;
Rightward key: Open the next level directory;
Switch key + page up key: Page up for one page in the content;
Switch key + page down key: Page down for one page in the content;

Moreover, the search function exists in the programming help, in the alarm help and in the parameter help, and G code, the alarm number and parameter number can be input in the corresponding interface, which is shown in Fig.3-64 below. On the programming help page, press SEARCH and input G01 in the dialogue box and press Enter. For example, G01 code can be directly found, which is shown in Fig.3-65.

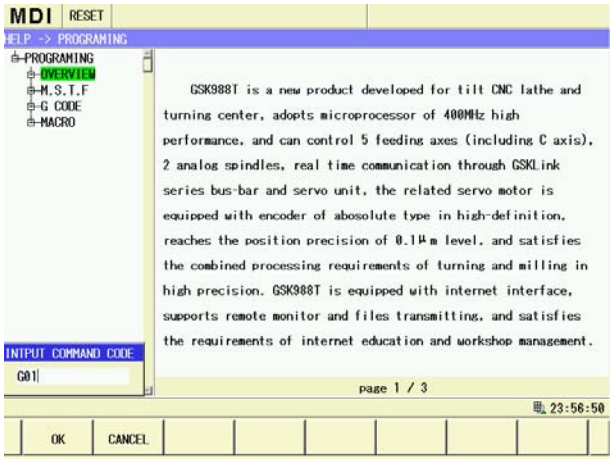


Fig.3-64

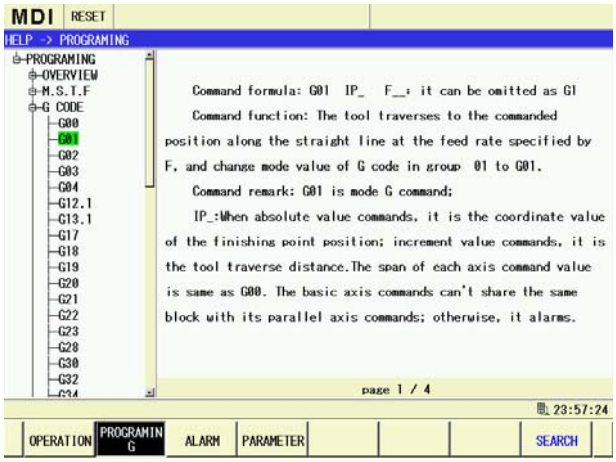


Fig.3-65

Chapter 4 Editing and Managing the Program

On the program page, the program can be searched, created, selected, copied and deleted, and the program can also be imported and exported.

To prevent the program is rewritten and deleted by accident, GSK988TA/988TA1/988TB sets the program switch. After editing and rewriting the program, the program switch must be ON; And about the details of setting the program switch, refer to Section 3.4.2.1.

Note: The file with the name 'NCPROG' is on the U disc, and the file is placed on the file, the operation on the U disc directory is consistent with that of the local directory on the program page, please refer to the operation of the local directory when the U disc directory is used.

4.1 Creating a Program

4.1.1 New a Program

When the operation authority level is equal to or higher than level [4], the program can be created and edited.

（1）Press PROGRAM function key and press LOCAL software key to enter the program page set:

（2）On the program page set, press NEW software key to enter the creating page.

（3）Input the new program name in INPUT NEW PRG NAME O, such as inputting 0123, press OK software key to enter # O0123 program editing page, which is shown in Fig.4-1:

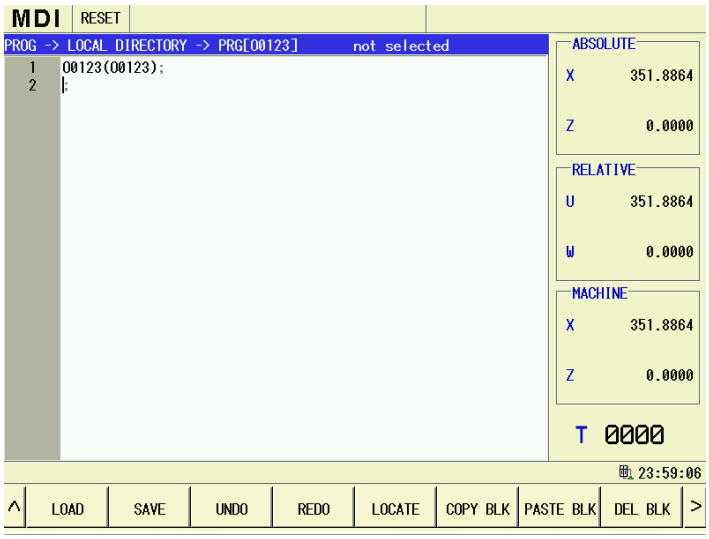


Fig.4-1

4.1.2 Opening a Program

（1）Press PROGRAM function key and press LOCAL software key to enter the program page set.

（2）On the program page set, press , , , , to move the cursor to select the program to be opened; or press SEARCH software key to search, input the program name to be opened, and press OK software key to be searched, and move the cursor to position the program name; the background of the selected program name is changed into green;

（3）Press OPEN software key and the program code of the selected program can be opened in the screen, which is shown in Fig.4-2.

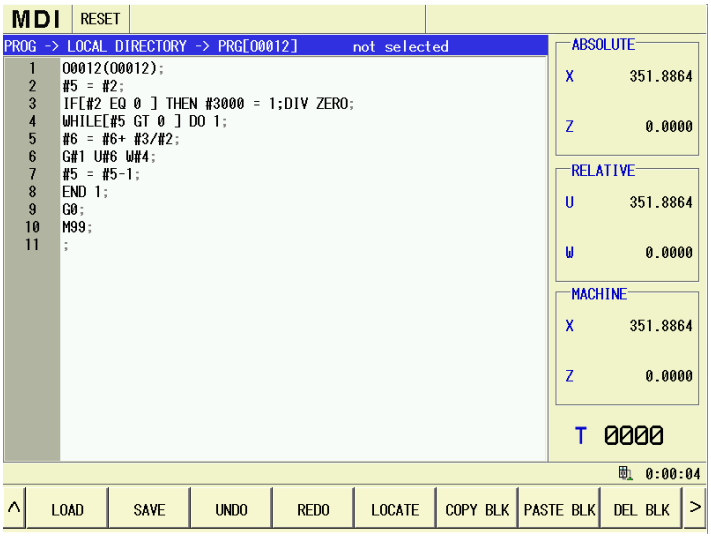


Fig.4-2

Then, the current program can be edit or rewritten, while the current executed program can only be edited in Edit mode.

Note: When the parameter 3404.6 is 0, the program must be with the end codes M02, M30 and M99, etc; otherwise, when PRG CHECK software key is pressed during checking the program, the reminder will be popped up; and the alarm occurs when the program is running.

4.1.3 Renaming a Program

On the program page set, press , to move the cursor for selecting the program and

press RENAME software key to rename the selected program. Input the new program name on the pop-up dialogue box INPUT PRG NAME / RENAME PRG00010 / O and press OK software key to rename the selected program and to return. Press CANCEL software key to cancel renaming and the system returns to the previous menu.

Note 1: No renaming the loaded or running file.

Note 2: Only when the operation authority level is equal to or higher than level [4], the program can be renamed.

4.1.4 Saving as

On the program page set, press ⇧, ⇩ keys to move the cursor for selecting the program, press SAVE AS software key to save the selected program as the other name. Input a new program name in pop-up dialogue box SAVE AS / SAVE PRG00010 AS / O and press OK software key to save the program as the other name. For example, input 2222, press CANCEL software key to save #00010 program as #O2222 one, and the cursor is jumped into the new program name, which is shown as Fig.4-3:

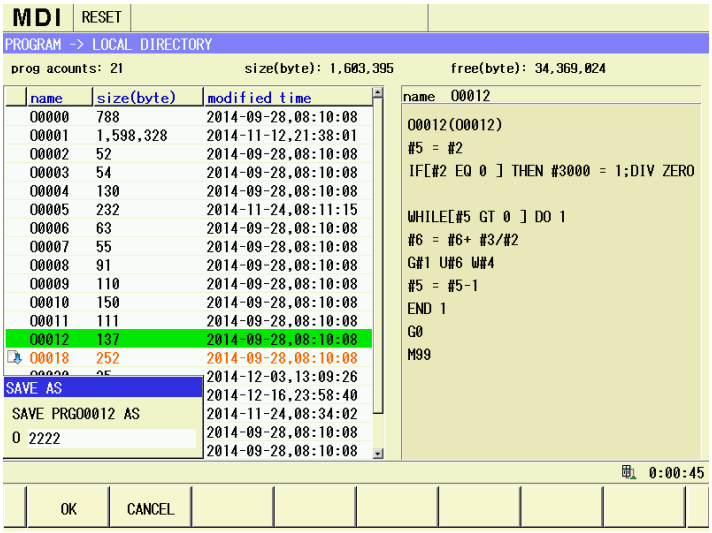


Fig.4-3

Note: Only when the operation authority level is equal to or higher than level [4], the program can be saved as the other name.

4.1.5 Deleting a Program

（1）On the program page set, press , , , key to move the cursor for selecting the program to be deleted, and the background of the selected program is changed into green.

（2）Press DELETE software key to delete the selected program.

4.1.6 Outputting a Program

The system USB port is with the USB flash disc, an U disk icon appears at lower right corner of the screen, and there is a key for an U disk catalog in the program page, which is shown in Fig.4-4.

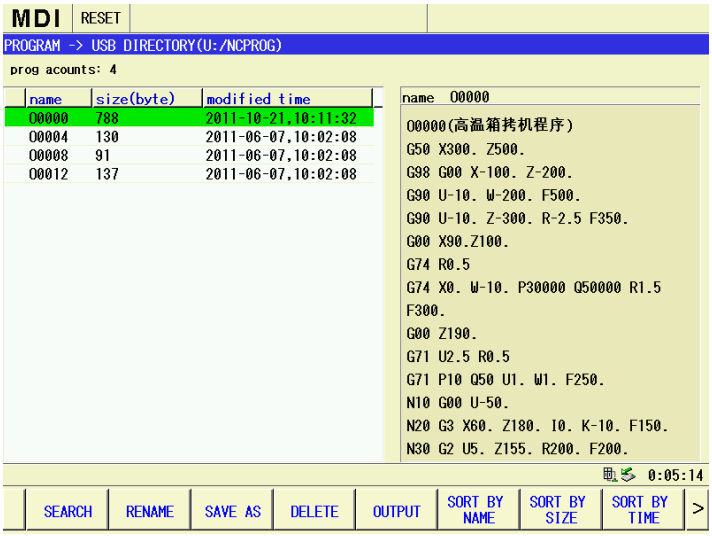


Fig.4-4

The programs of the USB flash disc directory can be copied into the local one by pressing OUTPUT software key, vise versa. The files in the USB flash disc are copied into the system, the detailed steps are as below:

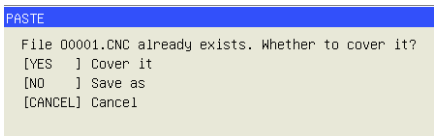
（1）Press USB software key to enter the USB flash disc file directory;

（2）Press , cursor key to select the programs to be copied, and press > and OUTPUT, the selected programs in the USB flash disc are copied into the local directory;

（3）Similarly, the programs of the local directory are copied into the USB flash disc, LOCAL software key is directly pressed to enter the system program file directory.

（4） The program to be copied is selected by pressing ⇧, ⇩ cursor keys, and the software key OUTPUT is pressed, the selected program in the local directory is copied into the directory of the USB flash disc;

（5） When the copied program has been existed, the pop-up dialogue box



will remind: The program has been existed, whether cover it?

（6）Press “Yes” software key to cover the existed program, press “No” to remind the program is saved as the other name, press “cancel” to cancel the operation.

Note 1: Only the operation authority level is equal to or higher than level [4], the program can be copied and pasted.

Note 2: If the output file is too big, the copying time is too long and the progress bar will be displayed; the page can be switched and other operation on the pages can be executed during coping.

4.1.7 Arranging a Programs

On the program page set, firstly press > software key, SORT BY NAME, SORT BY SIZE and SORT BY TIME software keys occur, and the programs are displayed in the required sequence.

SORT BY NAME, SORT BY SIZE, SORT BY TIME software keys are repeatedly pressed, the program arrangement sequence will be switched between ascending and descending order of each arranging sequence.

4.2 Rewriting a Program

4.2.1 Editing a Program

Based on the steps introduced in Section *4.1.2*, a new program is created, which is shown as Fig. 4-5:

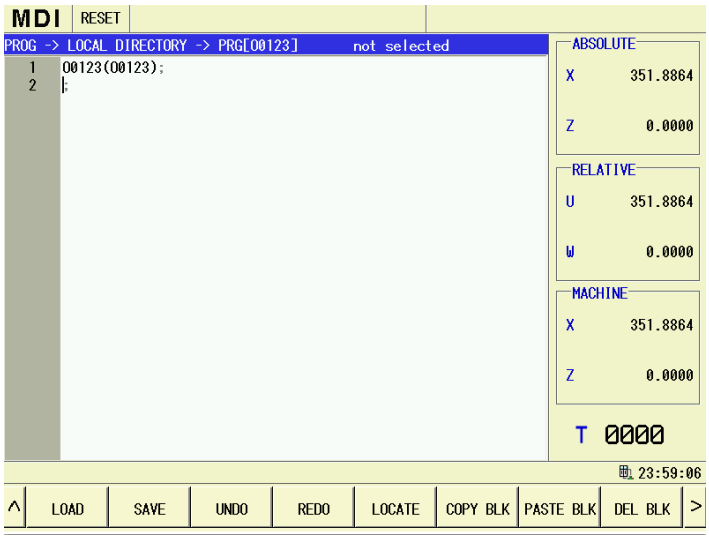


Fig.4-5

Introduction of the software keys on edit interface

LOAD: After the program editing is completed, the program can be executed after pressing the key, and the page is skipped into the position page set and the just loaded program is displayed in the program column of the position page set. Press AUTO key to switch into Auto mode and press CYCLE START key to execute the loaded program.

SAVE: Save the currently editing program.

UNDO: The operation of one step before editing the program can be canceled by pressing the key (the operation of the maximum latest 10 steps can be cancelled.).

REDO: The just cancelled program can be restored by pressing the key.

LOCATE: The specified program line can be exactly and rapidly positioned by pressing the key.

COPY BLK: The block on which the cursor is can be copied.

PASTE BLK: The just copied block can be pasted on the position on which the cursor is by pressing the key.

DEL BLK: The block on which the cursor is can be deleted by pressing the key.

On the current page, press >, the three software keys

SEARCH, PRG CHECK, CALCULATOR occur.

SEARCH: The character string can be rapidly found by pressing the key and the cursor is positioned behind the character string which has been searched. During searching, the three searching modes of FROM TOP, NEXT and PREV can be selected.

PRG CHECK: After the program is edited, press PRG CHECK software key to check whether the program has the syntax error; if it has, the reminder will occur below the screen, and the program should be checked and rewritten based on the reminder.

Note:

1: When the parameter 3404.6 is 0, the program must be with the end code, like M02, M30 or M99, etc; Otherwise, the system reminds the error during checking the program after pressing PRG CHECK software key, and the alarm occurs when the program is started.

2: The big file exceeding 10,000 lines can't be edited.

3: Besides manually saving programs, the CNC system owns an automatic save function during the course of edit, and it automatically saves files every 90 seconds.

CALCULATOR: the CNC system owns an automatic save function. To count some points during programming, press the soft key CALCULATOR, and the system pops up a counter which is shown in Fig.4-6.

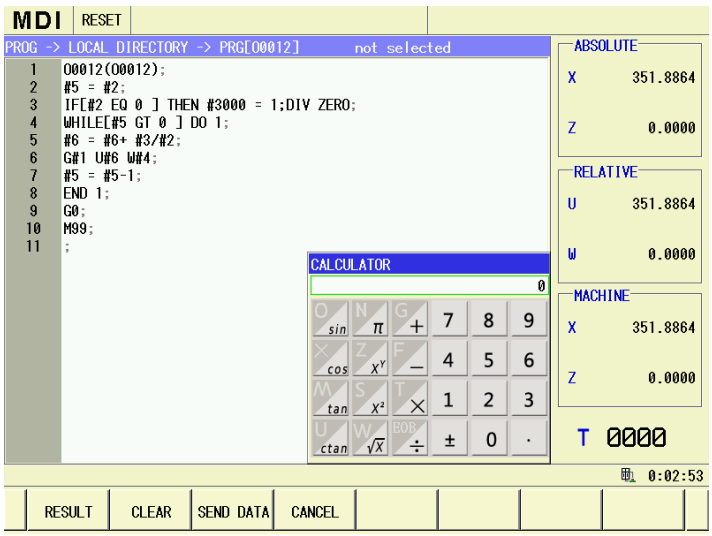


Fig.4-6

CALCULATOR: The system is with the calculator. If calculation is required during programming, CALCULATOR software key is pressed, the calculator will be popped up.

RESULT: equal to "=" in the counter.

CLEAR: clear the counter's data.

SEND DATA: send the current counter's value to the page where the cursor is.

CANCEL: exit the count function.

4.2.2 Rewriting a Program

（1）The program is opened based on Section 4.1.4;

（2）The cursor is moved toward the program line to be rewritten by pressing , , , keys; the cursor is moved toward the character to be rewritten by pressing , keys; the block or the character to be rewritten can be rapidly found by pressing LOCATE and SEARCH;

（3）The program code is rewritten based on the address and the numerical keys on the edit keypad;

（4）One character before the position, on which the cursor is, is deleted by pressing BACK SPACE key;

（5）One character after the position, on which the cursor is, is deleted by pressing BACK SPACE key;

（6）Press SAVE software key to save the currently rewritten program.

4.2.3 Shortcut Keys

During the program editing, the system provides some shortcut keys to edit and rewrite the program.

Adjusting the cursor

Press CHANGE and meanwhile to move the cursor to the file ahead;

Press CHANGE and meanwhile to move the cursor to the file end;

Press CHANGE and ⇦ meanwhile to move the cursor to the line ahead;

Press CHANGE and ⇦ meanwhile to move the cursor to the line end;

Selecting the arbitrary block

Press SHIFT + ⇧, ⇩, ⇦, ⇨ to move the cursor to the code to be copied, and then, the program in the middle is selected and displayed in the reverse color;

Deleting a block

After selecting the arbitrary block, press DELETE to delete;

Copying a arbitrary block

Press CHANGE and P O meanwhile to select the block to be copied;

Cutting a arbitrary block

Press CHANGE and C X meanwhile to select the block to be cut;

Pasting a arbitrary block

Press CHANGE and Q N meanwhile to paste the copied or cut block.

4.3 Block Notes

When the block should be explained, press “EOB” and add “；”after the block to be explained, the content after “；” is the note.

Example:

O0001；

G50 X0 Z0； The coordinate zero is set;

G00 X100 Z100；Rapidly move to the position of X100，Z100;

M30；

In the above program, the note is added in the 2nd and the 3rd blocks. The content after the 1st semicolon is the note, and the 2nd semicolon is the block end code and the 2nd semicolon is automatically added by the system after pressing INPUT for line feed when one block editing is completed.

Note: Because the system doesn’t support the input in Chinese; if the Chinese note should be input, it should be edit on the computer.

4.4 Generating a Block Number

In the program, the block number can be input or not, the program is executed based on the inputting sequence (except for calling).

On the setting page set, CNC sets the page, when “automatic sequence number” switch is OFF, CNC doesn’t automatically generate the block number, while the block number can be input manually during programming.

On the setting page set, when “automatic sequence number” switch is ON, CNC automatically generates the block number, and the block number of the next block is automatically generated by pressing INPUT key for line feed, and the incremental value of the block number is set by CNC data parameter #3216.

4.5 Program Backstage Editing

When Auto or DNC mode is being operated, the key PROGRAM is pressed to enter the program page, and the program to be edited can be opened or created, the operation method is same as that introduced previously.

Note 1: The currently running program can’t be edit;

Note 2: RESET key can’t be pressed when Auto or DNC mode is being operated and the program is being edited at the backstage; otherwise, the running program will stop due to resetting.

4.6 Program Run

In Auto or MDI mode, press CYCLE START key after the program is open, and the program automatically runs.

Chapter 5 Manual Operation

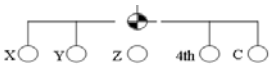
5.1 Manual Reference Position Return

A CNC machine tool is provided with a fixed position for setting the position of the machine worktable. And the fixed position is called as the reference position. Normally, tool change or setting the coordinate system is performed at this position. After power on, the tool is moved into the reference position. Manual reference position return is to move the tool into the reference position by the switches and the buttons on the operation panel.

The three methods of setting GSK988TA reference position: zero return with the block, zero return without the block and zero return of the absolute encoder.

Setting the reference position with the block:

When the parameter DLZx (the 1st bit of #1005) is 0, setting the reference position without the block is invalid (Setting the reference position with the block is valid), and the machine tool can return the reference position only after installing the deceleration switch.

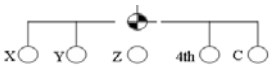
Detailed process: The tool is moved in the direction specified by the parameter ZMI (the 5th bit of #1006), the tool is moved to the deceleration point at the rapid traverse rate and returned to the reference position at FL speed. The reference position return finish lamp (LED) 

is ON, and the reference position return ends and the coordinate system is automatically set.

Note: Each axis rapid traverse rate, the rapid movement override F0 speed and FL speed of reference position return is respectively set by the parameters (#1420, #1421 and #1425).

Setting the reference position without the block:

When the parameter DLZx (the 1st bit of #1005) is 1, setting the reference position without the block is valid. The machine tool can return the reference position without installing the deceleration switch.

Detailed process: After the machine tool is powered on each time, the reference position return is executed, the tool is moved in the direction specified by the parameter ZMI (the 5th bit of #1006). After the system detects the 1st PC signal of the motor, the reference position return finish lamp (LED) 

is ON, and the reference position return ends and the coordinate system is automatically set.

Note: Because setting the reference position without the block is to detect the 1st PC signal of the current position and it is taken as the reference position, so the set reference positions after power on may be different, the tool offset must be reset with the method.

Setting the reference position of the absolute encoder

When the machine tool is configurated with the absolute position encoder and the absolute position encoder reference position function is valid, the reference position return of the absolute position encoder should be executed when the system hasn't set the reference position. After the tool is returned to the reference position, the reference position return finish lamp (LED) is ON, and the coordinate system is automatically set.

Steps of reference position return:

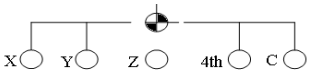
（1）Press the reference position return switch 

, and it is one of the mode selection switch;

（2）Press one of the rapid movement override switches 

for deceleration;

（3）After pressing the feeding axis corresponding the reference position return and the direction selection switch 

, the reference position return is started. The tool is moved to the deceleration point at the rapid traverse rate (The absolute position encoder zero return doesn't have the deceleration point and it can be directly returned to the reference position), and the tool is returned to the reference position at FL speed set by the parameter. After the tool is returned to the reference position, the reference position return finish lamp (LED) 

is ON;

（4）The same operation is executed for other axes.

Note 1: The manual reference position return can only be returned to the 1st reference position, the coordinate system is automatically set after the manual reference position return ends.

Note 2: Once the reference position return is completed, "the reference position return finish" indicator lamp is ON, the machine tool doesn't move unit the reference position return switch is OFF.

Note 3: When "leaving off the reference position" is executed, the reference position return finish indicator lamp is OFF.

Note 4: The direction of each axis reference position return is set by the 5th bit of the parameter #1006.

Note 5: The 2nd bit of the parameter 1404: After setting the reference position, the reference position return is executed in Manual mode, the tool is moved to the reference position at the rapid feedrate or at the manual rapid feedrate.

Note 6: After setting the system reference position with the absolute encoder, the system automatically sets the coordinate system after power on, again without setting the reference position. About the non-absolute encoder system, the system should execute the reference position return after power on, again.

The above mentioned is one example, please refer to the user manual from the machine tool builder for operation.

5.2 Manual Feeding

In manual mode, press the feeding axis and the direction selection switches on the machine operation panel and the machine tool is moved along the selected direction of the selected axis.

Each axis manual continuous feedrate is set by the parameter (#1423), and each axis manual continuous feedrate is adjusted by the manual continuous feedrate override dial.

Feedrate override dial

After the rapid movement switch 

is pressed, the machine tool is moved at the rapid traverse rate (#1424 parameter) no matter where the position of the manual feedrate override dial is, and the function is called as the manual rapid movement. Many axes can be moved meanwhile in Manual mode.

Manual feeding steps:

（1）Press the manual continuous switch 

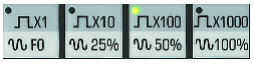
, and it is one of the mode selection switches;

（2）Press the feeding axis and the direction selection switch 

, the machine tool is moved along the corresponding direction of the corresponding axis. After the switch is pressed, the machine tool is moved at the feedrate set by the parameter (#1423); after the machine tool is released, the machine tool feeding stops;

（3）The manual continuous feedrate is adjusted by the manual continuous feedrate override dial;

（4）If the rapid traverse switch 

is pressed when the feeding axis and the direction selection switches are pressed, the machine tool is moved at the rapid traverse rate when the rapid traverse switch is pressed, and the rapid movement override 

is valid during rapid traverse;

The above mentioned is one example, please refer to the user manual from the machine tool builder for operation.

Note 1: Acceleration/deceleration: the manual rapid traverse rate, the time constant of acceleration/deceleration and the acceleration/ deceleration mode can be set by the parameters 1610 and 1624.

Note 2: Changing the mode: during manual feeding, the manual feeding is invalid when the mode is switched into the other mode. The manual feeding mode is entered firstly, and then the feeding axis and the mode selection switches are pressed, and then, the manual feeding is valid.

Note 3: Rapid movement before reference position return: if the reference position return isn't executed after power on, the button of "rapid traverse" is pressed, the rapid movement can't be executed while the manual continuous movement is maintained. The function is set by the parameter RPD (bit 0 of #1401).

Note 4: Whether the manual override is valid in Manual mode is set by bit 2 of parameter 1402, and the override is fixed as 100% when the manual override is invalid.

5.3 Incremental Feeding

Whether the incremental feeding is valid or not in Manual mode or MPG feeding mode is set by the parameter JHD (0 bit of #7100),and the corresponding relation is shown as the following list:

	JHD=0		JHD=1	
	Manual mode	MPG mode	Manual mode	MPG mode
Manual feeding	O	×	O	×
MPG feeding	×	O	O	O
Incremental feeding	×	×	×	O

O: Valid
×: Invalid

In incremental mode, press the feeding axis and the direction selection switch 

on the machine tool operation panel and the machine tool is moved one step in the selected axial direction. The minimum distance moved by the machine tool is the minimum input increment. Each step can be 1 time, 10 times or 100 times of the minimum input increment.
The incremental feeding steps:

（1）Press 

to select MPG mode;

（2）The movement distance of each step is selected by the override switch 

; Moreover, the single step movement distance selected by the override switch 

can be rewritten by the parameters 7113 and 7114;

（3）After the feeding axis and the direction selection switch 

is pressed, the machine tool is moved along the selected axial direction. The switch is pressed once, the machine tool is moved one step and the feedrate is the manual rapid traverse rate.

（4）The rapid movement switch is invalid during pressing the feeding axis and the direction selection switches.

Note: The least input unit (input) and the least code increment (output) is set by the 1st bit of the parameter #1004.The least input increment is the minimum unit of the program movement distance, and the least code increment is the minimum unit of the tool movement on the machine tool, and two increments are represented by mm or inch.

5.4 MPG Feeding

Press 

key to enter MPG mode, the MPG outline is shown as Fig.5-1:：

Fig.5-1

In MPG mode, the machine tool can be continuously moved by MPG on the machine tool operation panel. The moving axis is selected by the switches 

. When MPG is rotated for one scale, the minimum distance moved by the tool equals to the least input increment. When MPG is rotated for one scale, the tool movement distance can be scaled up for 10 times or for two magnifications set by the parameters (#7113 and #7114).

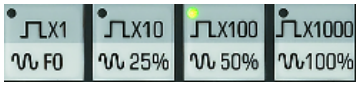
Steps of MPG feeding:

（1）Press 

key to enter MPG mode;

（2）Press MPG feeding axis selection switch 

to select the axis to be moved by the machine tool;

（3）Press MPG feedrate override switch 

to select the override of the machine tool movement. When MPG is rotated for one scale, the minimum distance moved by the machine tool equals to the product of the least input increment timing the current override. The override selected by the override switches 

can be rewritten by the parameters 7113 and 7114;

（4）After MPG is rotated, the machine tool is moved along the selected axis and MPG is rotated for 360 degrees, and the machine tool movement distance equals to that of the current pulse equivalent *100.

The MPG feeding direction is set by MPG rotation direction. Normally, MPG CW rotation is positive feeding, CCW rotation is negative feeding.

The above mentioned is one example, please refer to the user manual from the machine tool builder for operation.

Note 1: In manual mode (JHD), the validity of MPG;

Whether MPG is valid in Manual mode is set by the parameter JHD (bit 0 of #7100), MPG feeding and incremental feeding all are valid when parameter JHD (bit 0 of #7100) is set as 1.

The corresponding relation is shown as the following list:

	JHD=0		JHD=1	
	Manual mode	MPG mode	Manual mode	MPG mode
Manual mode	O	×	O	×
MPG mode	×	O	O	O
Incremental feeding	×	×	×	O

O: Valid
×: Invalid

Note 2: The manual pulse generator（MPG）executed speed exceeds the rapid traverse rate (HPT):
The specification of parameter HPT (the 4th bit of #7100):
Setting as 0: When the feedrate is limited in the rapid traverse rate, the pulse exceeding the rapid traverse rate is invalid (The machine tool movement distance may be not consistent with the scale of MPG).
Setting as 1: The feedrate is limited in the rapid traverse rate, and the pulse exceeding the rapid traverse rate is valid and accumulated in CNC.(Although the MPG is not rotated, the machine tool can't stop immediately. After MPG stops, due to the effect of the pulse in CNC, the machine tool still moves.) The allowable value of the memory is set by the parameter #7117, then, the exceeding pulse is also ignored.

Note 3: The axis movement direction and the rotary direction of MPG:
When the tool is moved along the axis, MPG direction is switched by the parameter HNGx (0 bit of #7102) and it is corresponding to the rotation direction of MPG.

Note 4: Quantity of MPG:
Maximum two manual pulse generators can be connected, which is set by the parameter #7110. Two manual pulse generators can operate one selected axis meanwhile.

5.5 MPG Retreating

The MPG retreating function is to use the MPG (manual pulse generator) in automatic run to make the program positively/reversely move. The actual machine operation and executing the MPG can simply check the program errors.

Check mode

In the mode, execute the program along the clockwise or anticlockwise to check programs. To set to the check mode, the CNC should be in the automatic run mode and the check signal MMOD<G67.2> is set to 1. When the CNC is set to the check mode and the program positively move. It is set to a check mode, the system creates a reverse moving data when the program positively moves. To execute the motion which is synchronous with the MPG's pulse in the check mode, besides executing the above settings, set the MPG check signal MCHK<G67.3> to 1. Thus, execute the MPG's program check.

Positive movement

The "positive movement" is to make the program clockwise execute by the MPG positively rotating(MPG check signal MCHK<G67.3> is "1"). The program' execution speed and MPG's speed are proportional. Once the MPG rapidly rotates in positive direction, the speed quickens. When the MPG slowly rotates in positive direction, the speed becomes slowly.

Reverse movement

The "Reverse movement" is to make the positive movement's blocks being reversely executed by the MPG reversely rotation (anticlockwise rotation). The reverse program's execution speed and the MPG' speed are proportional.

5.5.1 MPG Retreat Operation Method

In automatic run mode, set the check mode signal MMOD<G67.2> to "1" and set the CNC to the check mode, start executing the program by the cycle start.

This moment, when the MPG check signal MCHK<G67.3> is set to "1", the MPG's pulse controls the program execution in the check mode, and a run which is synchronous with the pulse is executed.

When the check mode signal MMOD<G67.2> in program run becomes "1", the block in the next buffer starts to become the check mode, namely, even if the check mode signal is set to "1", the system does not immediately become the check mode.

When the CNC becomes the check mode, the check mode confirmation signal MMMOD<F91.3> becomes "1".

5.5.2 Speed Control based on the MPG

The MPG's machine movement speed is determined by No.6410 and the MPG override. The machine movement speed is counted according to the following formular during the actual rotating the MPG:

[Code speed]×[1 second' MPG quantity]×[MPG override]×([the parameter's setting value]/100) ×(8/1000)(mm/min or inch/min)

Example ：

The code speed is 30mm/min，MPG override is 100， No.6410=1: 1 rotation（100 pulses）/second's speed rotating the MPG, the movement speed is counted by the following formular:

[movement speed]=30×100×100×（1/100）×（8/1000） = 24[mm/min]

Limitation：when the override exceeds 100% because of rapidly rotating the MPG, it is clamped at 100%, namely:

[1 second's MPG quantity]×[MPG override]×([the parameter's setting value]/100) ×(8/1000)

When the MPG's pulse exceeds 1, the speed is clamped.

The rapid traverse speed is clamped at 10% speed. When RPO(No.6400.0) is set to "1"，the override 100% is clamped. Besides, when No.6405 is set to any values, any override value can be clamped. When No.6405 setting value is more than 100, it is clamped at 100% value. When No.6405 is set to 0，RPO（No.6400.0）setting value is valid.

When the MPG control is executed, the single block stop signal and feed hold signal are invalid. When the single block stops or feed hold completely stops and the cycle start signal is not valid, the program remains stop state.

Blocks for movement and pause can control the program's execution speed by the MPG rotation. For the block only with M, S, T, F code, without movement and pause blocks, the control transfers to the next block even fi the MPG does not rotate.

The spindle speed is different from the MPG's pulse, even if in the check mode, it rotates by the executed speed. The feed per rotation is executed when the spindle speed in the moment is changed to the value equivalent to the feed per minute.

Note: the MPG's retreacting function using the 1st MPG but the 2nd MPG cannot be used.

5.5.3 Rules for Each Code's Reverse Movement

Its modal message is saved when the G code positively moves and it is used when it reversely moves.

G code

When the G making the modal message change reversely moves, its modal message before change is executed.

5.5.4 Notes

1. When M2 or M30 is executed, the MPG retreat ends. The CNC cannot execute the reverse movement from M2 or M30 block，the check mode signal and MPG check signal are set to "0" after the program ends;
2. In the check mode, the CNC cannot use the dry run, and the dry run signal must be set to "0";
3. Cannot switch the workpiece mode during the program execution;
4. When M198 calls a subprogram, although it can execute the positive movement, but must not permit the reverse movement operation;
5. In the blocks for executing the thread cutting, the MPG pulse is invalid, and the thread cutting is executed at the override 100%, and the reverse movement is forbidden;
6. When the MPG retreat reversely moves, the multi-spindle function may create a mistakne operation;
7. The axis based on the PLC axis cannot be controlled by the function;
8. When a macro program in the MPG retreat is executed, a mistaken movement may occur;
9. The system cannot support the cuntion in DNC mode.

Chapter 6 Auto Operation

6.1 Auto Operation

The program is preset in the memory, when one program is selected and the cycle start button 

on the machine tool operation panel is pressed, and the program is started to run and the cycle start indicator is ON. During cycle start and when the feed hold button 

on the machine tool operation panel is pressed during the cycle start period, and the automatic running pauses and stops. When the cycle start button is pressed, again, the automatic running is restored. When 

key is pressed on MDI panel, automatic running ends and the system enters the resetting status.

6.1.1 Select the Program to Run

（1）Press 

function key to enter the program page set.

（2）On the program page set, press  ,  ,  and  keys to move the cursor to select the program name. Or press 

key to search the program name to run, the base color of the selected program is green, which is shown as Fig. 6-1:

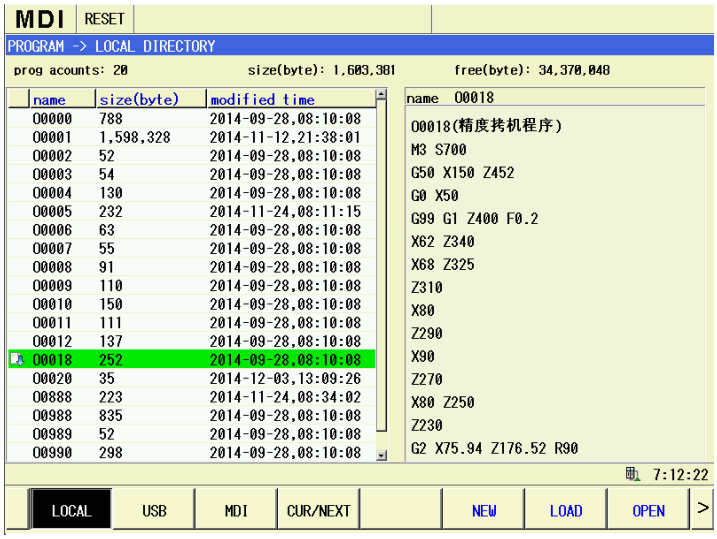


Fig.6-1

(3)During resetting, press 

software key, and the selected program is loaded into the block area in the position page set, and it can be executed currently, and the page is skipped into the position display page, which is shown as Fig. 6-2:

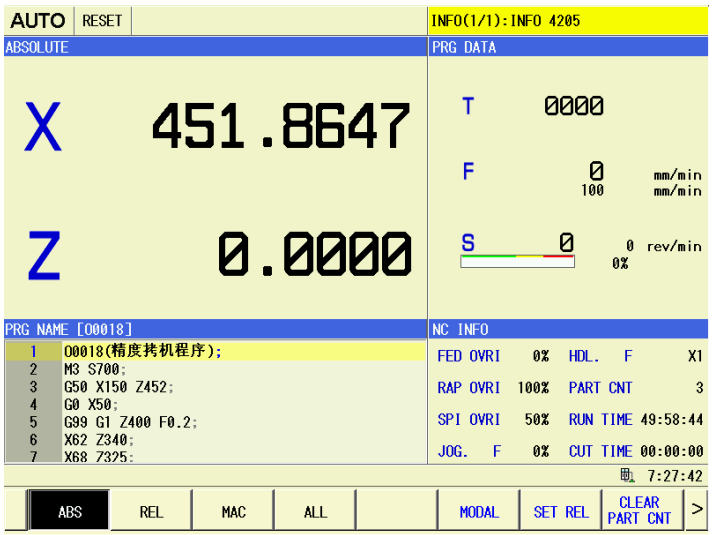


Fig. 6-2

Note: The file can be loaded during the resetting status.

6.1.2 Program Running

（1）Press  key to select Auto mode;

（2）Press  to start the program and the program is automatically started and the cycle start indicator is ON. When automatic running ends, the cycle start indicator is OFF. When the last block is specified with M99 and after running ends, the program is repeatedly executed from the program ahead.

（3）To stop during running or cancel the memory running, there are several methods:

1）Memory running stops

Press the feed hold button  on the machine tool operation panel. The feed hold indicator lamp is ON while the cycle start indicator lamp is OFF, and the machine tool responds as below:

a. When the machine tool is being moved, the feeding decelerates and stops.

b. When pause (the tool stops) is executed, running stops.

c. When M, S or T function is executed, running stops after M, S or T function is completed.

When feed hold indicator is ON, press the cycle start button  on the machine tool operation panel, the machine running is restarted.

2）The memory running end

Press  key on MDI panel, automatic running ends and the system enters the resetting status.

Note: The program running is started from the line on which the cursor is, so firstly check whether the cursor

is on the block to run before pressing  **key.**

6.1.3 Running from the Arbitrary Block

（1）Press AUTO key to enter Auto mode, press PROGRAM software key to enter the program interface and press or key to select the program content page; press or to move the cursor to the block to run; On the program page set, press or to select the program to run, press OPEN software key to enter the program edit interface and then press or key to move the cursor to the block to start and then press LOAD software key to return the position display page.

（2）If the mode（G, M, T or F command）of the block on which the cursor is, is defaulted, and the mode on which the block is running isn't consistent, the corresponding mode must be executed, and then the following step can be executed;

（3）Press AUTO key to enter Auto mode and press CYCLE START to start the program and the program is started form the selected block.

6.1.4 Block Skip

When the symbol "/"is added at the block ahead, press SKIP switch to start the block skip mode, and the block skip indicator is ON, and press CYCLE START to run the program, the block is skipped while it's not operated, which is shown as the 4th line of the program:

O0001；

G50 X0 Z0；Set the coordinate zero;

G01 X100 Z100；Rapidly move to the position of X100，Z100;

/G0 X0 Z0；

M30；

Press CYCLE START to run the program and the 4th line is skipped.

6.1.5 G31 Skip

G31 code is edit before the block; during the code is being executed, if the external skip signal (X0.4) is input, the code execution is interrupted and the system transfers to the next block. The function is for dynamically measuring the workpiece dimension (like the grinding machine), the tool setting for measuring, etc, which is shown as the 2nd line of the following program:

O0002;
G31 Z200 F100; During executing the block, if the external skip signal(X0.5) is input, the block is interrupted and the next block is executed.
G01 X100 Z300;
……;
M30;

Note: About the details of G31 skip code, please refer to *Programming Manual*.

6.1.6 Automatic Running Stop

There are several methods to stop the memory running: Command stopping or press the relative keys on the machine tool operation panel to stop.

- **Command stopping (M00, M01, M02, M30)**

After executing the block including M00, M01 (the machine tool panel optional stop button is valid) , automatic running stops, the modal function and status are all saved. The program running is continued after pressing 

key.

After reading M02 or M30 (command at the main program end), the program running ends and the system enters the resetting status.

For the different machine tools, the operations are different. About the details, please refer to the user manual from the machine tool manufacturer.

- **Pressing the relative keys to stop**

1. During automatic running, after pressing 

key, the machine tool is displayed in the following status:

（1）The machine tool feeding is decelerated and stopped;

（2）The modal function and the status are saved.

（3）After pressing 

key, the program is continued to run.

2. Pressing the reset key 

（1）All axes movements are decelerated and stopped;

（2）M and S function output is invalid (After pressing 

key, whether automatically switch off the signals like the spindle CW/CCW, lubrication, cooling, etc is set by the parameter);

（3）After automatic running ends, the modal function remains.

3. Press the emergency stop button

During the machine tool running, press the emergency stop button in danger or in the emergency situation, (the external emergency stop signal is valid), CNC enters the emergency stop process, the machine tool movement stops immediately, all output (such as the spindle revolution, the coolant) is

switched off. After the emergency stop button is released, the emergency stop alarm is released, CNC enters the resetting status.

4. Switching the operation mode

During automatic running, the system switches into the reference position return, MPG/single step and the manual mode, the current block “dwells” immediately; during automatic running, when the system is switched into Edit or MDI mode, the currently running block stops after running is completed.

Note 1: The emergency stop alarm is released after the trouble is shot;

Note 2: Before power on or off, press the emergency stop button to reduce the electric shock upon the equipment;

Note 3: After the emergency stop alarm is released, the reference position return is executed, again to guarantee the correctness of the coordinate position.

6.2 Manual Data Input (MDI) Running

6.2.1 Editing the Program in MDI mode

（1）On the program display page set, press 

key to enter MDI mode, and the page is shown as Fig. 6-3:

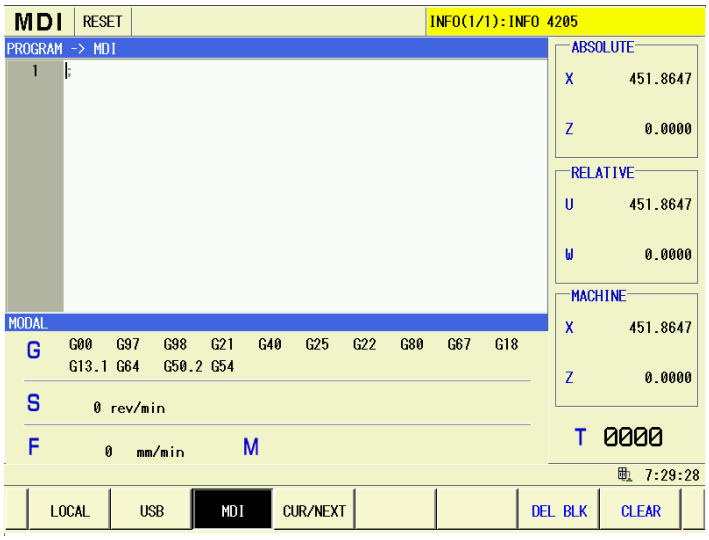


Fig.6-3

（2）On the program edit column below the block (MDI), input the block (maximum 10 lines) to run, the method of editing the program to be executed is similar with the common one. The program set in MDI mode is valid for rewriting and deleting the word. About editing the program, refer to the 5th chapter.

（3）After inputting the block, the cursor is moved to the program ahead and the program is started to be executed (if the cursor is on some block, the system is executed from the block). After pressing 

key, MDI command word is started to be executed from the line on which the

cursor is. After the program running ends, M02 program end code is executed and the cursor doesn't return the program ahead; while M30 program end code is executed, the cursor is returned to the program ahead, after the program running ends, the system enters the stopping status.

（4）During running, press FEED HOLD key, RESET key and the emergency stop button to make MDI command word stop running.

Note 1: Deleting the program

a. In MDI mode, press DEL BLK software key to delete the block on which the cursor is and press CLEAR software key to delete all blocks in MDI edit column.

b. When parameter MCL（NO.3203#7）is set as 1, the program is automatically cleared after pressing RESET key.

c. When parameter MER（NO.3203#6）is set as 1, MDI program running is completed and deleted in MDI mode.

Note 2: When MDI running stops, after edit operation ends, CYCLE START is pressed for running, again, the running is started from the position on which the current cursor is.

Note 3: The program created in MDI mode can't be stored.

Note 4: The sub-program calling and macro program calling functions can not be performed in the MDI mode.

6.2.2 Running from Arbitrary Block

On position display page, in MDI mode, ⇧ or ⇩ key is pressed to move the cursor to the block to be started running, and then CYCLE START is pressed to start the program, the program is executed from the block on which the cursor is.

6.2.3 Stopping MDI Operation

Stopping MDI running is similar with stopping automatic running, please refer to the operation method in Chapter 6.1.4.

6.3 DNC Running

GSK988TA/988TA1/988TB is with DNC function, and DNC communication software can be connected with CNC to realize the running at high speed and with the big capacity.

After the machine tool panel key DNC is pressed to enter DNC mode, and PC port is ready, and the machine tool panel cycle start key is pressed to start the program DNC machining.

About the detailed operation method, refer to DNC communication software.

（1） The machining program is selected and opened with the communication software GSKComm, which is shown as Fig.6-4:

Fig.6-4

（2） Connect CNC system, which is shown as Fig.6-5.

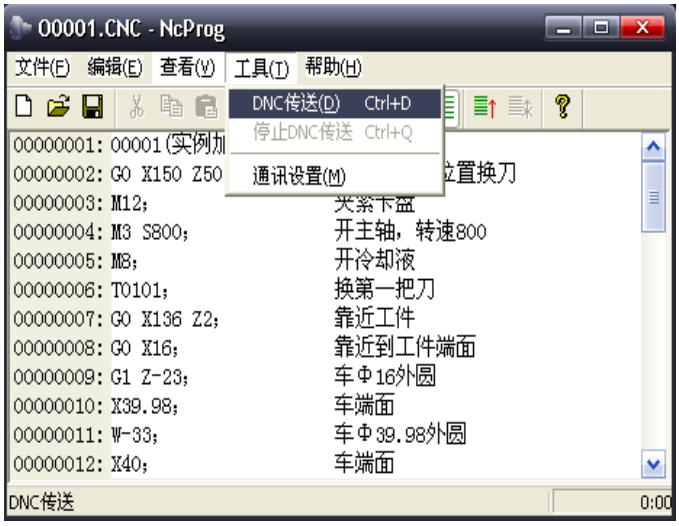


Fig.6-5

（2） Press 

key to select DNC operation mode, which is shown as Fig.6-6.

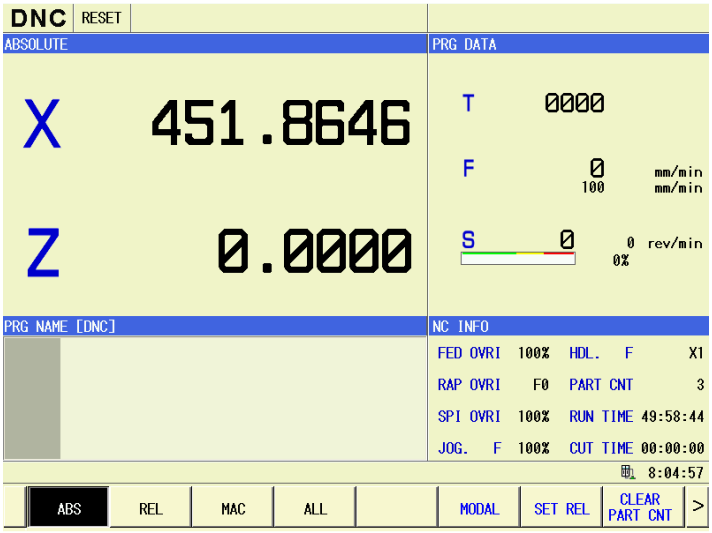


Fig.6-6

（3） The key  is pressed to start the program, the program is automatically started and the cycle start indicator is ON. When the automatic running ends, the cycle start indicator is OFF, which is shown as Fig. 6-7:

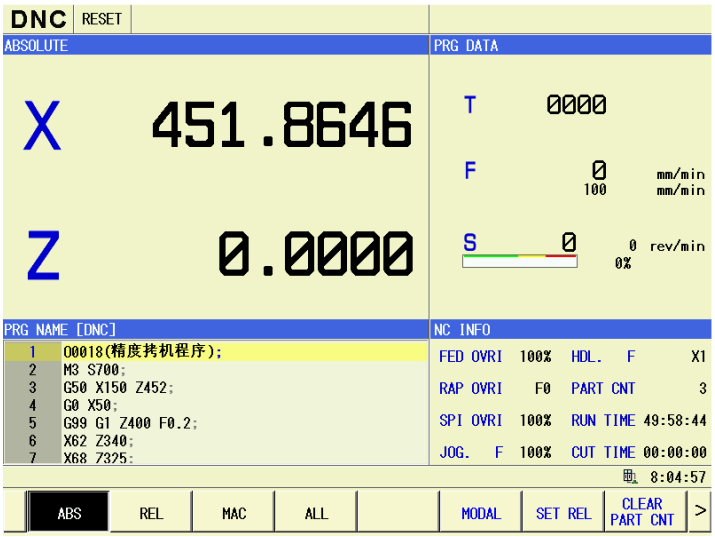


Fig.6-7

（5）Stopping during running

Press the feed hold button  on the machine tool operation panel. The feed hold indicator lamp is ON while the cycle start indicator lamp is OFF. The machine tool is operated as below:

a. When the machine tool is being moved, the feeding is decelerated and stopped.

b. When the pause (stopping the tool) is being executed, running stops.

c. When M, S or T function is being executed, running stops after M, S or T function is completed.

When the feed hold indicator lamp is ON, the cycle start button  on the machine tool operation panel is pressed and the machine tool running is restarted.

（6）Running stop

The key 

on MDI panel is pressed or M30 command is executed by DNC program, running ends and the system enters the resetting status, which is shown as Fig.6-8:

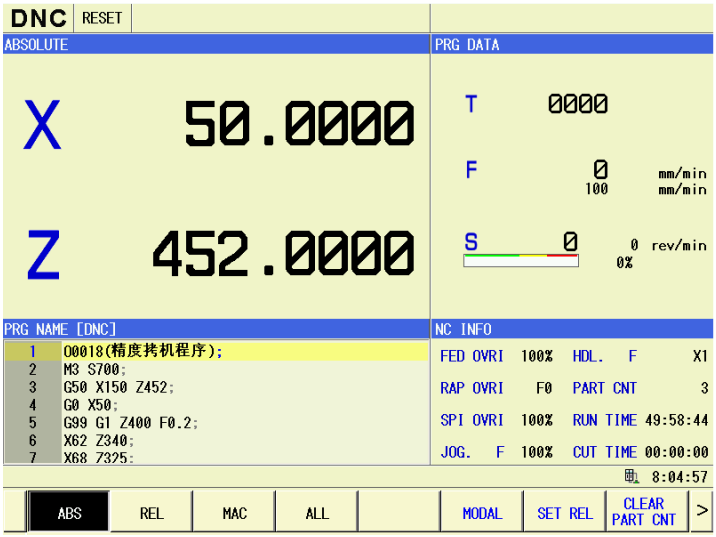


Fig.6-8

Note: In DNC program, the program calling and the program skip commands can't be executed.

6.4 Automatic Running Status Control

6.4.1 Machine Lock and the Miscellaneous Lock

When the tool isn't moved, while the coordinate position changing is displayed, the machine lock function is used. After using the function, all axes are locked and all axes movements are stopped. Except for that, M, S and T commands are locked by the miscellaneous function. And it is same as the machine tool locked to check the program.

6.4.1.1 The Machine Lock

When the machining program is executed, while the machine remains still, only the tool position change is displayed, the machine can be locked to check the program. When the machine is locked, all axes movements are stopped.

Press the machine locked switch 

on the operation panel. The machine isn't moved while each axis position on the monitor is being changed. About the machine locked, please refer to the user manual from the machine tool builder.

Note 1: During automatic running, the position relation between the workpiece coordinate system and the machine coordinate system may be different before and after the machine locked. Then, the command can be set by the coordinate or the workpiece coordinate system can be confirmed with the manual reference position return.

Note 2: When the machine is locked, G28 or G30 command is sent, although the command is received, the tool can't return the reference position and the reference position return indicator lamp is OFF.

6.4.1.2 M.S.T Lock

M, S or T command is locked by the miscellaneous function, and it is same as the machine locked to check the program.

Press the M.S.T function locked switch 

on the operation panel. M, S and T codes are invalid in the program and can't be executed. About the miscellaneous function locked, please refer to the user manual from the machine tool builder.

Note:

1）When the machine is locked, M, S and T commands are still executed;

2）Even M.S,T functions are locked, M00, M01, M02, M30, M98 and M99 (the subprogram calling function) commands are also be executed.

6.4.2 Dry Run

Press dry run switch 

on the operation panel, the machine tool is moved at the speed set by the parameter excluding the feedrate specified in the program, and the function is used to check the machine tool movement when the workpiece is removed from the worktable.

Dry run steps:

Before automatic running, press dry run switch on the machine operation panel, the machine tool is moved at the feedrate set by the parameter and the feedrate can be changed by the rapid movement switch. About the details of the dry run, please refer to the user manual from the machine tool builder.

Based on the rapid movement switch and the parameters, the dry run speed is changed as below:

Rapid traverse button	**Program command**	
	Rapid movement	Feeding
ON	Rapid traverse rate	Dry run speed *JVmax
OFF	Rapid traverse rate or the dry run speed *JV（①）	Dry run speed *JV

JVmax: The maximum scale value of the feedrate override

JV: The scale value of the feedrate override

Note 1: When the parameter RDR（NO.1401＃6）is "1", the running speed is the product of the dry run speed *JV; it is "0", the running speed is the rapid traverse rate.

Note 2: The maximum cutting feedrate is set by the parameter #1422;

Note 3: The rapid traverse rate is set by the parameter #1420;

Note 4: The dry run speed is set by the parameter #1410.

Note 5: During automatic running, the dry run can't be switched, while it can be switched during pause.

Note 6: During drilling and boring by G83 and G85, it can be switched into dry run mode after the single block stops or pauses, while the actual cutting speed remains as the original one. After cutting is completed, the speed can be switched. During executing the program, the dry run function can't be real-time switched ON or OFF, and the dry run function is switched only after the single block stops or pauses, it can be switched into dry run function. During drilling and cutting, even the system switches into the dry run mode, the speed remains as the original feedrate (The dry run

status is executed at the dry run speed before cutting; otherwise, it is at the commanded speed after cutting), only after cutting is completed, the speed can be changed.

6.4.3 Single Block Running

When the program is executed at the initial time, the single block running can be selected to avoid the unexpected situation due to the programming mistake.

In automatic mode, the method of turning on the single block switch is as below:

The single block switch 

is pressed to start the single block mode, and the single block running indicator lamp is ON. In the single block mode, when the cycle start button 

is pressed, the machine tool stops after one block of the program execution is completed; if the next block should be operated, the key 

should be pressed, again, and such operation is repeatedly executed until the program execution is completed. In the single block mode, the program can be checked by executing the program in one block by one block in single block mode.

Steps of single block running:

（1）Press the single block switch on the machine tool operation panel and press the cycle start button 

to execute one block of the program. The machine tool stops after the current block execution is completed;

（2）Press the cycle start button to execute the next block and the machine tool stops until the block execution is completed.

Note 1: The reference position return and single block running: The single block of the intermediate position is valid if G28, G29 or G30 command is sent.

Note 2: The subprogram block and single block running: The single block stops in the block with M98P_; M99; or G65.

Note 3: About the canned cycle and the multi-level cycle execution in the single block mode, refer to the relative content of the code manual.

6.4.4 Feedrate Override

The programmed feedrate can be reduced or increased by selecting the percentage (%) of the override scale. The characteristic is to check the program. For example, when the feedrate is specified as 100mm/min in the program, the override scale is set as 50% and the machine tool is moved at the speed of 50mm/min.

Steps of changing the feedrate override: Before automatic running or during running, the feedrate override scale on the machine operation panel is set as the expected percentage value (%).

Feedrate override knob

The override is in the specified range from 0 to 150%. For the special machine tool, the range is remarked in the manual from the machine tool builder.

Override during the thread cutting: During thread cutting, the override is invalid and the feedrate specified by the program is maintained.

6.4.5 Rapid Movement Override

The rapid movement speed has override of four levels (F0，25%，50% and 100%). Each axis rapid traverse rate is set by the parameter #1420; F0 is set by the parameter #1421.

The steps of changing the rapid traverse override: During the rapid traverse, select one override with the rapid traverse override buttons 

.

The rapid movements of the following types are all valid and the rapid traverse overrides all apply to them:

(1) G00 rapid traverse
(2) Rapid traverse in the canned cycle
(3) Manual rapid traverse
(4) Rapid traverse of manual reference position return
(5) Rapid traverse in G28 and G30

6.5 Program Restart

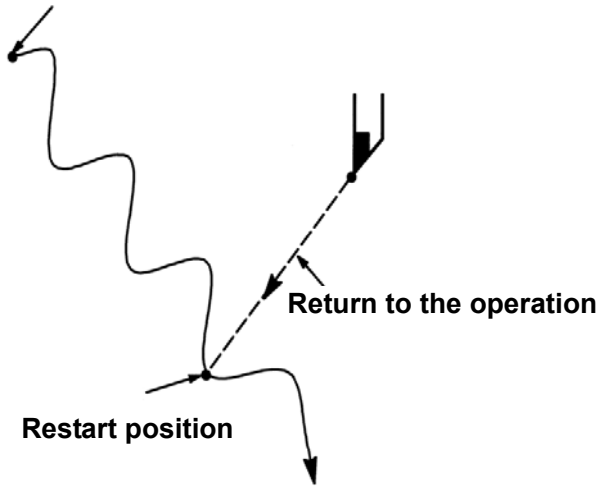
When the tool is damaged in the Auto operation, alternatively, after resting for several days, if the interrupted machine will be restarted from the resting position, the expected restart sequence number of the block, or the program section number (from the beginning to the desired restart block) can be specified; that is, the machining can be performed again from this block.

There are two methods for the restarting:

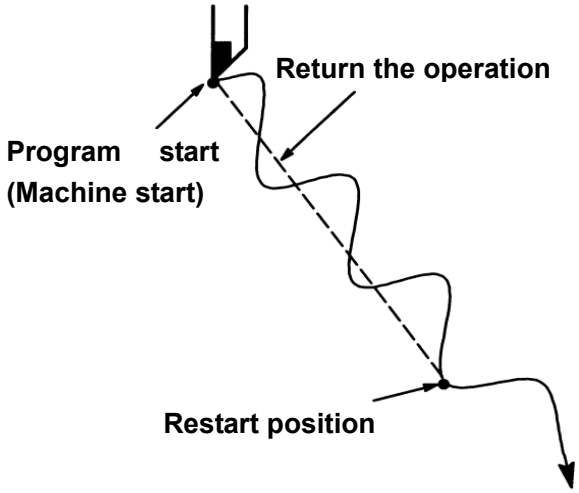
P: Restart executes when the tool is already damaged.

Q: Restart executes after the power is turned off (after resting for several days) or the ESP is released.

Type P	Program restarts from any position. Restart performs when the tool is damaged.



Type Q	It is necessary to move the tool to the program start (Machining start) before restarting the operation.



6.5.1 Steps of Program Restart

✧ **Step one**

Type P: 1. Retract the tool and replace it by a new one. Change the tool offset value in essential. (Enter to the step 2)

Type Q: 1. When the system is turned on again or the ESP is cleared, the necessary operations should be performed including the reference point return.

2. Move the tool to the program start manually (machining start); modal data and coordinate system are held at same state at the beginning of the machining.

3. Change the offset value if necessary. (Enter to the step 2)

✧ **Step 2 (Type P and Type Q are universal.)**

1. Press the PROG. RESTART button on the machine tool operation panel.

2. Find and then press the PROGRAM RESTART softkey under the program page, the Fig. 6-9 shows, and then enter the program restart page.

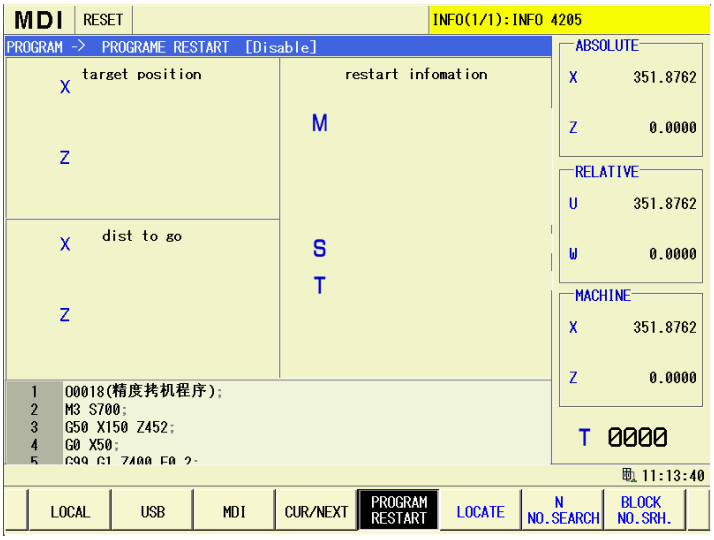


Fig. 6-9

1. Input the desired restarting program of the N number or the block no. by N NO.SEARCH or BLOCK NO.SRH., and then select the corresponding index mode by the software SEARCH TYPE P or SEARCH TYPE Q, refer to the Fig.6-10:

Note: The Macro statement, macro program call or subprogram call statement in the block number can also be calculated as the block.

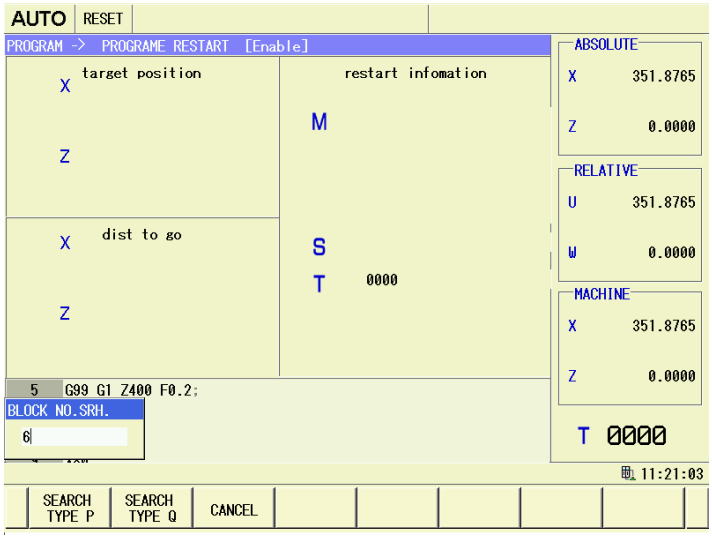


Fig.6-10

1. After indexing, the page becomes the program restart screen, refer to the Fig.6-11:

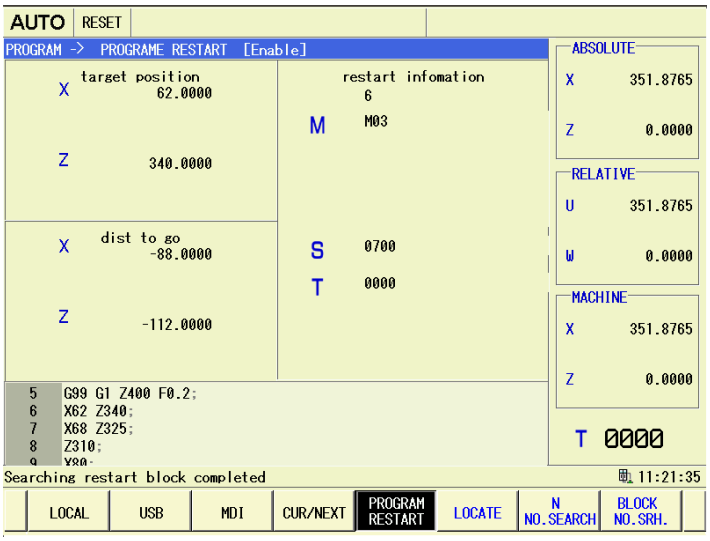


Fig.6-11

Objective position: The position of starting machining.

Residual movement value: Move to the distance of restarting machining from the current tool position.

Program restart information:

M: Recently specified 35 M codes.

S: The last specified S code

T: Recently specified T code by twice.

The codes are displayed based upon its specified sequence, the overall codes can be specified by the program restart, and the commands can be eliminated at the cycle start in the resetting state.

1. Cut off the 

button, check whether the distance of the residual movement value is correct, and confirm whether the tool may touch the workpiece or other materials when it restarts its machining position, if does, move the tool to the position without any other materials by hand.

2. Press the cycle restart button. Tool moves to the restart machining position one axis to another under the dry run speed along with sequence set by parameter (No.7310), and then the restart machining is performed again.

6.5.2 M.S.T Function Treatment of Program Restart

There are two methods for the miscellaneous function M, S and T after indexing the desired restart block; 1: It executes before moving to the restart machining position; 2: It executes before reaching to the restart machining position;

I. It executes before moving to the restart machining position.

(1) The last M,S and T codes can be automatically output to the PLC.

When parameter MOU (Bit 7 of parameter No.7300) sets to 1 and MOA (Bit 6 of parameter No.7300) sets to 0, after the desired restart block is indexed, press the cycle start switch; and then automatically output the M, S and T codes to the PLC before the tool moves to the restart machining position.

In the single block stop state, the cycle start switch should be controlled again after outputting the last M, S and T codes, the tool then can be moved to the restart machining position.

As for the last S codes, if the S codes are specified in the indentical block of G50 (G code of set Series A)/G92 (G code of Series B), it will then output as the top spindle speed; the rest of S codes will output as the command spindle speed. However, the last S codes display at the program restart screen is only shown the last specified one S code no matter how it is shared the same block with the G50 or G92.

(2) Also, the overall M codes by sampling and the last S and T codes can be automatically output to the PLC after indexing the desired restart block.

When parameter MOU (Bit 7 of parameter No.7300) sets to 1 and MOA (Bit 6 of parameter No.7300) sets to 0, after the desired restart block is indexed, press the cycle start switch; and then automatically output the overall M codes and the last S and T codes to the PLC before the tool moves to the restart machining position.

However, up to 35 M codes can be sampled. If the M codes to be sample is more than 35, the updated 35 codes is then output to the PLC.

<For example> When collecting the M10, M11, M12, M13, M14, T0101 and S1000, before the tool moves to the restart machining position, the program is performed based upon the following format.

M10 T0101 S1000;

M11;

M12;

M13;

M14;

Whether performing (1) or (2) is shifted by parameter MOA (Bit 6 of parameter No. 7300).

II. It executes before reaching to the restart machining position

When the parameter MOU (Bit 7 of parameter No.7300) sets to 1, during the period after the desired self-index restarting block is performed till to the restart machining position, the corresponding M,S and T codes can be specified in MDI mode.

6.5.3 Function Limitation

1. Type P restart

The Type P restart can not be performed in the following conditions:

(1) When the Auto operation is not performed yet after the power is turned on.

(2) When the Auto operation is not performed yet after the ESP resetting is executed.

(3) After the coordinate system or the offset (alter the external workpiece zero offset) is changed, the Auto operation is not performed yet.

2. Restarting block

The restarting block is not the necessary one by interrupting, which can be started from arbitary block. However, if it is the Type P restart, it should be shared the same block with coordinate system during interrupting.

3. Singl block

When th tool moves to the restart position, and when the single block is in the case of the ON, the single block stops after performing each axis operation. However, the MDI interference, in this case, can not be performed.

4. Manual interference

During moving to the restart position, the axis that does not yet perform the operation return is interfered by hand. However, the axis that has been performed the return will not be moved due to the operation return.

1. MDI

After the index is completed, the movement command can not be specified by MDI before performing the axis movement.

6. Resetting

Never attempt to perform the resetting operation during the beginning of the index to the restarting machining. Otherwise, the restarting operation should be performed again from the first step.

7. Feed hold

When the feed hold operation is performed in the index, it is necessary to perform the restarting operation again from the first step.

8. Manual absolute value switch

The manual operation can be performed in the state of the manual absolute method sets to ON (switch on) regardless of the machining is already started.

9. Reference point return

If there is no absolute position detector (absolute pulse encoder), after the power is switched on, the reference position return should be performed before the restarting operation is executed.

10. Program restarting switch

The program restarting will not perform even if the cycle start switch is controlled in the state of power-on.

11. The command can not be restarted the program again.

- Polar coordinate interpolation (G12.1)
- Cylindrical interpolation (G7.1)
- Metric/inch shifting (G20, G21)

- Thread cutting (G32, G33, G34), thread cutting cycle (G92 or G78) and complex fixed thread cutting cycle command (G76)
- Cs outline control
- Tapping command
- Spindle positioning
- Rigid tapping
- Spindle positioning

12. Never attempt to use the M, S and T command in the overstored method.
13. Macro variable treatment

- Polar coordinate interpolation (G12.1) macro statement, the macro program calling and the block called by subprogram are regarded as attachment with sequence number, and therefore, indexing this block will generate “Fail to find the specified sequence number or block number” alarm. In this case, it is necessary the previous block only.
- If the system variable operation generates during the index procedure, the alarm will then occur.

14. The selected program restarting operation is finish-turning shape block of the G71~G73, the alarm will then issue.

6.5.4 Cautions

1. Principally, tool can not return to the correct position in the following situations:
   It is essential to note that the alarm will not generate in the following conditions.

- The manual operation can be performed in the manual absolute OFF.
- When the manual operation is performed in the mechanical locking.
- There is no coordinate system at the beginning of the program when the main command is incremental method.
- When the manual interference is performed during the axis movement of operation return.
- When one block is specified a restarting program between the skip cutting block and the subsequent absolute command block, when indexing the skip, it is retreated as the current skip switch state.
- When the mechanical locking is eliminated after the program restarting is specified in the mechanical locking state.
- When the block at the midway of the compound fixed cycle is specified the program restarting.
- The coordinate system is set, changed and offset after the indexing is completed.

2. The cautions when the program used with macro variable is performed the program restarting operation.

- Common variable
  In the program restarting, the common variable uses the previous one instead of automatically resetting. Therefore, the desired variable should be initialized as the value at the last automation

operation before performing the program restarting.

- DI/DO

In the program restarting, although the DI can be read based upon the system variable, DO can not be output.

- Clock

In the program restarting, although the time can be got from the clock based upon the system variable, the clock can not be reset.

- Tool offset value, workpiece origin offset value

In the program restarting, although the offset value can be read based upon the system variable, the alternation of the offset value is only suitable type Q.

Chapter 7 Tool Offset & Tool Setting

To simplify programming, the actual position of the tool isn't included during programming; GSK988TA/988TA1/988TB provides the tool setting method, like tool setting in the fixed position and the trial tool cutting, and the tool offset data can be acquired by tool setting.

7.1 Setting the Tool Offset Value and Wearing Value

7.1.1 Direct inputting Method

（1）On the setting page set, press TOOL SETTING software key to enter the tool offset administration page, which is shown as Fig. 7-1:

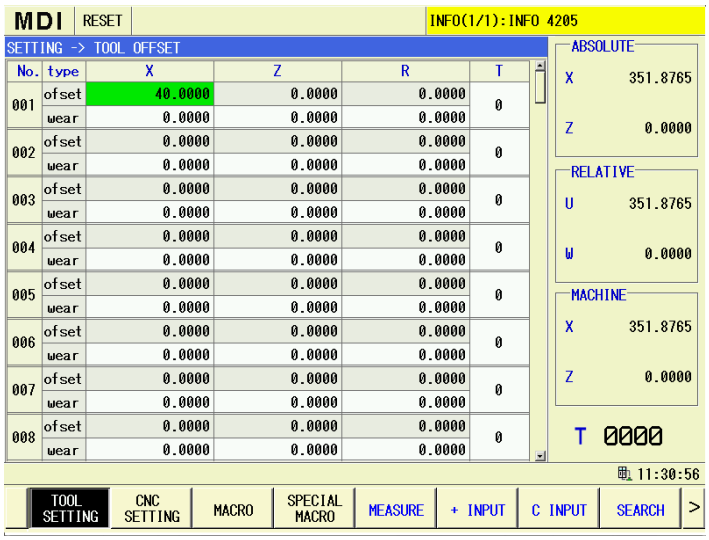


Fig.7-1

（2）On the page, press the page up key and page down key to select the page, press the up/down direction keys , to select the tool offset number to be rewritten and press the left/right direction keys , to select the axis tool offset data, the wearing data to be rewritten or T value in the assumed tool nose direction, which is like X axis offset of #001 tool offset in the above figure; About the corresponding relation of the assumed tool nose, refer to *The Programming Manual*.

（3）The tool offset data, the wearing data or the corresponding assumed tool nose direction number T can be directly rewritten with the digit key and the backspace key; or the selected tool offset value can be input by pressing INPUT key, which is like X axis offset of the tool offset of #001 in Fig. 7-2, and then the tool offset data, the wearing data or the corresponding assumed tool nose direction number T can be rewritten with the digit key and the backspace key;

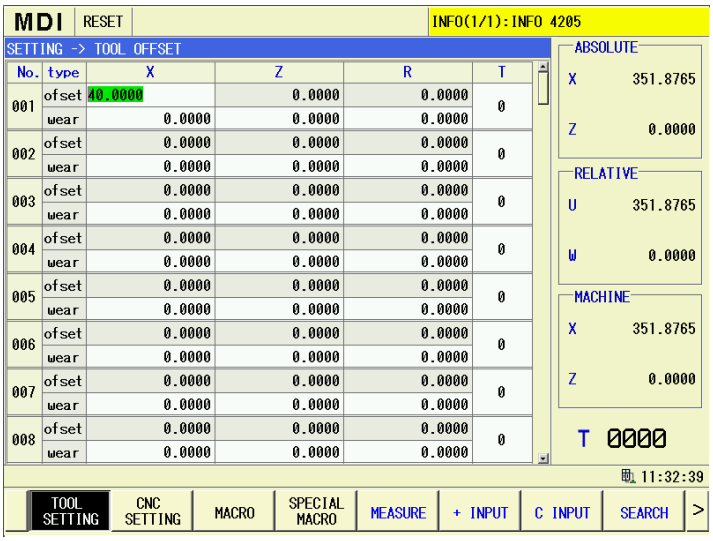


Fig.7-2

（4）Press INPUT key on the edit keypad to confirm inputting or rewriting is completed.

（5）Move the cursor to set the other tool offset value, the wearing value or the assumed tool nose direction T value.

Note: The maximum setting value of the tool wearing compensation value can be rewritten by the parameter 5013.

7.1.2 Measuring Input Mode

（1）On the setting page set, press TOOL SETTING software key to enter the tool offset administration page;

（2）Press the page up key and page down key to select the page, press the up/down direction keys ⇧, ⇩ to select the tool offset number to be rewritten and press the left/right direction keys ⇦, ⇨ to select the axis tool offset data and the wearing data to be rewritten.

（3）Press MEASURE software key to enter the measuring input page for measuring the tool offset value, which is shown as Fig.7-3:

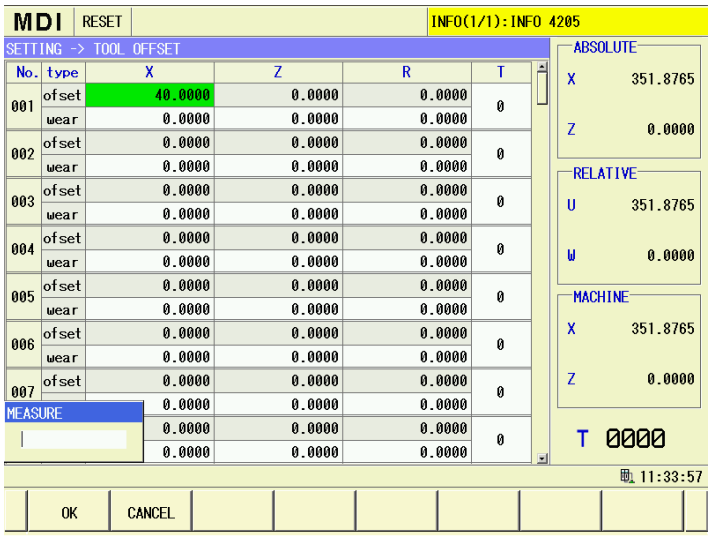
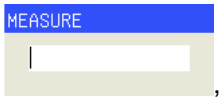
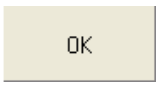


Fig.7-3

（4）In 

, input “the coordinate axis number + coordinate value” to be measured, press 

software key or 

key for positioning measuring;

（5）Calculating the tool offset value:

If the cursor is on the tool offset bar, the tool offset value is cleared into 0, the tool offset value = the relative coordinate value – the input coordinate value;

If the cursor is on the tool wearing bar, the tool wearing value remains unchanged, the tool offset value = the relative coordinate value – the input coordinate value – the wearing value corresponding to the coordinate axis.

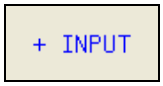
Note: The relative coordinate value is the value not set in the relative coordinate in the coordinate system; if the relative coordinate value has been set, the value cleared by the relative coordinate is automatically added.

7.1.3 + Input Mode

（1）On the setting page set, press 

software key to enter the tool offset administration page;

（2）Press the page up key  and page down key  to select the page, press the up/down direction keys  , to select the tool offset number to be rewritten and press the left/right direction keys  , to select the axis tool offset data and the wearing data to be rewritten;

（3）Press 

software key, add one input value to the current selected tool offset value or the wearing value, which is shown as Fig.7-4:

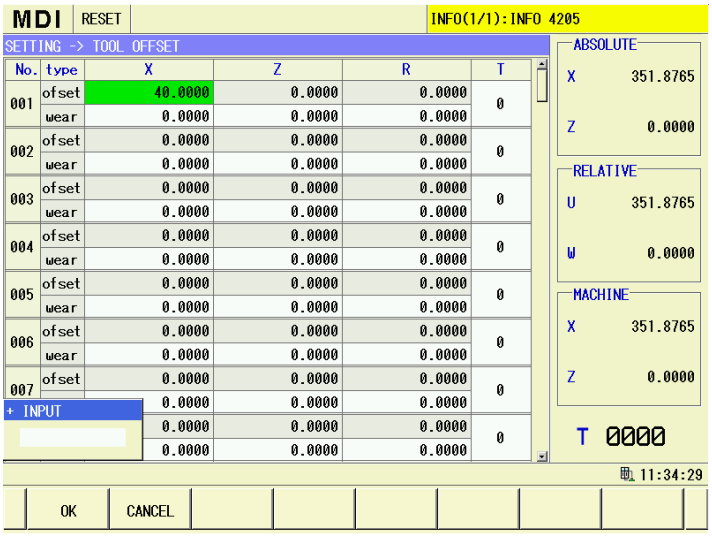
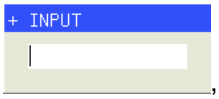


Fig.7-4

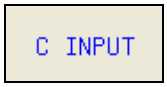
（4）Input one numerical value in  , the input numerical value is negative. Press  software key or  key to complete inputting;

（5）Calculating the offset value: The offset value or the wearing value = the original offset value or the wearing value + the input numerical value.

7.1.4 C Input Mode

（1）On the setting page set, press  software key to enter the tool offset administration page;

（2）Press the page up key  and page down key  to select the page, press the up/down direction keys  ,  to select the tool offset number to be rewritten and press the left/right direction keys  ,  to select the axis tool offset data and the wearing data to be rewritten.

（3）Press  software key to enter C input page, which is shown as Fig.7-5:

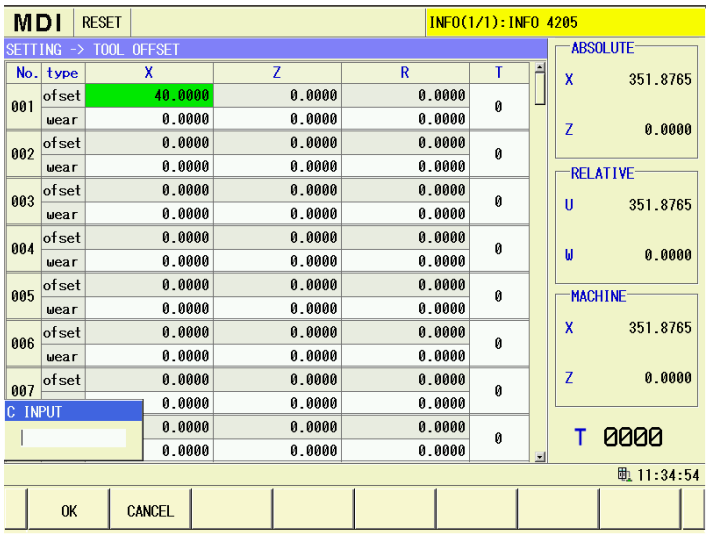
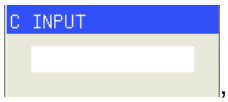


Fig.7-5

（5）Input the coordinate axis name in 

, press 

software key for the positioning measuring;

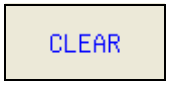
（6）Then, calculate the offset value:

Press C input button to input the axis number.

If the cursor is on the tool offset bar, the tool wearing remains unchanged, the rewritten tool offset value = the relative coordinate value – the tool wearing value;

If the cursor is on the tool wearing bar, the tool offset value remains unchanged, the rewritten tool wearing value = the relative coordinate value – the tool offset value.

7.1.5 Clearing the Tool Offset Value or the Wearing Value

On the tool offset administration page, press the page up key  or the page down key  to select the page, press the up/down direction keys  ,  to select the tool offset number to be rewritten and press the left/right direction keys  ,  to select the tool offset data, the wearing data or the tool number to be cleared; press 

software key to clear the current selected tool offset value or the wearing value corresponding to the axis or to clear the assumed tool nose direction number.

7.2 Tool Setting in the Fixed Position

The tool setting in the fixed position is to set the tool offset data with C input mode.

The operation steps are as below:

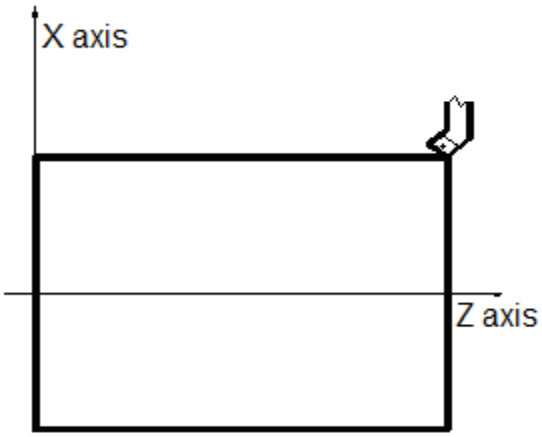


Fig.7-6

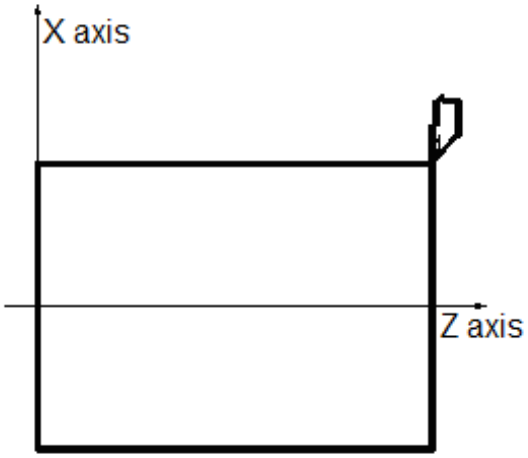


Fig.7-7

（1）Firstly, confirm whether the tool compensation value is zero or not in X and Z directions; if it is not zero, all cutter compensation values of all tool numbers must be cleared.

（2）The offset number of the tool is 00 (such as T0100，T0300);

（3）Select any tool (normally, the 1st tool during machining is taken as the reference tool)

（4）The tool nose of the reference tool is positioned in some point (tool setting point), which is shown as Fig.7-6:

（5）In MDI mode, on the program status page, the workpiece coordinate system is set by G50 X__ Z__ command;

（6）The coordinate values of the relative coordinate (U，W) are cleared;

（7）After the tool is moved into the safe position, and the other tool is selected and moved into the tool setting position, which is shown as Fig.7-7:

（8）On the setting page set, press TOOL SETTING software key to enter the tool offset administration page and then press SEARCH software key to select the tool offset number or press the page up key or the page down key to select the page, press the up/down direction keys , to select the tool offset or the wearing data to be rewritten;

（9）Press C INPUT software key to enter C input page, and input the axis name on the input page C INPUT, and after pressing OK software key, the tool offset value or the wearing value is set into the corresponding offset number;

（10）Other tools can also be set by repeatedly executing the steps 7～9.

7.3 Trial Tool Cutting (The Machine Zero Return Tool Setting)

The tool setting method doesn't exist the reference tool, when one tool is worn or should be adjusted, the tool is just reset. The machine zero should be returned before tool setting; after power off, the machining can be continued just after power on, again, and the operation is simple and convenient.

The trial tool setting is to set the tool offset value with the measuring input method after the coordinate system is set.

The operation steps are as below: (the workpiece coordinate system is set on the workpiece end face):

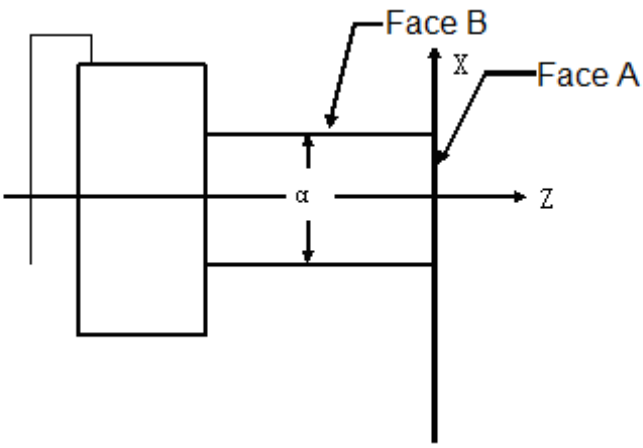


Fig.7-8

（1）Confirm that each axis of the machine tool has already returned the machine zero.

（2）Select any tool and set the tool offset number as 00 (like T0100，T0300)

（3）The tool is moved along face A;

（4）When Z axis is not moved, the tool is retracted along X axis, and the spindle is stopped revolving;

（5）On the setting page set, press TOOL SETTING software key to enter the tool offset administration page, and then press SEARCH software key to select the tool offset number or press the page up key or the page down key to select the page, press the up/down direction keys ⇧, ⇩ to select the tool offset or the wearing data to be rewritten;

（6）Press MEASURE software key to enter the measuring input page, and input Z0 on the input page MEASURE, and after pressing OK software key, the tool offset value or the wearing value of Z axis is set into the corresponding offset number;

（7）The tool is moved along face B;

（8）When X axis is not moved, the tool is retracted along Z axis and the spindle is stopped

revolving

（9）Measure the diameter "α" (Assume α=15.0）;

（10）Press MEASURE software key to enter the measuring input page, and input X15.0 on the input page MEASURE, and then OK software key is pressed, the tool offset value or the wearing value of X axis is set into the corresponding offset number;

(11) Move the tool into the safe tool change position, and change the tool;

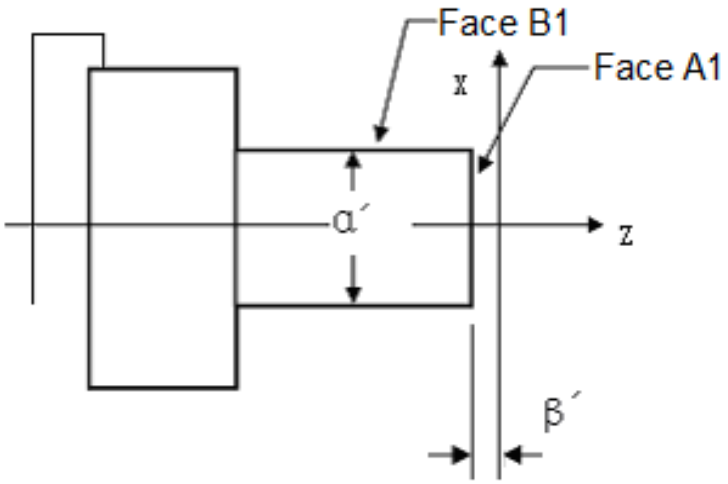


Fig.7-9

(12) The tool is cut along surface A1, which is shown as Fig. 7-9;

(13) When Z axis is not moved, the tool is retracted along X axis and the spindle is stopped revolving; Measure the distance "β´" between surface A1 and the origin of the workpiece coordinate system (assume β´= 1.0);

(14) On the setting page set, press TOOL SETTING software key to enter the tool offset administration page, and then press SEARCH software key to select the corresponding tool offset number or press the page up key or the page down key to select the page, press the up/down direction keys ⇧, ⇩ to select the tool offset or the wearing data to be rewritten;

(15) Press MEASURE software key to enter the measuring input page, and input Z-1.0 on the input page MEASURE, and then OK software key is pressed, the tool offset value or the wearing value of Z axis is set into the corresponding offset number;

(16) The tool is cut along surface B1;

(17) When X axis is not moved, the tool is retracted along Z axis and the spindle is stopped revolving;

(18) Measure the distance "α′" （assume α′＝14.5）;

(19) Press MEASURE software key to enter the measuring input page, and input X＝14.5 on the input page MEASURE, and OK software key is pressed, the tool offset value or the wearing value of X axis is set into the corresponding offset number;

(20) Other tools can also be set by repeatedly executing the steps 11～19.

The measuring input method is to set the differential value between the tool reference position (such as, the tool nose position) and the tool nose position of the tool actual used during machining as the tool offset value. For example, when the coordinate value of face B is 50.0, the actual measured α=49.0, so the tool offset value in X direction of this tool offset is 1.0.

Note: After the tool setting with the machine zero, G50 can't be commanded for setting the workpiece coordinate system.

7.4 Position Record

When the parameter PRC（NO.5005#2）is set as 1, the position record button on the machine operation panel is valid.

The method of tool setting with position record:

1）The excircle or the end face of the workpiece is cut in Manual mode.

2）After pressing TOOL OFFSET button on the operation panel on the machine side, the workpiece coordinate values of X axis (three basic axes) and Z axis (three basic axes) are recorded into CNC.

3）And then, the tool is retracted, if it is in the excircle direction, the diameter should be measured. After press MEASURE, input X+ measured value in the tool compensation number corresponding to the tool offset, and then press enter. On the end face, measure the length off the reference face, (the reference face is assumed as Z=0), and press MEASURE software key, input Z+ measured value in the tool compensation number corresponding to the tool offset, and then press enter.

Note 1: If TOOL OFFSET button is pressed for many times, CNC only records the coordinate position when TOOL OFFSET button is pressed at the last time.

Note 2: During the tool setting in the fixed position, the position record is also used: After setting the

**reference tool, press 

button after the other tools reach the tool setting position, and the current coordinate position is recorded, and the tool offset is input based on the tool setting in the fixed position after tool retraction.**

7.5 Automatic Tool Compensation

If the machine tool is installed with the automatic tool setting device, CNC can send the command for automatic measuring, and then CNC can automatically measure or determine the tool compensation amount. Firstly, the command for measuring is sent and the tool is moved into the measured position. CNC automatically measures the difference between the coordinate value of the measuring point and that of the commanded (the expected) measuring position and the difference is taken as the compensation amount of the tool. When the tool has been compensated, the tool is moved into the measured position after compensated. The difference between the coordinate value of the measured point and that of the commanded coordinate value is accumulated into the current set compensation amount.

Note: The parameter IGA（NO.6140#7）is set as 0 when the automatic tool compensation function is used.

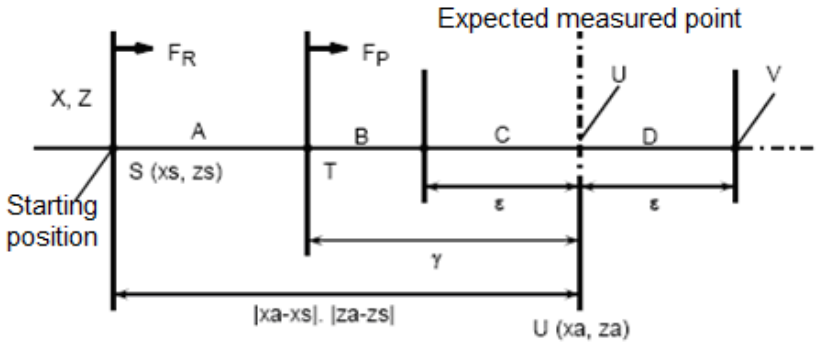
Automatic measuring code:
X axis: G36
Z axis: G37

The measuring position arrival signal
XAE(X0.6 1st path signal/X0.0 2nd path signal) ————corresponding to G36
ZAE(X0.7 1st path signal/X0.1 2nd path signal) ————corresponding to G37

Usage of G36 and G37 automatic tool offset codes

When G36 or G37 is in the commanded block, the tool is moved into the commanded measured position in the rapid traverse mode，which is shown as Fig.7-10. Moreover, the tool is decelerated and stopped in the position which is the distance γ from the measured position, and then moved into the measured position at the measured speed set by the parameters (No.6241～6243).

Moreover, close to the distance ε, only in the overtravel distance ε, when the measured position arrival signal corresponding to the program command becomes "1", and the above compensation amount is refreshed meanwhile, the movement command of the block ends. If the measured position arrival signal doesn't become "1" in the measured position overtravel distance ε, the control device enters the alarm status, and the movement command of the block ends without refreshing the compensation amount.



FR: Rapid movement
FP: Feedrate set by the parameter (No.6241)
γ: Parameter (No.6251, No.6252)
ε: Parameter (No.6254, No.6255)

Fig.7-10

Note 1: About the detailed usage of G36, G37, please refer to *GSK988TA/988TA1/988TB Programming Manual*.

Note 2: About the usage of the automatic tool setting device, please refer to the user manual provided by the machine tool manufacturer;

Note 3: Parameter 6241: The feedrate of X axis is set during automatic tool compensating;
Parameter 6251: γ value of X axis is set in the automatic tool compensation function;
Parameter 6254: ε value of X axis is set in the automatic tool compensation function;
Parameter 6242: The feedrate of Z axis is set during automatic tool compensating;
Parameter 6252: γ value of Z axis is set in the automatic tool compensation function;
Parameter 6255: ε value of Z axis is set in the automatic tool compensation function;

Chapter 8 Graph Setting & Display

8.1 Setting the Graph Parameters

Before setting the execution path, the relative information, like the path display or the graph simulation, is set.

Setting the graph information is mainly for the graph display, like the offset amount of each coordinate axis, the machining length, the diameter, the graph magnification and the graph simulation proportion. The detailed setting steps are as below:

（1）Press GRAPH function key to enter the graph page set;

（2）On the graph page set, press GRAPHSET software key to enter the graph parameter setting page, which is shown as Fig. 8-1:

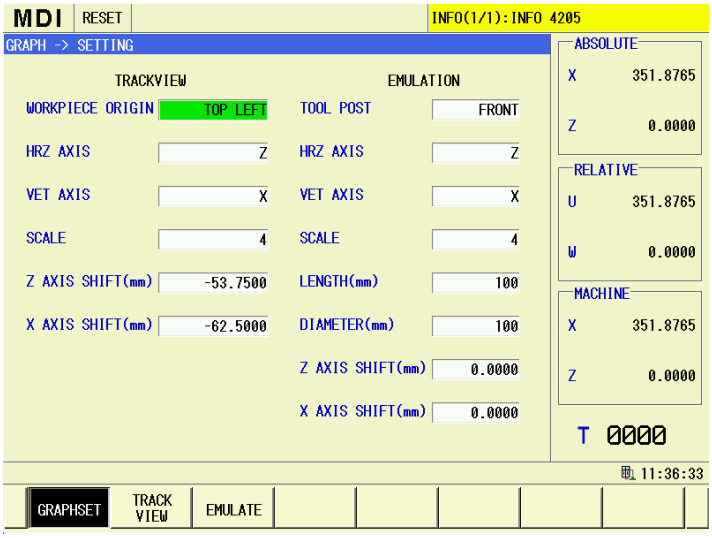


Fig.8-1

（3）Press ⇦, ⇨ to switch between two channels;

（4）Press ⇧, ⇩ key to select the setting item, like selecting the horizontal axis shown as the above figure;

（5）Press INPUT key to make the selected item to be input;

（6）Press ⇧, ⇩ key to select the setting item, and then press INPUT key to confirm the rewriting is completed.

（7）Repeatedly operate to set other parameters.

Note: 1. In this page, the setting is only used the path display and diagram simulation

2. The path display and the graph simulation all are executed with the machine coordinate; if the path and the graph are not displayed, the coordinate axis offset should be rewritten.

3. The horizontal axis must be the basic Z axis or the axis parallel to Z axis.

8.2 Path Graph Display and Operation

The tool path can be real-time checked with the graph path display.

（1）Press GRAPH function key to enter the graph page set;

（2）On the graph page set, press TRACK VIEW software key to enter the path display page and the program path currently executed is displayed, which is shown as Fig. 8-2:

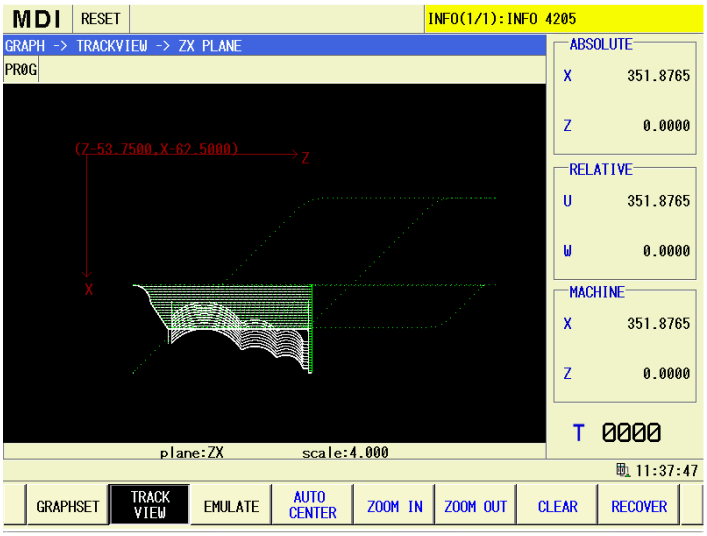


Fig.8-2

In the figure, the current displayed coordinate plane and the path graph magnification are displayed below the path display screen. On the graph top, the current system running mode and status are displayed, on the screen right side, the current absolute coordinate value, the relative coordinate value and the modal command are displayed.

The path graph is operated as below:

（1）Press ZOOM IN, ZOOM OUT software key to scale up/down the path graph and the previous path graph is cleared;

（2）Press CLEAR software key to clear the screen path;

（3）Press RECOVER software key to restore the path graph into the original normal position, and the previous path graph is cleared.

（4）Respectively press ⇧, ⇩, ⇦ and ⇨ keys to move the path graph upward/downward/leftward/rightward.

Note 1: The name of each axis is set by the parameter #1020, and each axis is set as the different names with the letters, and then, on the path display page, the name of each coordinate plane and that of the path coordinate can be switched correspondingly.

2. The system is only with the front and back tool posts without the left and right ones, so the horizontal axis can only be set as Z axis of the basic one.

8.3 Simultaneous Graph Display and Operation

All cutting process of the part can be real-time checked with the graph simulation.

（1）Press GRAPH function key to enter the graph page set;

（2）On the graph page set, press EMULATE software key to enter the simulation graph display page, which is shown as Fig. 8-3:

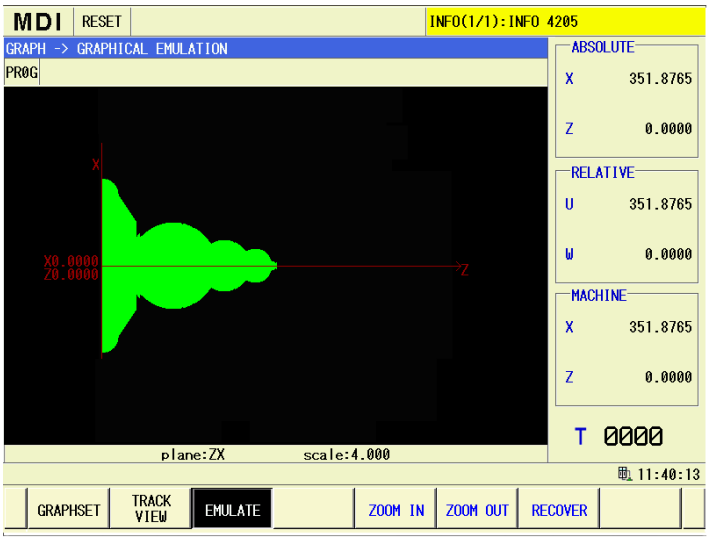


Fig.8-3

On the graph top, the running mode and status of the current system is displayed, the screen right side is displayed the current absolute coordinate value, the relative coordinate value and the current tool number, etc.

In the figure, the simulation graph information on XZ coordinate plane is displayed, the coordinate plane on which the current simulation graph is and the magnification of the simulation graph is displayed at the bottom of the graph simulation display screen.

During the graph simulation process, the simulation graph is operated as below:

（1）Press ZOOM IN, ZOOM OUT software key to scale up/down the simulation graph, and the previous simulation graph is cleared;

（2）Press RECOVER software key to restore the simulation graph into the original size and the position, and the previous simulation graph is cleared;

（3）Respectively press ⇧, ⇩, ⇦, ⇨ key to move the simulation graph upward/downward/leftward/rightward.

Note: The name of each axis is set by the parameter #1020, and each axis is set as the different names with the letters, and then, on the path display page, the name of each coordinate plane and that of the path coordinate can be switched correspondingly.

Chapter 9 Usage of USB Flash Disk

9.1 Sending the Program

Firstly, a new file folder is created under the root content of the USB flash disc, and the file folder is renamed as NCPROG and the program to be sent is copied into the file folder, which is shown as Fig. 9-1:

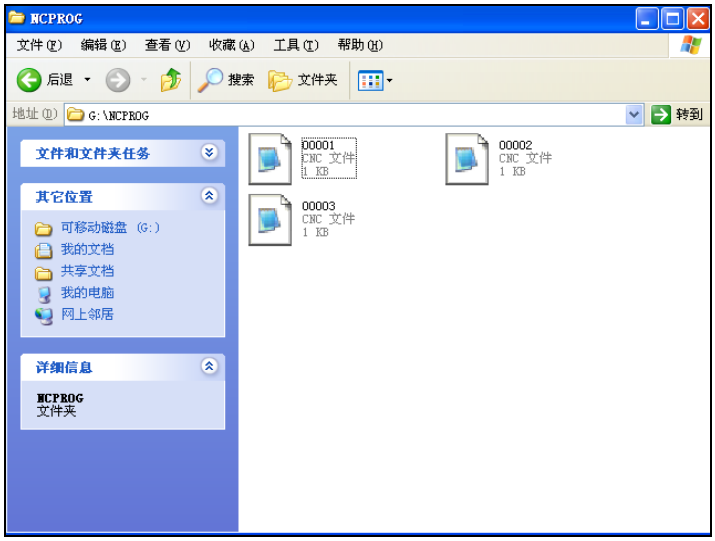


Fig.9-1

After the above operation, the USB flash disc is inserted into the USB interface of the system communication area, the symbol  occurs at the right lower corner of the system screen, which means the USB flash disc has been connected; and then, press PROGRAM, and press USB software key to enter the USB file content, which is shown as Fig. 9-2, and the program to be copied is selected by pressing ⇧, ⇩ cursor keys and press OUTPUT software key, and the selected program in the USB flash content is copied into the local content.

For example, O0002 program in the USB flash disc is copied into the local content, O0002 program in the USB flash disc content is selected by the cursor, and > software key is pressed, and then OUTPUT is pressed, O0002 program is copied into the local content.

The program of the local content is copied into the USB flash disc, and the user operates in the local content page based on the previous steps.

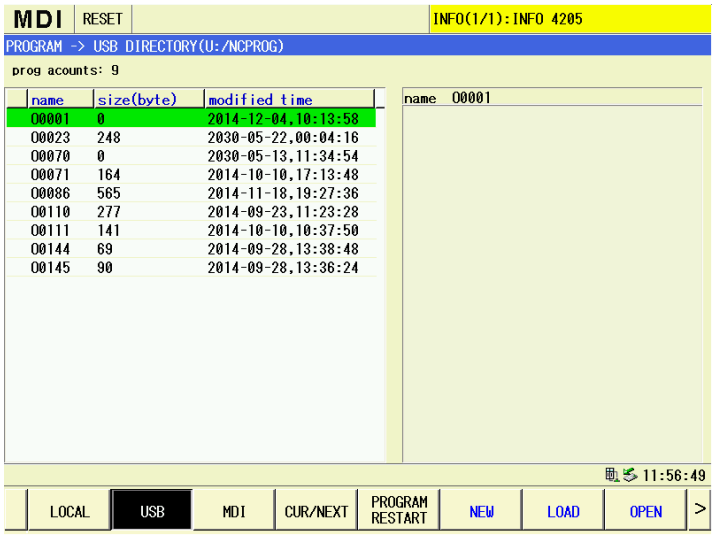


Fig.9-2

9.2 Data Backup

GSK988TA/988TA1/988TA system can backup the system file and the parameter into the USB flash disc for restoring later.

9.2.1 System File Backup

The data, like the system parameters, the tool offset, the screw pitch compensation, the tool lifetime and the macro variable, etc can backup with the USB flash disc, so the restoring can be operated to avoid the misoperation when the data error occurs. The operation steps are as below:

1. Insert the USB flash disc to confirm the system has already read the USB flash disc;

2. Press SYSTEM function key to enter the system page and press MEMORY DEVICE software key to enter the file management display page, and the page is shown as Fig. 9.3:

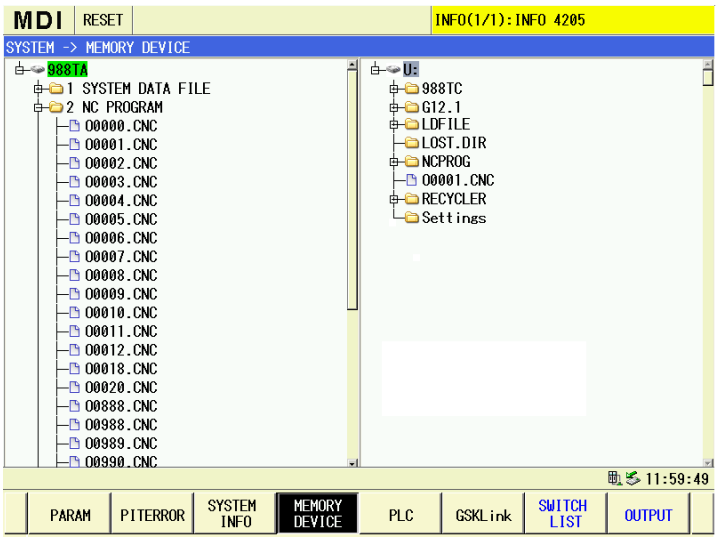


Fig.9-3

3. On the page, there are 5 sub-files under the system file, they are respectively: MACRO.MCO

macro variable, PARAM.PAR parameter, TLIF.TLL tool lifetime, TOFF.CMP tool offset and WOFF.WMP screw pitch compensation. The user should move the cursor upward/downward on the file to backup, and press INPUT key to select the file, and the cursor is on the file, and then, press INPUT key to select all files of the folder, which is shown as Fig. 9-3.

4. After selecting the file, OUTPUT software key is pressed, “please select the output path” is popped up, and OK software key is pressed, the file is copied. After copying the file is completed, please pull out the USB flash disc.

5. When the backup should be restored, the user should insert the USB flash disc, press SWITCH LIST software key to switch the cursor into the USB flash disc directory, move the cursor to find the file to restore the data and press INPUT key to select and then press OUTPUT to directly cover the file under the local content. After restoring the file, please power on the system, again; otherwise, some data may become invalid.

Note 1: For restoring the parameters, the parameter switch must be ON, and some parameters can’t be restored because the authority level is too low.
2. Restoring the screw pitch compensation must be with the authority level 2.

9.2.2 Backup of Servo Parameter

9.2.2.1 Lead-out of Servo Parameter

1. Insert the U disk, and then confirm the system is already read it.

2. Enter the system page by SYSTEM function button, and then the GSKLink ⇨ SERVO softkey to enter the servo administration display page, lastly press SERVO PARAM softkey to enter the servo parameter page. The page display is shown as the Fig.9-4:

Fig.9-4

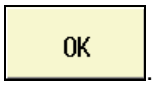
3. The above-mentioned figure shows: select X axis servo parameter, press  , and then press the  , refer to the Fig.9-5; the leading-out file name changes into X, because the previous selected one is X axis; the parameter file of X axis can be backup to the U disk by  .

Fig.9-5

4. Select the other axes, and then repeat the above operations, subsequently, the parameter of other axes can be backup to the U disk.

5. The folder “SERVOPARAM” can be set up in the U disk after the operations are performed by the step 3, the parameter file with backup before will reserved at this folder. Refer to the Fig 9-6.

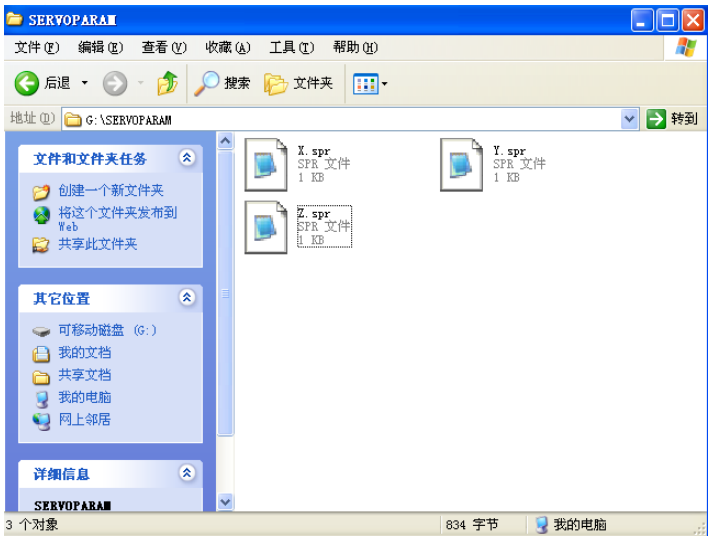


Fig.9-6

9.2.2.2 Leading-in of Servo Parameter

1. Ensure that the folder “SERVOPARAM” is already set up in the U disk, and the servo by backup holds at this folder. Refer to the above figure 9-6.

2. Insert a U disk, confirm that the system is already read to the U disk.

3. On the servo parameter administration page, select the desired parameter axis, for example, X axis, press [>] softkey, and then [IMPORT PARAM]; select the correct parameter file in the shown dialog, the parameter in the U disk can be led to the system by [OK].

4. Repeat the above-mentioned operations, the parameter file of other axes can be led to the system accordingly.

Note 1: The machine tool can only be used after the power is turned on again if the servo parameters of the overall axes are led in.

Note 2: The leading-in and leading-out of the servo parameter files should be operated in the MDI mode and the 3 levels of the operation authorities.

Chapter 10 Machine Example

10.1 Excircle End Face Machining

1） The workpiece is machined as Fig. 10-1, and the bar stock dimension is Φ50mm×100mm.

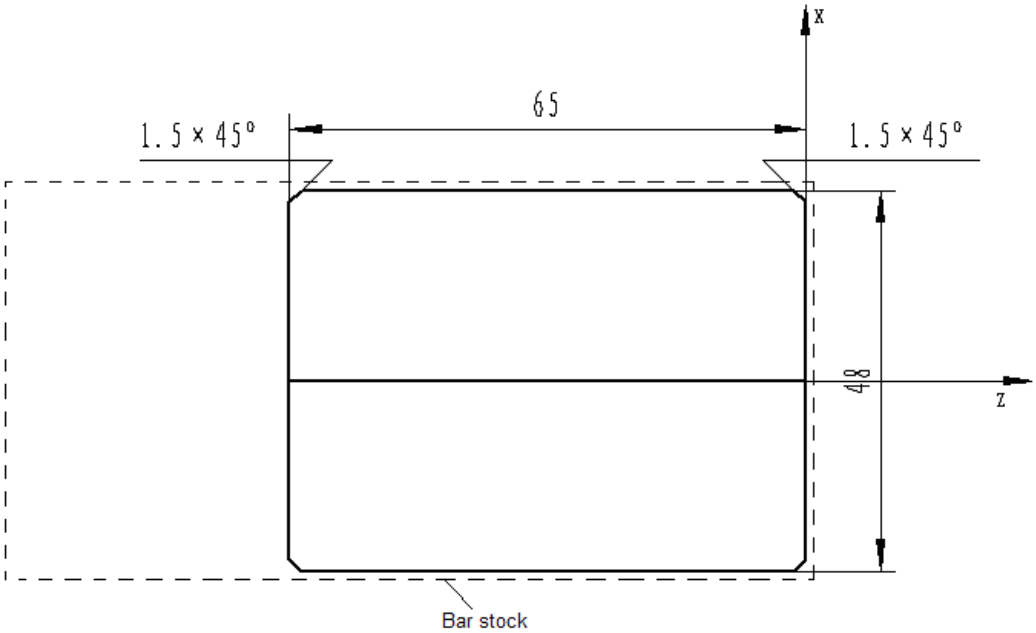
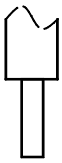


Fig.10-1

2）Two tool are machined, and the details are as below:

Tool number	Tool shape	Remark
#1 tool		Excircle tool
#2 tool		Cut-off tool, the tool width is 3mm.

3）Editing the program

Based on the explanation of the codes of the mechanical machining process and the manual, the workpiece coordinate system is set shown as Fig.10-1, the program is edit as below:

O0001;		Part program name
N00000	G0 X150 Z150;	Position to the safe position for tool change
N00010	M12;	Clamp the chuck
N00020	M3 S800;	800 Switch on the spindle and the speed is 800
N00030	M8;	Switch on the coolant
N00040	T0101;	Change the 1st tool
N00050	G0 X51 Z2;	Close to the workpiece
N00060	G01 X49 F150;	The excircle is machined twice, and feed 1mm at the 1st time
N00070	G01 Z-69 F150;	MachineΦ49 excircle
N00080	G0 X51;	Tool retraction
N00090	Z2;	Retract to the starting position
N00100	X0;	X axis feeding
N00110	G01 Z0 F100;	Z axis feeding
N00120	X45;	Turning end face
N00130	X48 W-1.5;	Machine chamfering angle 1.5*45
N00140	Z-69;	TurningΦ48 excircle
N00150	X51;	Retraction in X direction
N00160	G0 X150 Z150;	Return to the tool change point after the excircle machining is completed
N00170	T0202;	Change to #2 tool and execute #2 tool offset
N00180	G0 Z-68 X50;	Close to the workpiece
N00190	G01 X43 F100;	Grooving
N00200	G01 X49 F300;	Return
N00210	G01 W1.5;	
N00220	G01 X48;	
N00230	G01 X45 W-1.5 ;	Chamfering
N00240	G01 X0 F80;	Cut off
N00250	G0 X50;	Retraction
N00260	G0 X150 Z150;	Return to the safe position

N00270	T0100;	Change to #1 tool
N00280	M5;	Switch off the spindle
N00290	M9;	Switch off the coolant
N00300	M13;	Release the chuck
N00310	M30;	Program end

4） Tool setting and running

（1）Move the tool into the safe position, execute T0100 in MDI mode and on the program status page, change the tool and cancel the tool offset;

（2）Move the tool and the tool is cut along the workpiece end face, which is shown as Fig.10-2;

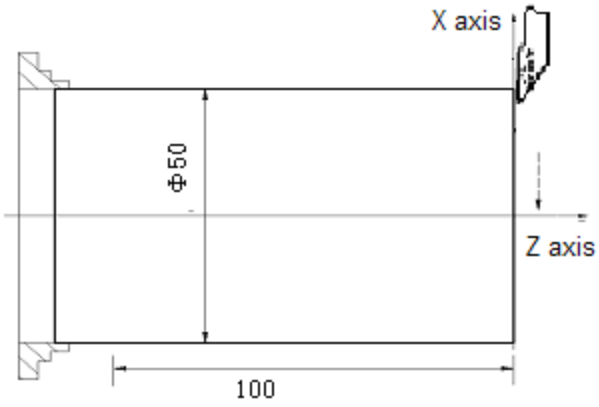


Fig.10-2

（3）When Z axis remains still, the tool is released along X axis, the spindle is stopped revolving, and the system switches into the tool offset page, and the cursor is moved toward #001 offset, the software key MEASURE is pressed to enter the measuring input page, Z0 is input on the input page MEASURE, and OK software key is pressed, the tool offset value of Z axis is input.

（4）The tool is moved and the tool is cut along the workpiece excircle, which is shown as Fig. 10-3;

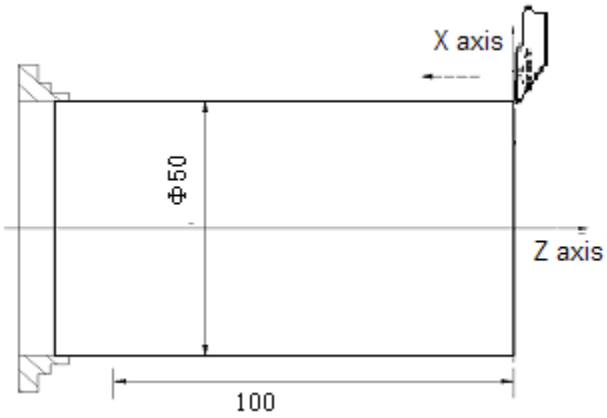


Fig.10-3

（5）When X axis remains still, the tool is released along Z axis, the spindle is stopped revolving, the workpiece excircle dimension is measured (the measured value is 49.5mm); the system switches to the tool offset page and moves the cursor to #001 offset, and MEASURE software key is pressed, X axis tool offset is input.

（6）Move the tool into the safe position, and execute the 2nd tool by pressing the tool change key in Manual mode;

（7）Start the spindle and move the tool into the tool setting position, which is shown as Fig. 10-4, like point A:

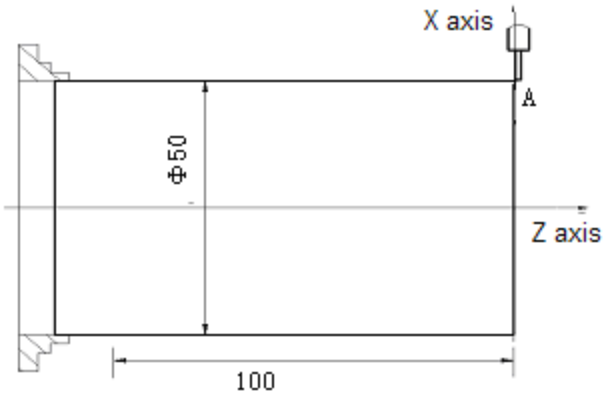


Fig.10-4

(8) The system switches into the tool offset page, and the cursor is moved toward #002 offset, the software key MEASURE is pressed to enter the measuring input page, X49.5 is input on the input page MEASURE, and OK software key is pressed, Z0 is input with the same method;

(9) After the tool setting is completed, the tool is moved into the safe position;

(10) In Auto mode, press 

for automatic machining;

(11) Measure the workpiece dimension, if it is different with the actual part dimension, the tool wearing value can be rewritten until the part dimension is within the tolerance limit.

10.2 Combined Machining

1）The workpiece is machined as Fig. 10-5 and the bar stock dimension is Φ136×190mm.

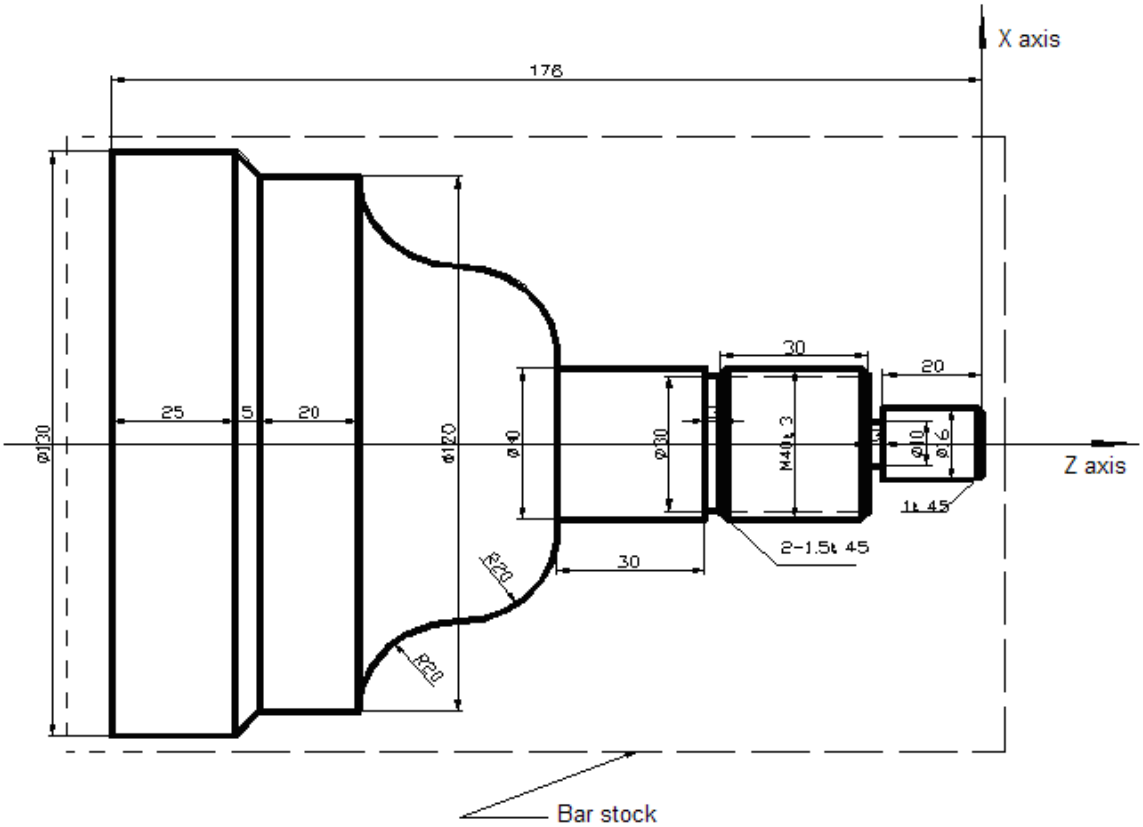
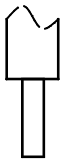


Fig.10-5

2）The tools of four types are machined, and the details are as below:

Tool number	Tool type	Remark
#1 tool		Excircle roughing tool
#2 tool		Excircle finishing tool

Tool number	Tool type	Remark
#3 tool		Groove tool, the tool width is 3mm.
#4 tool		Thread turning tool, the tool nose angle is 60°.

3） Editing the program

Based on the explanation of the codes of the mechanical machining process and the manual, the workpiece coordinate system is set shown as Fig. 10-1, the program is edit as below:

O0001;		Part program name
N0000	G0 X150 Z50;	Position to the safe position for tool changing
N0005	M12;	Clamp the chuck
N0010	M3 S800;	Switch on the spindle, the speed is 800
N0020	M8;	Switch on the coolant
N0030	T0101;	Change the 1st tool
N0040	G0 X136 Z2;	Close to the workpiece
N0050	G71 U0.5 R0.5 F200;	Cutting depth is 1mm and the tool retraction for 1mm
N0055	G71 P0060 Q0150 U0.25 W0.5;	X axis is reserved 0.5mm surplus, Z axis is reserved 0.5mm surplus
N0060	G0 X16;	Close to the workpiece end face
N0070	G1 Z-23;	TurningΦ16 excircle
N0080	X39.98;	Turning the end face
N0090	W-33;	TurningΦ39.98 excircle
N0100	X40;	Turning the end face
N0105	W-30;	TurningΦ40 excircle
N0110	G3 X80 W-20 R20;	Turning the convex arc
N0120	G2 X120 W-20 R20;	Turning the concave arc
N0130	G1 W-20;	TurningΦ120 excircle
N0140	G1 X130 W-5;	Turning taper

N0150	G1 W-25;	TurningΦ130 excircle
N0160	G0 X150 Z185;	Roughing is completed and the tool change position is returned
N0170	T0202;	Change #2 tool and execute #2 tool offset
N0180	G70 P0060 Q0150;	Turning cycle
N0190	G0 X150 Z185;	Roughing is completed and the tool change position is returned
N0200	T0303;	Change #2 tool and execute #2 tool offset
N0210	G0 Z-56 X42;	Close to the workpiece
N0220	G1 X30 F100;	CutΦ30 groove
N0230	G1 X37 F300;	Return
N0240	G1 X40 W1.5;	Chamfering
N0250	G0 X42 W30;	Leave the grooving tool width
N0260	G1 X40 ;	
N0262	G1 X37 W1.5;	Chamfering
N0264	G1 X10;	CutΦ10 groove
N0266	G0 X17 Z-1;	
N0268	G1 X16;	
N0270	G1 X14 Z0 F200;	Chamfering
N0280	G0 X150 Z50;	Return to the tool change point
N0290	T0404 S100;	Change #4 tool and set the spindle speed 200
N0300	G0 X42 Z-54;	Close to the workpiece
N0310	G92 X39 W-34 F3;	Cut the thread cycle
N0320	X38;	Feeding for 1mm and cut for the 2nd time
N0330	X36.4;	Feeding for 0.6mm and cut for the 3rd time
N0332	X36;	Feeding for 0.4mm and cut for the 4th time
N0340	G0 X150 Z50;	Return to the tool change point
N0350	T0100;	Change into #1 tool
N0360	M5;	Switch off the spindle

N 0 3 7 0	M9;	Switch off the coolant
N 0 3 8 0	M13;	Release the chuck
N 0 3 9 0	M30;	Program end

4） Tool setting and running

（1）Move the tool into the safe position, execute T0100 in MDI mode and on the program status page, change the tool and cancel the tool offset;

（2）Move the tool and the tool is cut along the workpiece end face, which is shown as Fig. 10-6;

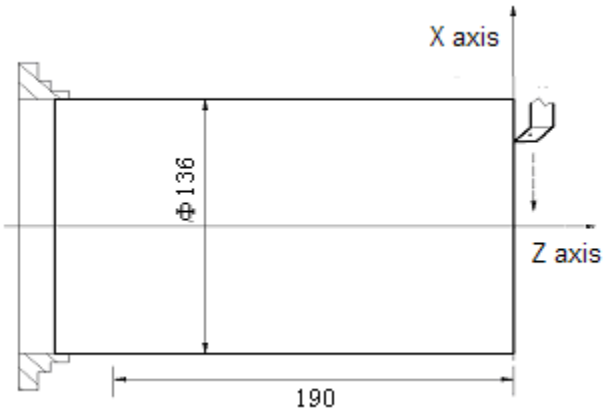


Fig.10-6

（3）When Z axis remains still, the tool is released along X axis, the spindle is stopped revolving, and the system switches into the tool offset page, and the cursor is moved toward #001 offset, the software key MEASURE is pressed to enter the measuring input page, Z0 is input on the input page MEASURE, and OK software key is pressed, the tool offset value of Z axis is input.

（4）The tool is moved and the tool is cut along the workpiece excircle, which is shown as Fig. 10-7:

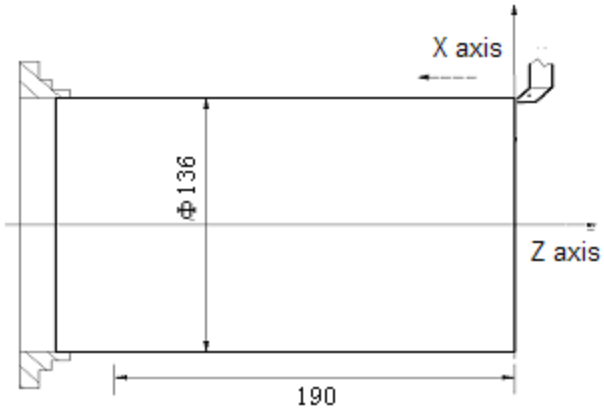


Fig.10-7

（5）When X axis remains still, the tool is released along Z axis, the spindle is stopped revolving,

the workpiece excircle dimension is measured (the measured value is 135mm); the system switches to the tool offset page and moves the cursor toward #001 offset, and MEASURE software key is pressed to enter the measuring input page, and X135 is input on the input page MEASURE, and then OK is pressed, X axis tool offset is input.

（6）Move the tool into the safe position, and execute the 2nd tool by pressing the tool change key in Manual mode;

(7) Start the spindle and move the tool into the tool setting position, which is shown as Fig. 10-8, point A:

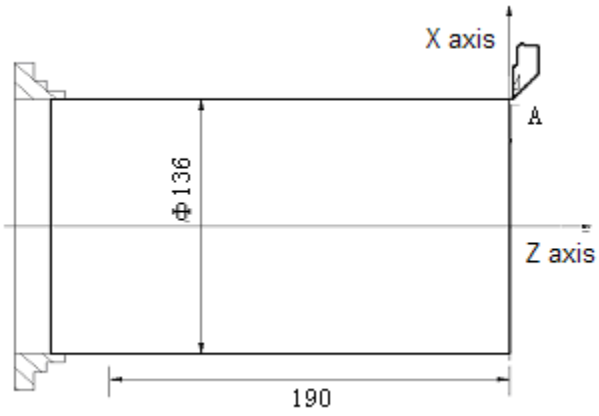


Fig.10-8

(8) The system switches into the tool offset page, and the cursor is moved toward #002 offset, the software key MEASURE is pressed to enter the measuring input page, X135 is input on the input page MEASURE, and OK software key is pressed, Z0 is input with the same method;

(9) Move the tool into the safe position, and execute the 3rd tool by pressing the tool change key in Manual mode;

(10) Start the spindle and move the tool into the tool setting position, which is shown as Fig. 10-9, like point A:

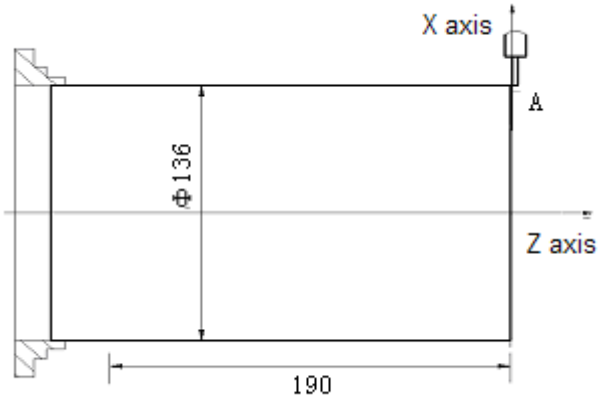


Fig.10-9

(11) Switch the tool to the tool offset page and move the cursor into #003 offset, input X135 and Z0, and the input operation is same as the 8th step;

(12) Move the tool into the safe position, and execute the 4th tool by pressing the tool change key in Manual mode;

(13)Move the tool into the tool setting position, which is shown as Fig. 10-10, point A:

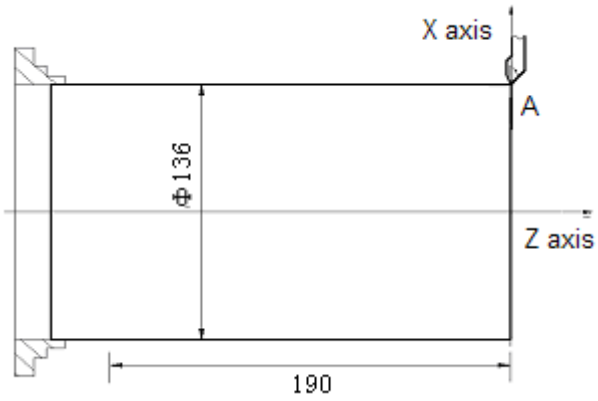


Fig.10-10

(14) Switch the tool to the tool offset page and move the cursor into #004 offset, input X135 and Z0, and the input operation is same as the 8th step;

(15)After the tool setting is completed, the tool is moved into the safe position;

(16) In Auto mode, press 

for automatic machining;

(17) Measure the workpiece dimension, if it is different with the actual part dimension, the tool wearing value can be rewritten until the part dimension is within the tolerance limit.

APPENDIX

Appendix 1 Parameters

This chapter mainly introduces CNC state and Value parameters through setting different parameters to realize the different requirements of function. The parameter Value mainly includes the following six types:

Value Types		Range
(1) Bit	1001 INM 0 0 0 0 0 0 0 0	8 digits,0 or 1
(2) Bit axis	1006 ZMIx DIAx ROSx ROTx X 0 0 0 0 1 0 0 0 Z 0 0 0 0 0 0 0 0	
(3)Bit spindle type		
(4) Word	0123 BPS 115200	The setting value range is different according to the variable parameters, refer to the parameter
(5) Word axis	1020 CAN X 88 Z 90	
(6) Word spindle type	3720 CNT S1 1024 S2 1024	

For the word axis parameter (3) and (4), the exact Value range is determined by specified parameters.

Each parameter should include the following information:

『Modification authority』: System authority (1st level), Machine authority (2nd level), Equipment management authority (3rd level), Operation authority (4th level), Limited authority (5th level)

『Parameter Type』: Bit, bit axis, word, word axis, Bit axis type, Bit spindle type, Word spindle type

『Validate method』: Become valid immediately or after power-on

『Value Range』: In interval, by enumerating or special judgement)

『Default Setting』: 8 digits in binary system, or 32-digit integral value

Note 1: The 『Data Range』of bit type parameters is 0 or 1.

Note 2: When 『Validate method』 is not stated, the parameter will become valid immediately.

Note 3: When 『Parameter Type』 is not stated, the parameter is of bit type or word type.

Appendix 1.1 Parameter for "Setting"

	#7	#6	#5	#4	#3	#2	#1	#0
0000			**SEQ**			**INI**		

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#2 INI Input unit

0: Metric system

1: Inch system

#5 SEQ whether insert the sequence number automatically

0: No

1: Yes

Note: In EDIT or MDI mode, sequence number can be inserted automatically. The incremental value of sequence number is set in parameter of NO.3216.

Appendix 1.2 Parameters of the Interfaces of Input and Output

0123	**Serial port baud rate (BPS)**

『Modification authority』:Equipment management

『Value Range』: 4800, 9600, 19200, 38400, 57600, 115200

『Defualt Setting』: 115200

	#7	#6	#5	#4	#3	#2	#1	#0
0138		**OWN**						

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#6 OWN When NC Value or the programs are input or output,:

0: whether the covered file information is displayed

1: covered file, is not displayed

	#7	#6	#5	#4	#3	#2	#1	#0
0930						MODBUS	NDSVR	RMEN

『Modification authority』:Machine

『Validate method』: After power-on

『Default Setting』: 0000 0000

#0 RMEN Whether use the remote monitoring function

0：YES

1：NO

#1 NDSVR Whether open the Ethernet data communication service

0：Close

1：Open

#2 MODBUS Whether open Modbus communication

0：Close

1：Open

Appendix 1.3 Parameters of Axis Control/Setting Unit

	#7	#6	#5	#4	#3	#2	#1	#0
1001								INM

『Modification authority』: Machine

『Validate method』: After power-on

『Default Setting』: 0000 0000

#0 INM The least movement increment of linear axis is in:

0: Metric system (metric machine)

1: Inch system (inch machine)

	#7	#6	#5	#4	#3	#2	#1	#0
1002					AZR		DLZ	

『Modification authority』: Machine

『Default Setting』: 0000 0000

#1 DLZ Whether reference setting without dog is valid:

0: Invalid

1: Valid (for all axes)

Note: When DLZ is 0, parameter 1005#1 (DLZx) can set valid/invalid for each axis.

#3 AZR G28 command when the reference point is not set:

0: Reference point return with deceleration dog

1: alarm occurs

Note: The function of reference point return without dog (when parameter 1002#1 (DLZ) is 1 or parameter 1005#1 (DLZx) is 1) is not related to the setting of AZR.

	#7	#6	#5	#4	#3	#2	#1	#0
1004		**RPR**					**ISC**	

『Modification authority』: Machine

『Validate method』: After power-on

『Default Setting』: 0000 0000

#1 ISC Set the least input increment and least command increment

0：0.001mm, 0.001deg or 0.0001inch（IS-B）

1：0.0001mm, 0.0001deg or 0.00001inch（IS-C）

#6 RPR Set the least command increment of the rotation axis and the multiplication of ISC parameter

#7 IPC Set the least command increment of the rotation axis and the multiplication of ISC parameter

00：×1 times

01：×10 times

10：×100 times

	#7	#6	#5	#4	#3	#2	#1	#0
1005					**HJZx**		**DLZx**	**ZRNx**

『Modification authority』: Machine

『Parameter Type』: Bit axis

『Default Setting』: 0000 1000

#0 ZRNx Whether the system alarms if the other traverse commands are specified except G28 before setting the reference point in auto running (AUTO, DNC or MDI).

0: Alarm

1: Not alarm

#1 DLZx Whether setting the reference point free of the link stopper is valid.

0: Invalid

1: Valid

Note: Parameter DLZ (No.1002#1) is valid when it is "0". When DLZ (No.1002#1) is "1", there is no connection with the parameter, and setting the reference point free of the link stopper is valid for all axes.

#3 HJZx After the reference point is set, manually return to the reference point.

0: Use the deceleration link stopper to return to the reference point

1: No connection with the deceleration link stopper, rapidly position in the reference point.

	#7	#6	#5	#4	#3	#2	#1	#0
1006			**ZMIx**		**DIAx**		**ROSx**	**ROTx**

『Modification authority』: Machine

『Validate method』: After power-on

『Parameter Type』: Bit axis

『Default Setting』: 0000 0000

#0, #1 ROTx, ROSx set linear axis or rotary axis

ROSx	ROTx	Content
0	0	Linear axis Metric/inch conversion All coordinate values are of the linear axis type. The stored pitch error compensation is of the linear axis type.
0	1	Rotary axis (type A) No metric/inch conversion Machine tool coordinate value circularly displays based upon the value of the parameter 1260. The relative coordinate value is relevant to the parameters 1008#2, 1008#0, and the absolute coordinate is related with the 1008#0 The stored pitch error compensation is the rotary axis type. Automatically return to the reference point at the direction of the reference point return (G28 and G30), the traverse amount can not exceed one-turn.
1	0	Invalid setting
1	1	Rotary axis (type B) No metric/inch conversion Machine tool coordinate value, relative coordinate value (it is relevant to the parameter 1008#2) and absolute value are linear axes (It can not be circularly display by parameter 1260) The stored pitch error compensation is of the linear axis type.

#3 DIAx sets the traverse amount of each axis

0: specified by the radius

1: specified by the diameter

#5 ZMIx sets the direction of each axis reference point return

0: positive direction

1: negative direction

	#7	#6	#5	#4	#3	#2	#1	#0
1007	**RZDx**							

『Modification authority』:Machine

『Value Range』: Bit axis

『Default Setting』: 0000 0000

#7 RZDx Rotation axis (type A) is in the state of reference point establishment, whether it is the approximate selection direction when reference point returns.

0: Disabled

1: Enabled

	#7	#6	#5	#4	#3	#2	#1	#0
1008						**RRLx**	**RABx**	**ROAx**

『Modification authority』: Machine

『Way of Validating』: After power-on

『Parameter Type』: Bit axis

『Default Setting』: 0000 0000

#0 ROAx sets whether the cycle display function of the rotary axis valid.

0: Invalid

1: Valid

Note: ROAx is just valid for the rotary axis and parameter ROTx (No.1006#0) must be 1.

#1 RABx sets the rotation direction of the axis during the absolute command.

0: Rotation direction close to the target

1: Direction specified by the command value coder

Note: RABx is valid only when parameter ROAx is 1.

#2 RRLx Relative coordinate

0: Not cycle as the movement amount of each turn

1: Cycle as the movement amount of each turn

Note 1: RRLx is valid only when ROAx is 1.

1010	**Quantity of CNC controlled axes (CCA)**

『Modification authority』: Machine

『Validate method』: After power-on

『Value Range』: 0～total number

Set the total number of axes which is directly controlled by CNC, the other can be controlled by PLC.

Note: The overall controllable axes numbers are determined by parameter No.8130, and its setting value

of this parameter can not be more than the one of the No.8130.

	#7	#6	#5	#4	#3	#2	#1	#0
1015	**DWT**	**WIC**						

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#6 WIC The offset measured value of the work piece origin is directly input

0: Only valid for the selected work piece coordinate system

1: Valid for all coordinate systems

#7 DWT When the pause time is specified by P, the Value units are

0: IS-B is 1ms, IS-C is 0.1ms.

1: 1 ms

1020	Programming name of each axis (CAN)

『Modification authority』: Machine

『Parameter Type』: Word axis

『Value Range』: 88（X），89（Y），90（Z），65（A），66（B），67（C），85（U），86（V），87（W）

Set the axial name of each controlled axis.

Note: The same axes names can not be set; U, V and W axes are only enabled in the G code of B set.

1022	The property of each axis in the basic coordinate system

『Modification authority』: Machine

『Validate method』: After power-on

『Parameter Type』: Word axis

『Value Range』: 0～7

To ensure the planes of the arc interpolation, the tool offset and the tool nose radius, etc.

G17: X－Y plane

G18: Z－X plane

G19: Y－Z plane

There are four controllable axes: 1 – X basis axis and parallel axis; 2 – Y basis axis and parallel axis; 3 – Z basis axis and parallel axis; 4 – Rotation axis. Only one axis of the basic three axes can be set: X, Y and Z; the parallel axes can be set as two more axes (which is paralleled with the basic axis).

Setting value	Meaning

0	They are neither basic three axes nor the parallel axes,
1	X axis of the basic three axes
2	Y axis of the basic three axes
3	Z axis of the basic three axes
5	Parallel axis of X axis
6	Parallel axis of Y axis
7	Parallel axis of Z axis

1023	Servo axis number of each axis (NSA)

『Modification authority』: Machine

『Validate method』: After power-on

『Parameter Type』: Word axis

『Value Range』: 0～99

Set the logic ID number (0～99;0 means that there is no slave station) of the feed servo slave station, its setting value should be corresponding with the one of the servo driver.

Appendix 1.4 Parameter of the Coordinate System

	#7	#6	#5	#4	#3	#2	#1	#0
201	WZR		EWZ	RWO	ZCR	ZCL		

『Modification authority』: Equipment management authority

『Default Setting』: 0000 0000

#2 ZCL After manually return to reference point, the part coordinate system

0: Not cancel

1: Cancel

#3 ZCR After the manual reference point return is completed, the workpiece coordinate system offset value set by G50:

0：Not cancel

1：Cancel

#4 RWO The workpiece coordinate system offset value set by G50 when the coordinate memories after the power is turned on.

0：Clear

1：Restore the memory value from the previous power-off

#5 EWZ The workpiece coordinate system when the power-on coordinate memories.

0：Do not return to G54

1：Return to G54

#7 WZR Work piece coordinate system during resetting

0: Not return to G54

1: Return to G54

	#7	#6	#5	#4	#3	#2	#1	#0
1202					**RLC**	**G50**	**EWS**	**EWD**

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#0 EWD The movement direction of the coordinate system caused by the external work piece origin offset amount

0: It is same as the direction specified by the external work piece origin offset amount.

1: It is opposite to the direction specified by the external work piece origin offset amount.

#1 EWS The work piece coordinate system movement amount and the external work piece zero point offset amount

0: Saved in each memorizer

1: Saved in one memorizer (the work piece coordinate system movement amount is same as the external work piece zero point offset amount

#2 G50 When G50 is commanded and the coordinate system is set,

0: Not alarm, but execute G50

1: P/S alarms (No.010), not execute G50

#3 RLC After resetting, the part coordinate system

0: Not cancel

1: Cancel

	#7	#6	#5	#4	#3	#2	#1	#0
1205								**MCE**

『Modification authority』: Equipment management

『Default Setting』：0000 0000

#0 MCE Whether the coordinate system is memorized with power-on when adapting with the incremental encoder.

0：Do not memory

1：Memory

1206	**The allowable value of the machine coordinate system with the absolute encoder after power on (MER)**

『Modification authority』: Equipment management

『Value Range』: 0～9999

『Parameter Type』: Word axis

『Default Setting』: 1000

It is for detecting the offset when the machine coordinate system is set at power on; if it is out of the range, the alarm occurs. The offset isn't detected when it is 0.

1220	**The origin offset amount of each axis external work piece coordinate system (EWO)**

『Modification authority』: Equipment management

『Parameter Type』: Word axis

『Value Range』: -9999 9999～9999 9999

This is one parameter to set the origin location of the work piece coordinate system (G54~G59). The parameter is the valid common offset amount for all work piece coordinate system.

Setting unit	IS-B	IS-C	Unit
Linear axis (input in metric system)	0.001	0.0001	mm
Linear axis (input in inch system)	0.0001	0.00001	inch
Rotary axis	0.001	0.0001	deg

1221	**Origin offset amount of each axis in G54 workpiece coordinate system (WO1)**

1222	**Origin offset amount of each axis in G55 workpiece coordinate system (WO2)**

1223	**Origin offset amount of each axis in G56 workpiece coordinate system (WO3)**

1224	**Origin offset amount of each axis in G57 workpiece coordinate system (WO4)**

1225	**Origin offset amount of each axis in G58 workpiece coordinate system (WO5)**
1226	**Origin offset amount of each axis in G59 workpiece coordinate system (WO6)**

『Modification authority』: Equipment management

『Parameter Type』: Word axis

『Value Range』: -99 999 999～+99 999 999

This is one parameter to set the origin location of the work piece coordinate system (G54~G59). The parameter is the valid common offset amount for all the work piece coordinate system.

SETTING UNIT	IS-B	IS-C	UNIT
Linear axis (input in metric system)	0.001	0.0001	mm
Linear axis (input in inch system)	0.0001	0.00001	inch
Rotary axis	0.001	0.0001	deg

1240	**Each axis machine coordinate value of the 1st reference point (RF1)**
1241	**Each axis machine coordinate value of the 2nd reference point (RF2)**
1242	**Each axis machine coordinate value of the 3rd reference point (RF3)**
1243	**Each axis machine coordinate value of the 4th reference point (RF4)**

『Modification authority』: Equipment management

『Way of Validating』: 1240 valid after power on; 1241～1243 valid immediately.

『Parameter Type』: Word axis

『Value Range』: -99 999 999～+99 999 999

Set the coordinate values from the 1st to the 4th reference points in the mechanical coordinate system.

SETTING UNITS	IS-B	IS-C	UNITS
Machine in metric system	0.001	0.0001	mm
Machine in inch system	0.0001	0.00001	inch
Rotary axis	0.001	0.0001	deg

1260	Each turn movement amount of each axis in rotary axis（PRA）

『Modification authority』: Equipment management

『Validate method』: After power-on

『Parameter Type』: Word axis

『Value Range』: 1000～9 999 999

Set the movement amount of each turn in rotary axis.

Appendix 1.5 Parameter of the Stroke Detection

Setting unit of stroke parameter Nos.1320~1327 is shown in the following table:

Setting unit	IS-B	IS-C	Unit
Metric machine	0.001	0.0001	mm
Inch machine	0.0001	0.00001	inch
Rotary axis	0.001	0.0001	deg

	#7	#6	#5	#4	#3	#2	#1	#0
1300	**BFA**	**LZR**	**RL3**			**LMS**		**OUT**

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#0 OUT The restricted area of the stroke detection 2 in memory type is set by parameters (No.1322 or No.1323).

0: Internal area

1: External area

#2 LMS Whether the switching signal EXLM of the stroke detection in memory type is valid

0: Invalid

1: Valid

Note: Stroke detection 1 in memory type possesses the parameter of the restricted area set by two groups, signals are switched through the stroke limit in memory type and the set restricted area is selected. **(1) Restricted area I: Parameter No.1320 or No.1321** **(2) Restricted area II: Parameter No.1326 or No.1327**

#5 RL3 Whether it is valid that the stroke detection 3 releases signal RLS0T3

0: Invalid

1: Valid

#6 LZR After power on before manual reference point return whether detect the stroke 1 in the memory type

0: Detect

1: Not detect

Note: There isn't any connection with the setting when the absolute position encoder is being using, the power is on and the reference point is set. After power on, the stroke is directly detected in memory type.

#7 BFA When the command of overrun memory is sent

0: Alarm after overrun

1: Alarm before overrun

	#7	#6	#5	#4	#3	#2	#1	#0
1301	**PLC**	**OTS**						

『Modification authority』: Equipment management

『Parameter Type』: Bit axis

『Default Setting』: 0000 0000

#6 OTS Whether output the signal in the overtravel alarm to PLC when the stored stroke detection alarm occurs.

0：Do not output

1：Output

#7 PLC Whether check the stroke before moving

0: No

1: Yes

	#7	#6	#5	#4	#3	#2	#1	#0
1310							**OT3x**	**OT2x**

『Modification authority』: Equipment management

『Parameter Type』: Bit axis

『Default Setting』: 0000 0000

#0 OT2X Whether each axis detects the stroke 2 in memory type

0: Not detect

1: Detect

#1 OT3X Whether detect the stroke 3 in memory type in each axis

0: Not detect

1: Detect

1320	**Coordinate value in positive direction boundary of each axis stroke detection 1 in memory type（PC1）**

1321	**Coordinate value in negative direction boundary of each axis stroke detection 1 in memory type（NC1）**

『Modification authority』: Equipment management

『Parameter Type』: Word axis

『Default Setting』: No.1320 is 99 999 999, No.1321 is －99 999 999

『Value Range』:－99 999 999～99 999 999

Respectively set the coordinate values of boundaries in positive/negative directions in the mechanical coordinate system in stroke detection 1 along each axis in memory type. Set the outside of boundary as the restricted area to tools.

1. The axes specified by diameter are set by diameter value.

2. When (parameter No.1320) < (parameter No.1321) and the limit is infinite, it can not detect the stroke 1 in memory type. (The stroke limit switching signal in memory type is invalid.) If the absolute command is specified, the coordinate value may overflow; the normal movement can not be executed.

1322	**Coordinate value in positive direction boundary of each axis stroke detection 2 in memory type（PC2）**

1323	Coordinate value in negative direction boundary of each axis stroke detection 2 in memory type（NC2）

『Modification authority』: Equipment management

『Parameter Type』: Word axis

『Default Setting』: 0

『Value Range』: －99 999 999～99 999 999

Respectively set the coordinate values of boundaries in positive and negative directions in the mechanical coordinate system in stroke detection 2 along each axis in memory type. The outside or inside of boundary is the restricted area, which is set by parameter OUT (No.1300#0).

Note: The axis specified by diameter must be set by the diameter value.

1324	Coordinate value in positive direction boundary of each axis stroke detection 3 in memory type（PC3）

1325	Coordinate value in negative direction boundary of each axis stroke detection 3 in memory type（NC3）

『Modification authority』: Equipment management

『Parameter Type』: Word axis

『Default Setting』:0

『Value Range』: －99 999 999～99 999 999

Respectively set the coordinate values of boundaries in positive and negative directions in the mechanical coordinate system in stroke detection 3 along each axis in memory type. Set inside of the boundary as the restricted area to tools.

Note: The axis specified by the diameter must be set by the diameter value.

1326	Coordinate valueⅡ in positive direction boundary of each axis stroke detection 1 in memory type（PC12）

1327	Coordinate valueⅡ in negative direction boundary of each axis stroke detection 1 in memory type（PC12）

『Modification authority』: Equipment management

『Parameter Type』: Word axis

『Default Setting』: 0

『Value Range』: －99 999 999～99 999 999

Respectively set the positive and negative boundary coordinate values in stroke detection 1 along each axis in memory type in the machine coordinate system. Set outside of the boundary as the restricted area. When parameter LMS (No.1300#2) is “1”, and the stroke limit switching signal EXLM (G7.6) in memory type is “1”, the restricted area is valid, but it is invalid if it is set by No.1320 and 1321.

1. The axes programmed by the diameter must be set by the diameter value.

2. The parameter is invalid when parameter LMS (No.1320#2) is “0”, or the stroke limit switching signal EXLM (G7.6) in the memory type is “0”. Then, the restricted area set by parameter No.1320 or No. 1321 is valid.

Appendix 1.6 Parameter of the Feedrate

	#7	#6	#5	#4	#3	#2	#1	#0
1401		**RDR**	**TDR**	**RF0**			**LRP**	**RPD**

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#0 RPD Manually rapid run from power on to the reference point return

0: Invalid (JOG speed)

1: Valid

#1 LRP Positioning (G00):

0：Non-linear interpolation positioning

1：Linear interpolation positioning

#4 RF0 When the cutting feedrate override is 0% during rapid traverse

0: tool does not stop moving

1: tool stops moving

#5 TDR During thread cutting or tapping, dry run is:

0: Valid

1: Invalid

#6 RDR To rapid traverse command, dry run is:

0: Invalid

1: valid

	#7	#6	#5	#4	#3	#2	#1	#0
1402						**JOV**		

『Modification authority』: Equipment management

Appendix

『Default Setting』: 0000 0000

#2 JOV JOG override

0: Valid

1: Invalid (fixed as 100%)

	#7	#6	#5	#4	#3	#2	#1	#0
1403	**RTV**		**HTG**					**MIF**

『Modification authority』: Equipment management authority

『Default Setting』: 0000 0000

#0 MIF The minimum unit of F command (the cutting feedrate) of feeding/min

0：1mm/min (input in metric system) or 0.01inch/min (input in inch system)

1：0.001mm/min (input in metric system)or 0.00001inch/min (input in inch system)

#5 HTG The speed command of the spiral interpolation is:

0: Specified by the linear speed of the arc

1: Specified by the linear speed with the linear axis

#7 RTV During thread cutting cycle, the override of the tool run-out is

0: Valid

1: Invalid

	#7	#6	#5	#4	#3	#2	#1	#0
1404						**F8A**	**DLF**	

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#1 DLF After setting the reference point, manually return to the reference point

0: Move to the reference point (No.1420) at the rapid feedrate

1: Move to the reference point (No.1424) at the manual rapid feedrate

#2 F8A F command range feed/min

0: Set according to parameter MIF (No.1403#0)

1:

SETTING UNITS	UNIT	IS-B	IS-C
Input in metric system	mm/min	1～60000.999	1～24000.999
Input in inch system	inch/min	0.01～2400	0.01～960
Rotary axis	deg/min	1～60000	1～24000

1410	**Dry run speed (DRR)**

『Modification authority』: Equipment management

『Value Range』:

SETTING UNITS	VALUE UNITS	VALID RANGE		DEFAULT SETTING
		IS-B	IS-C	
Machine in metric system	1mm/min	6～15000	6～12000	1000
Machine in inch system	0.1inch/min	6～6000	6～4800	

Set the speed during dry run.

1411	Feedrate in auto mode after power on（IFV）

『Parameter Type』: Word type

『Value Range』: 6～12000

『Default Setting』: 100

SETTING UNITS		VALUE UNITS
Machine in metric system	G98	1 mm/min
	G99	0.001 mm/rev
Machine in inch system	G98	0.1 inch/min
	G99	0.0001 inch/rev

It doesn't require changing the cutting speed in the machine during the processing. And the cutting feedrate can be set by the parameter, and then the cutting feedrate is not required to be set in the program. But the actual feedrate is limited by parameter NO.1422 which set the maximum cutting feedrate for all axes.

1420	Each axis rapid movement speed (RTT)

『Modification authority』: Machine

『Parameter Type』: Word axis

『Value Range』:

SETTING UNITS	VALUE UNITS	VALID RANGE		DEFAULT SETTING
		IS-B	IS-C	
Machine in metric system	1mm/min	30～100000	6～60000	8000

Machine in inch system	0.1inch/min	30～48000	6～24000	
Rotary axis	1 deg/min	30～100000	6～60000	

Set the rapid movement speed of each axis when the rapid movement override is 100%.

1421	F0 speed of each axis rapid override（F0R）

『Modification authority』: Equipment management authority

『Parameter Type』: Word axis

『Value Range』:

SETTING UNITS	VALUE UNITS	VALID RANGE		DEFAULT SETTING
		IS-B	IS-C	
Machine in metric system	1 mm/min	30～15000	30～12000	400
Machine in inch system	0.1 inch/min	30～12000	30～6000	
Rotary axis	1 deg/min	30～15000	30～12000	

Set the speed when the rapid movement override of each axis is F0.

1422	Maximum cutting feedrate of all axes (MFR)

『Modification authority』: Machine

『Value Range』:

SETTING UNITS	VALUE UNITS	VALID RANGE		DEFAULT SETTING
		IS-B	IS-C	
Machine in metric system	1mm/min	6～100000	6～60000	8000
Machine in inch system	0.1inch/min	6～48000	6～24000	

Set the maximum cutting feedrate for all axes.

1423	JOG feedrate of each axis (JFR)

『Modification authority』: Equipment management

『Parameter Type』: Word axis

『Value Range』:

SETTING UNITS	VALUE UNITS	VALID RANGE		DEFAULT SETTING
		IS-B	IS-C	
Machine in metric system	1mm/min	6～60000		1000

Machine in inch system	0.1inch/min		
Rotary axis	1 deg/min		

Set the feedrate of each axis during continually manual feeding (JOG feeding), the actual feedrate is limited by parameter NO.1422 (the maximum cutting feedrate of all axes).

1424	Manual rapid speed of each axis (MRR)

『Modification authority』: Equipment management authority

『Parameter Type』: Word axis

『Value Range』:

SETTING UNITS	VALUE UNIT	VALID RANGE		DEFAULT SETTING
		IS-B	IS-C	
Metric machine	1 mm/min	30～100000	30～60000	8000
Inch machine	0.1 inch/min	30～48000	30～24000	
Rotary axis	1 deg/min	30～100000	30～60000	

Set the speed of each axis manual rapid movement when rapid movement override is 100%.

Set the maximum speed of MPG feeding.

Note: If it is set as 0, use the setting value of parameter 1420.

1425	FL speed of each axis reference point return (FLR)

『Modification authority』: Equipment management

『Parameter Type』: Word axis

『Value Range』:

SETTING UNITS	VALUE UNITS	VALID RANGE		DEFAULT SETTING
		IS-B	IS-C	
Machine in metric system	1 mm/min	6～15000	6～12000	200
Machine in inch system	0.1 inch/min	6～12000	6～6000	
Rotary axis	1 deg/min	6～15000	6～12000	

After deceleration is performed, set the speed (FL speed) of each axis during the reference point return.

1428	Reference point return speed along each axis (RPF)

『Modification authority』: Equipment management

『Parameter Type』: Word axis

『Value Range』:

SETTING UNITS	VALUE UNITS	VALID RANGE	DEFAULT SETTING
Machine in metric system	1 mm/min	0，6～60000	5000
Machine in inch system	0.1 inch/min		
Rotary axis	1 deg/min		

Set the situation of the reference point return used the deceleration block, alternatively, the rapid traverse rate based upon the reference point return regardless of the state of reference point. When the parameter value sets to 0, parameter №1421 is enabled.

1434	The Max. feedrate of the Manual MPG along each axis (HMF)

『Modification authority』: Equipment management

『Parameter Type』: Word axis

『Value Range』:

SETTING UNITS	VALUE UNITS	VALID RANGE	DEFAULT SETTING
Machine in metric system	1 mm/min	0，6～60000	5000
Machine in inch system	0.1 inch/min		
Rotary axis	1 deg/min		

Set the Max. feedrate of the manual MPG of each axis. When its setting is 0, the setting value of parameter №1424 is enabled.

1466	The retracting feedrate during the thread cutting (FRT)

『Modification authority』: Equipment management

『Parameter Type』: Word axis

『Value Range』:

SETTING UNITS	VALUE UNITS	VALID RANGE		DEFAULT SETTING
Machine in metric system		IS-B	IS-C	
Machine in inch system	1 mm/min	6～100000	6～60000	8000
SETTING UNITS	0.1 inch/min	6～48000	6～24000	

Set the feedrate of end-retraction operation of the thread cutting machining. When this

parameter sets to “0”, that is, the speed of long axis is performed the end-retraction operation.

Appendix 1.7 Parameter of Control of Acceleration and Deceleration

	#7	#6	#5	#4	#3	#2	#1	#0
1601				**RTO**				

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#4 RTO During rapid running, the block is

0: No overlapping

1: Overlapping

	#7	#6	#5	#4	#3	#2	#1	#0
1610			**THLX**	**JGLx**				

『Modification authority』: Equipment management

『Parameter Type』: Word axis

『Default Setting』: 0000 0000

#4 JGLx The acceleration/deceleration for the manual feed

0：Exponential acceleration/deceleration

1：Linear acceleration/deceleration after interpolation

#5 THLX The acceleration/deceleration of the end-retraction operation in the thread cutting machining:

0：Exponential acceleration/deceleration

1：Linear acceleration/deceleration

1620	**Time constant T of linear acceleration and deceleration of each axis rapid movement (TT1)**

『Modification authority』: Equipment management

『Parameter Type』: Word axis

『Value Range』: 0～4000 ms

『Default Setting』: 100

Set the time constant of acceleration and deceleration during rapid movement.

1622	**Time constant of acceleration and deceleration during cutting and feeding after each axis interpolation (ATC)**

『Modification authority』: Equipment management

『Parameter Type』: Word axis

『Value Range』: 0～4000 ms

『Default Setting』: 100

Set the acceleration and deceleration of each axis cutting and feeding in index type, or the time constant of acceleration and deceleration in linear type after interpolation.

And the detailed type is set by parameter CTLx (NO.1610#0). If CTLx sets the acceleration and deceleration in linear type after linear interpolation, the maximum time constant of acceleration and deceleration is limited in 512ms and even it exceeds 512ms, it is still dealt as 512ms.

Note: Except the special usage of the parameter, all axes must be set as the same time constant. If the different time constants are set, the correct linear or circular can't be shaped.

1624	Time constant of acceleration and deceleration of each axis JOG feeding after interpolation (JET)

『Modification authority』: Machine

『Parameter Type』: Word axis

『Value Range』: 0～4000ms

『Default Setting』: 100

Set the acceleration and deceleration in index type of each axis JOG feeding, and the time constant of acceleration and deceleration in linear type after interpolation.

The detailed type is set by parameter JGLx (NO.1610#4). If JGLx sets the acceleration and deceleration in linear type after interpolation, the maximum time constant of acceleration and deceleration is limited in 512ms and even it exceeds 512ms, it is dealt as 512ms.

1625	FL speed of acceleration and deceleration in index type during each axis JOG feeding (FLJ)

『Modification authority』: Equipment management authority

『Parameter Type』: Word axis

『Value Range』:

SETTING UNITS	VALUE UNITS	VALID RANGE		DEFAULT SETTING
		IS-B	IS-C	
Metric machine	1 mm/min	0，6～15000	0，6～12000	30
Inch machine	0.1 inch/min	0，6～12000	0，6～6000	30
Rotary axis	1 deg/min	0，6～15000	0，6～12000	30

Set the low limit speed (FL speed) of acceleration and deceleration in index type during each axis JOG feeding.

1626	Time constant of acceleration and deceleration during each axis thread cutting cycle (TET)

『Modification authority』: Equipment management

『Parameter Type』: Word axis

『Value Range』: 0～4000ms

『Default Setting』: 100

Set the time constant of acceleration and deceleration in linear and index types during each axis thread cutting cycle.

1627	FL speed of acceleration and deceleration in index type during each axis thread cutting cycle (FLT)

『Modification authority』: Equipment management

『Parameter Type』: Word axis

『Value Range』:

SETTING UNITS	VALUE UNITS	VALID RANGE		DEFAULT SETTING
		IS-B	IS-C	
Machine in metric system	1 mm/min	0，6～15000	0，6～12000	30
Machine in inch system	0.1 inch/min	0，6～12000	0，6～6000	30

Set low limit speed (FL speed) of acceleration and deceleration in index type during each axis thread cutting cycle.

1628	The acceleration/deceleration time constant of the end-retraction motion in the thread cutting cycle along each axis (TST)

『Modification authority』: Equipment management

『Parameter Type』: Word axis

『Value Range』: 0～4000ms

『Default Setting』: 0

Set the acceleration/deceleration time constant of end-retraction short axis when the thread cutting cycle of each axis is performed; when this parameter setting value is “0”, use the No.1626

parameter value (0～4000ms).

Appendix 1.8 Parameter of Servo and Backlash Compensation

	#7	#6	#5	#4	#3	#2	#1	#0
1800	BDEC	BD8						

『Modification authority』: Machine

『Default Setting』: 1000 0000

#6 BD8: Impulse output frequency of the backlash compensation

0: Compensate at the frequency set by parameter #1853

1: Compensate at 1/8 of frequency set by parameter #1853

#7 BDEC:Backlash compensation mode

0: Fixed pulse frequency output, which is set by parameters #1853 and #1800.6.

1: Pulse frequency output based on the acceleration and deceleration characteristics.

	#7	#6	#5	#4	#3	#2	#1	#0
1811						POD		

『Modification authority』: Machine

『Validate method』: After power-on

『Parameter Type』: Bit axis

『Default Setting』: 0000 0000

#2 POD Selecting output directions of each axis pulse

0: Not inversed

1: Inversed

	#7	#6	#5	#4	#3	#2	#1	#0
1815			APCx	APZx				

『Modification authority』: Machine

『Validate method』: After power-on

『Parameter Type』: Bit axis

『Default Setting』: 0000 0000

#4 APZx The mechanical position and the absolute position detector position during using the absolute position detector

0: Not consistent

1: Consistent

Note: When use the absolute position detector, during the initial setting or after changing the absolute position encoder, the parameter must be set as 0, and connect power supply, again after power off and manually return to the reference point. Therefore, the mechanical position consists with that of the position encoder, and the parameter will be auto set as 1.

#5 APCx Position encoder

0: Not use the absolute position detector

1: Use the absolute position detector (the absolute pulse encoder)

1816	Each axis detection multiply ratio (DMR)

『Modification authority』: Machine

『Parameter Type』: Word axis

『Value Range』: 1~32767

『Default Setting』: 2

The detection multiply ratio (DMR) of each axis is set

1820	Command multiply ratio of each axis（CMR）

『Modification authority』: Machine

『Parameter Type』: Word axis

『Value Range』: 1~32767

『Default Setting』: 2

Gear ratio output by each axis＝CMR/ DMR

Detection unit=minimum movement unit/ CMR

The relations between the setting units and the minimum movement units:

		IS－B		IS－C	
	Input	Least input increment	Least command increment	Least input increment	Least command increment
Metric machine	Metric	0.001mm（Diameter）	0.0005mm	0.0001mm（Diameter）	0.00005mm
		0.001mm（Radius）	0.001mm	0.0001mm（Radius）	0.0001mm
	Inch	0.0001 inch（Diameter）	0.0005mm	0.00001 inch（Diameter）	0.00005mm
		0.0001 inch（Radius）	0.001mm	0.00001 inch（Radius）	0.0001mm
Inch machine	Metric	0.001mm（Diameter）	0.00005 inch	0.0001mm（Diameter）	0.000005 inch
		0.001mm（Radius）	0.0001 inch	0.0001mm（Radius）	0.00001 inch
	Inch	0.0001 inch（Diameter）	0.00005 inch	0.00001 inch（Diameter）	0.000005 inch
		0.0001 inch（Radius）	0.0001 inch	0.00001 inch（Radius）	0.00001 inch
Rotary		0.001deg	0.001deg	0.0001deg	0.0001deg

axis				

1851	Backlash compensation value of each axis (BCV)

『Modification authority』: Machine

『Parameter Type』: Word axis

『Value Range』: -9999～+9999 (Detection unit)

『Default Setting』: 0

Set the backlash compensation value of each axis.

After connecting power supply, it compensates the backlash at the first time when the machine moves in the direction opposite with that of the reference point return.

Detection units are related with parameter No.1820 (command multiply ratio CMR) and the minimum movement units, about the relations between the setting units and the minimum movement units, refer to parameter No.1820 introduction.

1853	The setting value of reverse interval compensation pulse frequency

『Modification authority』: Machine

『Parameter Type』: Word

『Value Range』: 1~32

『Default Setting』: 12

The setting value of reverse interval compensation pulse frequency (1~32)

2071	Each axis backlash acceleration and deceleration valid time constant (BAT)

『Modification authority』: Machine

『Parameter Type』: Word axis

『Value Range』: 0～100 ms

『Default Setting』: 40

Set each axis backlash acceleration and deceleration valid time constant.

Appendix 1.9 Parameter of Input/Output

	#7	#6	#5	#4	#3	#2	#1	#0
3001						RWM		

『Modification authority』: Machine

『Default Setting』: 0000 0000

#2 RWM Whether output the rewinding signal in the program back within the program memory (RWD)

0：Do not output

1：Output

	#7	#6	#5	#4	#3	#2	#1	#0
3003	**ESP**					**ITX**		**ITL**

『Modification authority』: Machine

『Default Setting』: 1000 0000

#0 ITL To interlock the signal of the overall axes

0：Disabled

1：Enabled

#2 ITX To interlock the signal of each axis

0：Disabled

1：Enabled

#7 ESP External emergency stop alarm input signal (X0.5)

0: When the signal is 0 (low level), emergency stop alarms

1: When the signal is 1 (high level), emergency stop alarms

	#7	#6	#5	#4	#3	#2	#1	#0
3004			**OTH**					**BSL**

『Modification authority』: Machine

『Default Setting』: 0010 0000

#0 BSL Block starts the interlocking signal and the cutting block starts the interlocking signal

0：Disabled

1：Enabled

#5 OTH Overtravel limit signal

0: Check

1: Not check

Note: After the overtravel alarm occurs, this parameter is altered to 1 (without detection), and the alarm will not be cleared pressing the resetting again; it is necessary to move inside the stroke by hand; and then set this parameter to 0, the alarm is eliminated accordingly.

#7	#6	#5	#4	#3	#2	#1	#0

Appendix

3006						EPS	EPN	GDC

『Modification authority』: Machine

『Default Setting』: 0000 0000

#0 GDC Deceleration signal of the reference point return

0: Use X signal

1: Use G196 (X signal is invalid)

#1 EPN In the external workpiece number index, select the signal for specifying the workpiece.

0：Usable signals PN1～PN16

1：Usable extension signals EPN0～EPN13

#2 EPS The start signal at the external workpiece number index

0：Use the automatic operation start signal ST

1：Use the external workpiece index start signal EPNS

	#7	#6	#5	#4	#3	#2	#1	#0
3008						XSG		

『Modification authority』: Machine

『Default Setting』: 0000 0000

#2 XSG The X address is distributed to the skip signal and measurement position arrival signal

0：It is the fixed address

1：Changeable any X addresses

	#7	#6	#5	#4	#3	#2	#1	#0
3009			DECx					

『Modification authority』: Machine

『Parameter Type』: Bit axis

『Default Setting』: 0010 0000

#5 DECx: Deceleration signal of the reference point return

0: When the signal is 0 (low level), decelerate.

1: When the signal is 1 (high level), decelerate.

3010	Dwell time of the gating signals MT, TF and SF (MFT)

『Modification authority』: Machine

『Value Range』: 16 ms～32767 ms

『Default Setting』: 16

Set the time from sending codes M, S, T and B, till MF, SF, TF and BF being sent.

3011	Minimum width (MAW)of completion signals (FIN)of M, T and S (MAW)

『Modification authority』: Machine

『Value Range』: 16 ms～32767 ms

『Default Setting』: 16

Set the minimum width of the completion signals (FIN) of M, S, T and B function.

Note: The time is set by 8ms, if its setting value does not the multiplication of the 8; the carry-bit is multiplication of the 8.

3012	Address to be assigned to skip signals

『Modification authority』: Machine

『Value Range』: 0～127

『Default Setting』: 0

Set the skip signal to assort the X address and measure the address of the position arrival signal (0～127).

3013	X Address to be assigned to reference position return deceleration signals

『Modification authority』: Machine

『Value Range』: 0～127

『Default Setting』: 3

Set the X address to be assigned to the reference position return deceleration signal for each axis（0～127）。

3014	Bit position to be assigned to reference position return deceleration signals

『Modification authority』: Machine

『Value Range』: 0～7

『Default Setting』: 0

Set the X bit position to be assigned to the reference position return deceleration signal（*DECn）for each axis（0～7）。

3017	Output time of the resetting signal (RST)

『Modification authority』: Machine

『Value Range』: 0～255

『Default Setting』: 32

Set the dwell time when the resetting signal RST is output.

RST signal output time =resetting time + the parameter value X 16ms.

3019	Distribute the address of tool compensation value write-in signal

『Modification authority』: Machine

『Value Range』: 0～127

『Default Setting』: 0

Set the address of tool compensation value write-in signal for distributing the X address.

3020	Distribute the bit address of the skip signal X address

『Modification authority』: Machine

『Value Range』: 0～7

『Default Setting』: 0

Set the bit address for distributing the skip signal X address.

3021	Distribute the bit address of the multistep skips signal SKIP2

『Modification authority』: Machine

『Value Range』: 0～7

『Default Setting』: 0

Set the bit address for distributing the multistep skips signal SKIP2

3022	Distribute the bit address of the multistep skips signal SKIP3

『Modification authority』: Machine

『Value Range』: 0～7

『Default Setting』: 0

Set the bit address for distributing the multistep skips signal SKIP3

3023	Distribute the bit address of the multistep skips signal SKIP4

『Modification authority』: Machine

『Value Range』: 0～7

『Default Setting』: 0

Set the bit address for distributing the multistep skips signal SKIP4

3030	Allowable digits of M code（MCB）

『Modification authority』: Machine

『Value Range』: 2～8

『Default Setting』: 4

Set the allowable digits of M code.

3031	Allowable digits of S code（SCB）

『Modification authority』: Machine

『Value Range』: 1～5

『Default Setting』: 4

Set the allowable digits of S code.(Maximum 5 digits in S code is allowed).

3032	Allowable digits of T code（TCB）

『Modification authority』: Machine

『Value Range』: 2～8

『Default Setting』: 4

Set the allowable digits of T code.

3033	Allowable number of digits for the B code (BCN)

『Modification authority』: Machine

『Value Range』: 0～8

『Default Setting』: 0

The allowable bit number (0～8) of B code (The 2nd miscellaneous function)

3050	I/O unit quantity (IOMAX) of the system control

『Modification authority』: Machine

『Value Range』: 0～4

『Default Setting』: 0

Set the I/O unit quantity (up to 4) controlled by system.

3051	The logic ID number (IOID1) of system control I/O unit 1

『Modification authority』: Machine

『Value Range』: 0,100~110

『Default Setting』: 0

Set the logic ID number (0 means that this I/O unit disconnects with the GSKLink) of the system

control I/O unit 1.

3052	The logic ID number (IOID2) of system control I/O unit 2

『Modification authority』: Machine

『Value Range』: 0,100~110

『Default Setting』: 0

Set the logic ID number (0 means that this I/O unit disconnects with the GSKLink) of the system control I/O unit 2.

3053	The logic ID number (IOID3) of system control I/O unit 3

『Modification authority』: Machine

『Value Range』: 0,100~110

『Default Setting』: 0

Set the logic ID number (0 means that this I/O unit disconnects with the GSKLink) of the system control I/O unit 3.

3054	The logic ID number (IOID4) of system control I/O unit 4

『Modification authority』: Machine

『Value Range』: 0,100~110

『Default Setting』: 0

Set the logic ID number (0 means that this I/O unit disconnects with the GSKLink) of the system control I/O unit 4.

3060	The logic ID number (GWID) of the system gateway control

『Modification authority』: Machine

『Value Range』: 0,200~254

『Default Setting』: 0

This parameter setting system controls the logic ID number of the gateway. (0 means not use the gateway

	#7	#6	#5	#4	#3	#2	#1	#0
3061							GWP	GWC

『Modification authority』: Machine

『Validate method』: After power-on

『Parameter Type』: Bit

『Default Setting』: 0000 0000

#0 GWC Whether the gateway data uses the CRC verification

0：Disabled

1：Enabled

#1 GWP Whether the gate data uses the communication agreement

0：Disabled

1：Enabled

Appendix 1.10 Parameter of Display and Editing

	#7	#6	#5	#4	#3	#2	#1	#0
3101				BGD				

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#4 BGD Background editing selects the programs selected at the foreground

0: Editable

1: Unedited

	#7	#6	#5	#4	#3	#2	#1	#0
3104	DAC	DAL	DRC	DRL				MCN

『Modification authority』:Machine

『Default Setting』: 1100 0000

#0 MCN Display the machine position

0: Display based on the output units

(There isn't any connection with the metric system or the inch system, the metric machine displays as the metric units, the inch machine displays as the inch units.)

1: Display based on the input units

(When it is input in the metric system, display in the metric system; when it is input in the inch system, display in the inch system)

#4 DRL Display the relative position

0: Display the actual position including the tool offset (T serial)

1: Display the programming position without the tool offset (T serial)

Note: In T serial, the movement coordinate system compensates the tool appearance, (parameter LGT (NO.5002#4) is 0), display the programming position which ignores the tool compensation (the parameter is set as 1). However, the programming position without the tool appearance compensation value can not display.

#5 DRC Display the relative position

0: Display the actual position including the tool nose radius compensation (T serial)

1: Display the programming position without the tool nose radius compensation (T serial)

#6 DAL Display the absolute position

0: Display the actual position including the tool offset (T serial)

1: Display the programming position without the tool offset (T serial)

Note: In T serial, the movement coordinate system compensates the tool appearance (parameter LGT (NO.5002#4) is 0), and display the programming position which ignores the tool compensation (the parameter is set as 1). However, the programming position without the tool appearance compensation value can not display.

#7 DAC Display the absolutely position

0: Display the actual position including the tool nose radius compensation (T serial)

1: Display the programming position without the tool nose radius compensation (T serial)

	#7	#6	#5	#4	#3	#2	#1	#0
3107					REV	DNC		

『Modification authority』: Equipment management

『Default Setting』: 0001 0000

#2 DNC Whether clear display of DNC running programs during resetting

0: Not clear

1: Clear

#3 REV Display the actual speed in feeding/rev mode

0: mm/min or inch/min

1: mm/rev or inch/rev

	#7	#6	#5	#4	#3	#2	#1	#0
3110						AHC		

『Modification authority』: Equipment management authority

『Default Setting』: 0000 0100

#2 AHC Whether the alarm resume can be cleared by soft keys

0: Yes

1: No

	#7	#6	#5	#4	#3	#2	#1	#0
3111	NPA							

『Modification authority』: Equipment management

『Default Setting』: 1000 0000

#7 AHC Whether switch to alarm/information window when alarm occurs or information is input:

0: No

1: Yes

	#7	#6	#5	#4	#3	#2	#1	#0
3114								IPC

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#0 IPC On the current interface, press the function keys

0: Switch into the interface

1: Not switch into the interface

	#7	#6	#5	#4	#3	#2	#1	#0
3115								NDPx

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#0 NDPx Whether displays the current position

0：YES

1：NO

	#7	#6	#5	#4	#3	#2	#1	#0
3200		PSR		NE9				

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#4 NE9 Whether forbid the operations, such as program editing, deletion, modification and copy, etc. followed with the program number 9000.

0：Allow

1：Forbid

#6 PSR Whether allow loading and checking the protected program

0：Forbid

1：Allow

	#7	#6	#5	#4	#3	#2	#1	#0
3202			CPD					NE8

『Modification authority』: Equipment management

『Default Setting』: 0010 0000

#0 NE8 Whether forbid the operations, such as program editing, deletion, modification and copy, etc. of the program number 8000～8999.

0：Allow

1：Forbid

#5 CPD When NC program is deleted, confirm information and keys

0: Not display

1: Display

	#7	#6	#5	#4	#3	#2	#1	#0
3203	**MCL**	**MER**						

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#6 MER When the single block runs in MDI mode, after the last block is executed in the program, whether the executed programs are

0: Not deleted

1: Deleted

Note: Even MER is 0, when "%" (end code) is read in and executed, the program is also deleted ("%"is auto inserted at the end of the program).

#7 MCL Whether delete the programs edited in MDI mode through resetting

0: Not delete

1: Delete

	#7	#6	#5	#4	#3	#2	#1	#0
3209								**MPD**

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#0 MPD When the subprogram is executed, whether display the main program number

0: Not display

1: Display

3212	**NE9 needs the program quantity (CN9) protected from program**

『Modification authority』: Equipment management

『Value Range』: 0～999

『Default Setting』: 0

The programs after the No.9000 to be protected are set on the quantity, the program number protection range is 9000~9000+(No.3212), 0 is the overall protections (0～999).

3216	**Increment value (INC) during the serial number being auto inserted (INC)**

『Modification authority』:Equipment management

『Value Range』: 1～9999

『Default Setting』: 10

When the serial number (parameter SEQ (NO.0000#5) is 1) is auto inserted, it is the increment value of the serial number in each block.

3281	Language displayed on the screen (LANG)

『Modification authority』: Machine

『Value Range』: 0～1

『Default Setting』: 1

0：English 1：Chinese

3282	Reminding days before power off in the limited time (NDAYS)

『Modification authority』: Machine

『Value Range』: 1～30

『Default Setting』: 3

Appendix 1.11 Parameter of Programming

	#7	#6	#5	#4	#3	#2	#1	#0
3401		GSB				NCK		DPI

『Modification authority』: Equipment management

『Default Setting』: 0000 0001

#0 DPI The address is with the decimal point, but when the decimal point is omitted, the setting is as below:

0: Take them as the minimum setting units

1: Take them as the units of mm, inch and sec

#2 NCK During grammar checking, there are same N numbers

0: Alarm

1: Not alarm

#6 GSB Set the G code format

0：G code system A

1：G code system B

	#7	#6	#5	#4	#3	#2	#1	#0
3402	G23	CLR		FPM	G91			G01

『Modification authority』: Equipment management

『Default Setting』: 0101 0000

#0 G01 Mode during connecting the power supply

0: G00 mode (orientation)

1: G01 mode (linear interpolation)

#3 G91 In the G code system B, the system defaults as:

0：G90 mode (Absolute command)

1：G91 mode (Incremental command)

#4 FPM System defaults after power on

0: Feeding/rev

1: Feeding/min

#6 CLR Press the resetting key on MDI panel, the external resetting signal and the emergency stops, G code mode and the feedrate are

0: Hold mode

1: Switched to the power on state

#7 G23 when the power supply is connected, it is

0: G22 mode (Check the memory stroke)

1: G23 mode (Not check the memory stroke)

	#7	#6	#5	#4	#3	#2	#1	#0
3403		AD2	CIR	RER				

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#4 RER During arc interpolation, when R goes over the minor finishing point and isn't in the arc, and the radius doesn't exceed error:

0: Calculate the new radius, the path is semicircle

1: P/S alarms

#5 CIR In arc interpolation commands (G02, G03), there are no distance (I, J and K) from the starting point of the command to the center, and the arc radius isn't commanded, either.

0: Linear interpolation moves to the finishing point

1: P/S alarms

#6 AD2 In one block, two or two more same addresses are commanded

0: The following commands are valid.

1: The program is taken as wrong, P/S alarms.

Note: It alarms when the parameter is 1and two or two more G codes of one group are commanded in one block.

	#7	#6	#5	#4	#3	#2	#1	#0
3404	M3B	EOR	M02	M30				

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#4 M30 During auto running, process M30 command

0: return to the beginning of the program.

1: doesn't return to the beginning of the program.

#5 M02 During auto running, process M02 command

0: return to the beginning of the program.

1: doesn't return to the beginning of the program.

#6 EOR During executing the program, read in "%" (program end)

0: P/S alarms (stop auto running, display alarm state)

1: Not alarm (auto running stops, the system resets)

Note: When performing the "%" (end-of-program), CNC resets instead of closing the miscellaneous function output.

#7 M3B The quantity of M codes which can be commanded in one block

0: One

1: Maximum three

	#7	#6	#5	#4	#3	#2	#1	#0
3405			DDP					AUX

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#0 AUS In the 2nd miscellaneous function, the command counter decimal point input or the command with decimal point, as well the override corresponding to the command value output

0：The metric input is identical with the inch input

1：The override set by inch input sets as the 10 times of the override for the metric input

#5 DDP The angle command is directly input based upon the drawing dimension

0：Common specification

1：Command supplementary angle

3410	Circular radius allowable error (CRE)

『Modification authority』: Equipment management

『Value Range』: 0～9999 9999

Setting unit	IS－B	IS－C	Unit

Appendix

Input in mm	0.001	0.0001	mm
Input in inch system	0.0001	0.00001	inch

『Default Setting』:0

Set the allowable error value of arc interpolation (G02, G03) starting point radius and its finishing point radius. P/S alarms when arc interpolation radius error is more than the limit value.

Note: When the setting value is 0, it doesn't require checking the arc radius error.

3411	**M code 1 for stopping the buffer (BLKM1)**
3412	**M code 2 for stopping the buffer (BLKM2)**
3413	**M code 3 for stopping the buffer (BLKM3)**
3414	**M code 4 for stopping the buffer (BLKM4)**
3415	**M code 5 for stopping the buffer (BLKM5)**
3416	**M code 6 for stopping the buffer (BLKM6)**
3417	**M code 7 for stopping the buffer (BLKM7)**
3418	**M code 8 for stopping the buffer (BLKM8)**

『Modification authority』: Equipment management

『Validate method』: Immediately

『Parameter Type』: Word

『Value Range』: 0～9999

『Default Setting』: 0

This parameter sets the M code for stopping the buffer. Before ending the treatment of the M function at the side of the machinery, it is necessary to perform the operation treatment specified by M code by machinery, and then set this code.

	#7	#6	#5	#4	#3	#2	#1	#0
3450								**AUP**

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#0 AUP In the 2nd miscellaneous function command, the counter decimal point input, the

command with decimal point and the negative value command

0：Disabled

1：Enabled

	#7	#6	#5	#4	#3	#2	#1	#0
3453								**CRD**

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#0 CRD Chamfering/corner R is valid (the parameter CCR(No.8134)="1"）

0: Chamfering/corner R is enabled.

1: Direct drawing dimension programming is enabled.

3460	**Address for the second miscellaneous function (BCA)**

『Modification authority』: Equipment management

『Validate method』: Immediately

『Parameter Type』: Word

『Value Range』: 0,65~67，85~87

『Default Setting』: 0

The address(0,65~67，85~87) for the second miscellaneous function, when it is 0, the second miscellaneous function is off.

Appendix 1.12 Parameter of Screw Pitch Error Compensation

	#7	#6	#5	#4	#3	#2	#1	#0
3605								**BDPx**

『Modification authority』: Machine

『Validate method』: After power-on

『Value Range』: Bit axis

『Default Setting』: 0000 0000

#0 BDPx Whether use the bi-directional pitch error compensation

0：NO

1：YES

3620	**Screw pitch error compensation number in each axis reference point (NPR)**

『Modification authority』:Machine

『Validate method』: After power-on

『Parameter Type』: Word axis

『Value Range』: 0～1023

『Default Setting』: 0

3621	Number of the furthest screw pitch error compensation point of each axis in negative direction (NEN)

『Modification authority』:Machine

『Validate method』: After power-on

『Parameter Type』: Word axis

『Value Range』: 0～1023

『Default Setting』: 0

The parameter sets the number of the furthest screw pitch error compensation point of each axis in negative direction.

3622	Number of the furthest screw pitch error compensation point of each axis in positive direction (NEP)

『Modification authority』:Machine

『Validate method』: After power-on

『Parameter Type』: Word axis

『Value Range』: 0～1023

『Default Setting』: 0

The parameter sets the number of the furthest screw pitch error compensation point of each axis in positive direction.

Note: The parameter setting value should be greater than that of parameter NO.3620.

3623	Each axis screw pitch error compensation override (PCM)

『Modification authority』:Machine

『Validate method』: After power-on

『Parameter Type』: Word axis

『Value Range』: 0～100

『Default Setting』: 0

Set the override of screw pitch error compensation along each axis.

If the override is set as 1, the detection unit is same as that of compensation.

If the override is set as 0, the override is same as one when it is set as 1.

3624	Each axis screw pitch error compensation point interval (PCI)

『Modification authority』:Machine

『Validate method』: After power-on

『Parameter Type』: Word axis

『Default Setting』: 0～9 999 999

『Default Setting』: 0

Setting unit	IS－B	IS－C	Unit
Input in metric system	0.001	0.0001	mm
Input in inch system	0.0001	0.00001	inch
Rotary axis	0.001	0.0001	deg

The screw pitch compensation points are distributed in equal interval, and the interval value of each axis is set respectively. The minimum value of the interval is limited and set by the following formula: the minimum value = the maximum feedrate (rapid feedrate) / 7500.

Unit: Screw pitch compensation minimum interval: mm, inch and deg.

Maximum feedrate: mm/min, inch/min and deg/min.

For example: When the maximum feedrate is 15000mm/min, the minimum value of the screw pitch error compensation interval is 2mm.

But, according to the setting override, when the absolute value of the compensation point value exceeds 100, the interval of the compensation point is magnified by the override which is calculated by the following formula.

Override = Max compensation amount (absolute value)/128 (round up the digits after the decimal point)

Screw pitch compensation minimum interval = Value, which is obtained from the above maximum feedrate X override.

Note: The unit of the screw pitch compensation value is same as that of the detection.

The detection unit is relative with parameter No.1820 (command magnify ratio CMR) and the minimum movement unit, about the relation between the setting units and the minimum movement units, refer to the introduction of parameter No.1820.

3626	The compensation point (NPN) of the closest negative side for the bi-directional pitch error compensation

『Modification authority』:Machine

『Validate method』: After power-on

『Parameter Type』: Word axis

『Default Setting』: 0～1023

『Default Setting』: 0

When using the bi-directional pitch error compensation, set the closest negative side compensation point number when the tool moves along with the negative direction.

3627	The pitch error compensation value (PCD) in the reference point moves to the reference point from the negative direction of the origin direction return

『Modification authority』:Machine

『Validate method』: After power-on

『Parameter Type』: Word axis

『Default Setting』: -32768～32767

『Default Setting』: 0

When the origin direction is set as positive/negative direction; the pitch error compensation value in the reference point when the movement is set from negative/positive direction based upon absolute value.

3628	The setting value of the pitch compensation pulse frequency (NPF)

『Modification authority』:Machine

『Parameter Type』: Word

『Default Setting』:1～32

『Default Setting』: 8

The setting value of the pitch compensation pulse frequency

Appendix 1.13 Parameter of the Spindle Control

	#7	#6	#5	#4	#3	#2	#1	#0
3700						CSB		CSC

『Modification authority』: Equipment management

『Parameter Type』: Bit type

『Default Setting』: 0000 0000

#0 CSC Whether the coordinate value is cleared (Bit 2 of parameter 3700 sets to 0, this parameter is enabled) when the CS outline control shifts to spindle mode.

0：Keep

1：Clear

#2 CSB Whether the coordinate system is automatically set up when CS outline control shifts to the position mode

0：Disabled

1：Enabled

	#7	#6	#5	#4	#3	#2	#1	#0
3703					**MPP**			

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#3 MPP Whether replaces the signal SWS to perform the spindle selection by program command in the multi-axis control.

0：NO

1：YES

	#7	#6	#5	#4	#3	#2	#1	#0
3704	**SCS3**	**SCS2**						

『Modification authority』: Machine

『Validate method』: After power-on

『Default Setting』: 0000 0000

#6 SCS2 Whether Cs contour control of the 2nd spindle is

0: Invalid

1: Valid

#7 SCS3 Whether Cs contour control of the 3rd spindle is

0: Invalid

1: Valid

Note: Parameters SCS2 and SCS3 can be enabled by using the Cs outline control (that is, bit 2 of parameter No.8133 (SCS) is "1")

	#7	#6	#5	#4	#3	#2	#1	#0
3705				**EVS**				

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#4 EVS For S command, use spindle control function (spindle analog output or spindle serial output)

0: Not output S code and SF

1: Output S code and SF

	#7	#6	#5	#4	#3	#2	#1	#0
3706						**MPA**		

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#2 MPA In the multi-spindle control, when the spindle selection of the address P is set, and when the P does not specify with the S command:

0：Alarm issues（PS5303）

1：Use the last P specified by S_ P_;. After the power is turned on, use the value of parameter (№3775) when never ever specifies the P.

	#7	#6	#5	#4	#3	#2	#1	#0
3708		**TSO**				**SSC**	**SAT**	**SAR**

『Modification authority』: Equipment management

『Default Setting』: 0000 0011

#0 SAR Whether check the spindle speed reaching signal

0: Not check

1: Check

#1 SAT Whether check the spindle speed reaching signal when the thread cutting block is begun to be executed.

0: Check or not, which is set by parameter SAR (NO.3708#0)

1: Must check, which isn't connected with parameter SAR

Note: When the thread cutting block is continuously executed, the spindle speed reaching signal isn't checked in the thread cutting block after the 2nd block.

#2 SSC Whether check the spindle speed when performs the cutting feed

0：Do not check

1：Check

#6 TSO Whether the spindle override is valid during thread processing or tapping cycle

0: Invalid (fixed as 100%)

1: Valid

Note: In rigid tapping, the override is fixed as 100%, and there isn't any connection with the setting of the parameter.

	#7	#6	#5	#4	#3	#2	#1	#0
3709						**MSI**		**SAM**

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#0 SAM Times of sampling in spindle average speed

0: Four times (Generally it is set as 0)

1: One time

#2 MSI SIND signal is valid during multi-spindle control

0: It is only valid for the 1st spindle. (SIND signal of the 2nd spindle becomes invalid.)

1: No matter whether each spindle is selected or not, it is valid for all spindles. (Each spindle has its own SIND signal.)

3710	Spindle number control of CNC (CCS)

『Modification authority』: System

『Validate method』: After power-on

『Value Range』: 1～3

『Default Setting』: 1

Set the spindle number of the CNC control

	#7	#6	#5	#4	#3	#2	#1	#0
3713		MPC						

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#6 MPC In the multi-spindle, when the spindle selection is performed based upon the program command of address P, whether is automatically perform position encoder feedback shifting used in the thread cutting or feed/rev. based upon the selected spindle

0：Do not shift

1：Shift

3717	The amplifier number of each spindle (NSS)

『Modification authority』: System

『Validate method』: After power-on

『Parameter Type』: Word axis

『Value Range』: -4～99

『Default Setting』: 1

Set the amplifier number distributing to each spindle

Set value by	Corresponding interface	Remark

parameter		
0	Disconnect the spindle amplifier interface	
1~99	Spindle connects the logic ID number by GSKLink	The setting value is identical with the servo spindle logic ID number
-1～-4	Four groups analog value output ports of the spindle interfaces 1 and 2 on the corresponding the I/O unit 1	It is used in the frequency-conversion spindle
-11～-14	Four groups analog value output ports of the spindle interfaces 1 and 2 on the corresponding the I/O unit 2	
-21～-24	Four groups analog value output ports of the spindle interfaces 1 and 2 on the corresponding the I/O unit 3	
-31～-34	Four groups analog value output ports of the spindle interfaces 1 and 2 on the corresponding the I/O unit 4	

3720	Revolution of each spindle coder（CNT）

『Modification authority』: Machine

『Validate method』: After power-on

『Parameter Type』: Word axis

『Value Range』: 100～99999999

『Default Setting』: 1024

The revolution of each spindle coder is set

3721	Number of position coder gear teeth for each spindle (GOE)

『Modification authority』: Machine

『Parameter Type』: Word axis

『Value Range』: 1～9999

『Default Setting』: 1

Set the number of position coder gear teeth for each spindle during the speed control (feeding per revolution, thread cutting, etc).

3722	Number of gear teeth for each spindle (GOS)

『Modification authority』: Machine

『Parameter Type』: Word axis

『Value Range』: 1～9999

『Default Setting』: 1

Set the number of gear teeth for each spindle during the speed control (feeding per revolution, thread cutting, etc).

3723	Channel number corresponding to each spindle coder (CSE)

『Modification authority』: Machine

『Validate method』: After power-on

『Parameter Type』: Word axis

『Value Range』: 0～2

『Default Setting』: 0

Set the channel number corresponding to each spindle coder。

Value set by the parameter	Corresponding channel interface	
0	The data of spindle encoder is transmitted from GSKLink	It is used by using the GSKLink spindle and without external encoder.
1	With the 1st coder channel interface	It is used by using the external encoder.
2	With the 2nd coder channel interface	

3730	Increment adjustment Value of the spindle speed analog output (AGS)

『Modification authority』:Machine

『Parameter Type』: Word spindle

『Default Setting』: 1000

『Value Range』: 500～2000

『Value unit』: 0.1%

Set the increment adjustment Value of the spindle speed analog output. (Adjusting method)

（1） Set the standard setting value 1000,
（2） Command the spindle speed when the spindle speed analog output maximum voltage is 10V.
（3） Measure the output voltage.
（4） Set the value in the following formula in parameter No.3730:

$$\text{setting value} = \frac{10(V)}{\text{measured voltage}(V)} \times 1000$$

(5) After setting the parameter, command the spindle speed analog output as the spindle speed of the maximum voltage, again, and confirm the output voltage as 10V.

3731	Compensation value of the spindle speed analog output offset voltage (CSS)

『Modification authority』:Machine

『Parameter Type』: Word spindle

『Value Range』: -1000～+1000

『Default Setting』: 0

The parameter sets the compensation value of the spindle speed analog output offset voltage.

1. Set the standard setting value as 0.
2. Command the analog output voltage as 0V, which is the theoretical spindle speed.
3. Measure the output voltage.
4. Set the value in the following formula in parameter No.3731.

$$\text{setting value} = \frac{-8191 \times \text{offset voltage}(V)}{12.5}$$

5. After setting the parameter, command the analog output voltage as 0V, again, which is the theoretical spindle speed and confirm the voltage as 0V.

3740	Dwell time of the detection spindle speed reaching signal（SAD）

『Modification authority』:Machine

『Value Range』: 5～32767ms

『Default Setting』: 1000

Set the dwell time from executing S function to detecting the spindle speed reaching signal.

3741	Spindle maximum speed of gear 1 (MSG1)

3742	Spindle maximum speed of gear 2 (MSG2)

3743	Spindle maximum speed of gear 3 (MSG3)

3744	Spindle maximum speed of gear 4 (MSG4)

『Modification authority』:Machine

『Parameter Type』: Word spindle

『Default Setting』: 6000

『Value Range』: 0～32767r/min

The parameter sets the spindle maximum speed of each gear.

3770	Axis as the calculation reference during the constant surface speed control (ACS)

『Modification authority』:Machine

『Value Range』: 0～quantity of the control axes

『Default Setting』: 0

The parameter sets the axis as the calculation reference during the constant surface speed control.

Note: When it is set as 0, default X axis. Then, P value commanded in G96 block is not significant to the constant surface speed.

3771	Constant surface speed control mode (G96) spindle minimum speed (CFL)

『Modification authority』:Machine

『Value Range』: 0～32767r/min

『Default Setting』: 50

The parameter sets the spindle minimum speed when the constant surface speed control. During the constant surface speed control (G96), if the spindle speed is lower than the speed set by the parameter, it is limited in the parameter speed.

3772	Maximum spindle speed (MSS)

『Modification authority』:Machine

『Parameter Type』: Word spindle

『Value Range』: 0～32767r/min

『Default Setting』: 6000

The parameter sets the maximum spindle speed. The actual spindle speed is limited by the

maximum speed set by the parameter when the commanded spindle speed exceeds the maximum spindle speed, or the spindle speed after override exceeds the maximum spindle speed.

Note: 1. When the constant surface speed controls, no matter whether G96 or G97 is commanded, the spindle speed is limited by the maximum spindle speed.
2. When the setting value is 0, it is not limited by the speed.

3775	The default spindle in the multi-spindle selects the P command value (MPD)

『Modification authority』: System

『Validate method』: After power-on

『Value Range』: 0～99

『Default Setting』: 0

In the multi-spindle control, when parameter MPP(NO.3703#3)=1 and MPA（NO.3706#2）=1; there is no specification for the P command value in the command S_ P_ after the power is turned on.

3781	In multi-spindle control, when code P is used for spindle selection (MPS)

『Modification authority』: System

『Validate method』: After power-on

『Value Range』: 0～99

『Default Setting』: 0

When MPP(NO.3703#3)=1，In multi-spindle control, code P used for spindle selection is set with the parameter. And P code and S commands are specified in the same block

	#7	#6	#5	#4	#3	#2	#1	#0
4900								SFLR

『Modification authority』: Equipment management

『Parameter Type』: Bit axis

『Default Setting』: 0000 0000

#0 SFLR The setting unit of parameters 4911 and 4912 during the FLR in the spindle speed fluctuation detection function.

0：1% is regarded as the unit

1：0.1% is regarded as the unit

4911	The allowable rate q of the spindle arrival commanded speed (SSQ)

『Modification authority』: Equipment management

『Way of Validating』:

『Value Range』:

『Default Setting』: 100

The allowable rate q of the spindle arrival commanded speed is set in the spindle speed changing detection function

4912	The rate r of spindle change without sending the spindle speed changing detection alarm (SSR)

『Modification authority』: Equipment management

『Way of Validating』:

『Value Range』:

『Default Setting』: 100

The rate r of spindle change is set without sending the alarm in the spindle speed change detection function.

4913	The change magnitude i of the spindle speed without sending the spindle speed change detection alarm (SSI)

『Modification authority』: Equipment management

『Way of Validating』:

『Value Range』: 0～99999

『Default Setting』: 100

The allowable magnitude i is set in the spindle speed change detection function without sending the alarm

4914	The time p from commanding the speed change to starting detecting the spindle speed change (SSP)

『Modification authority』: Equipment management

『Way of Validating』:

『Value Range』: 1～999999

『Default Setting』: 100

In the spindle speed change detection function, the time p from commanding the speed change to starting detecting the spindle speed change

Appendix 1.14 Parameter of Tool Compensation

	#7	#6	#5	#4	#3	#2	#1	#0
5001		EVO		EVR				

『Modification authority』: Equipment management

『Default setting』: 0000 0000

#4 EVR In tool nose compensation mode C, when the tool compensation value is changed

0: It becomes valid from the next block which specifies T code.

1: It becomes valid from the next buffer block.

#6 EVO The rewritten value becomes valid when the compensation value of the tool position compensation mode is changed.

0: It is valid from the next block which specifies T code.

1: It is valid form the next buffer block.

	#7	#6	#5	#4	#3	#2	#1	#0
5002		LWM		LGT		LWT		LD1

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#0 LD1 Tool offset number

0: Specify through the last two digits of T code

1: Specify through the last one digit of T code

#2 LWT Tool wear compensation

0: Compensate through the tool traverse

1: Compensate through the coordinate system offset (there isn't any connection with LWM, and compensate in the block of T code)

#4 LGT Tool offset compensation mode

0: Compensate through the coordinate system offset (there isn't any connection with LWM, and compensate in the block of T code)

1: Compensate through the tool traverse

#6 LWM

0: Execute in T code block

1: Execute with axis movement meanwhile

Note: When LGT is 0, the offset is executed in T code block, and there isn't any connection with the parameter.

	#7	#6	#5	#4	#3	#2	#1	#0
5003		**LVC**				**CCN**		

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#2 CCN In the tool nose radius compensation mode, when the auto reference point return (G28) is commanded,

0: the tool nose traverses to the intermediate point.

1: But it is canceled until it traverses to the reference point.

#6 LVC Tool offset value is

0: Not cleared during resetting

1: Cleared during resetting

Note: The tooll offset function elimination by resetting should be enabled in the non-MDI mode.

	#7	#6	#5	#4	#3	#2	#1	#0
5004					**TS1**		**ORC**	

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#1 ORC Tool offset value

0: Specified by the diameter value (axes programmed by the diameter value)

1: Specified by the radius value

#3 TS1 The tool compensation value is directly input the touch inspection of sensor in the B function

0：It performs by 4 contactors

1：It performs by 1 contactor

	#7	#6	#5	#4	#3	#2	#1	#0
5005			**QNI**			**PRC**		

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#2 PRC in direct input of tool offset compensation value and workpiece coordinate system offset amount, the PRC signal is

0: Used

1: Not used

#5 QNI The tool compensation measure value is directly input to the function B, the

selection of the tool compensation number:

0：Operator selects by cursor

1：It performs by inputting the signal from PLC

	#7	#6	#5	#4	#3	#2	#1	#0
5006							**TGC**	**OIM**

『Modification authority』: Equipment management

『Validate method』: After power-on

『Default Setting』: 0000 0000

#0 OIM Switch between the inch system and the metric system, whether the tool offset value is auto changed

0: Not changed

1: Changed

#1 TGC Command T code in G50, G04 or G10 block

0: Not alarm

1: P/S alarms

	#7	#6	#5	#4	#3	#2	#1	#0
5008		**CNS**	**CNF**	**MCR**	**CNV**		**CNC**	**CNI**

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#0 CNI The tool nose radius compensation is interference checked

0: Execute

1: Not execute

#1 CNC When the tool nose radius compensation is interference checked and the difference between the programming movement direction and the offset movement direction is 90～270°

0: P/S alarms

1: Not alarm

#3 CNV The tool nose radius compensation (T serial) is interface checked and the vector is cleared

0: Execute

1: Not execute

#4 MCR If G41/G42 tool nose radius compensation is commanded in MDI mode, whether alarm

0: Not alarm

1: P/S alarm

Note: In MDI mode, the tool nose radius isn't compensated even it is set by the parameter.

#5 CNF When the tool nose radius compensation is interference checked, whether alarm when the internal full circle is cut

0: P/S alarms

1: Not alarm

#6 CNS The tool nose radius compensation is interference checked, whether alarm when the step is less than the tool radius

0: P/S alarms

1: Not alarm

	#7	#6	#5	#4	#3	#2	#1	#0
5009				**TSD**				**GSC**

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#0 GSC The tool compensation measure value is directly input the offset write input signal in function B

0：It inputs from one side of machinery

1：It inputs from one side of PLC

#4 TSD The tool compensation measure value is directly input in function B, so that the movement direction distinguish specification is:

0：Disabled

1：Enabled

5010	**During the tool nose compensation, the limit value of the vector is ignored when the tool traverses along the corner outside (CLV)**

『Modification authority』: Equipment management

『Value Range』: 0～16383

SETTING UNITS	IS-B	IS-C	UNITS
Input in metric system	0.001	0.0001	mm
Input in inch system	0.0001	0.00001	inch

『Default Setting』: 0

The limit value of the minor traverse value is ignored when the tool nose radius compensation is set and the tool traverses along the corner outside.

5013	Maximum value of the tool wearing compensation value（MTW）

『Modification authority』: Equipment management

『Value Range』:

		IS-B	IS-C
SETTING UNITS	Input in metric system	0.001 mm	0.0001 mm
	Input in inch system	0.0001 inch	0.00001 inch
SETTING RANGE	Input in metric system	0～9 999 999	0～99 999 999
	Input in inch system		

『Default Setting』: 10

The parameter sets the maximum value of the tool wearing compensation value.

Note: When the set absolute value of the tool wearing compensation value exceeds the maximum value, it alarms: Input from MDI alarm: too many digits. Exceed range (XXXX——XXXX) (input range is in the bracket).
Input through G10 alarm: The offset value input by G10 is out of the specified range.

5015	In the manual tool measure, the distance (X1P) of the inspection sensor X+ contact surface

『Modification authority』: Equipment management

『Value Range』: -99999999～99999999

Set the record of each contact surface from measure reference position to inspection sensor. Specify the axis of diameter programming, setting value and diameter value.

5016	In the manual tool measure, the distance (X1M) of the inspection sensor X- contact surface

『Modification authority』: Equipment management

『Value Range』: -99999999～99999999

Set the record of each contact surface from measure reference position to inspection sensor. Specify the axis of diameter programming, setting value and diameter value.

5017	In the manual tool measure, the distance (Z1P) of the inspection sensor Z+ contact surface

『Modification authority』: Equipment management

『Value Range』: -99999999～99999999

Set the record of each contact surface from measure reference position to inspection sensor.

Specify the axis of diameter programming, setting value and diameter value.

5018	In the manual tool measure, the distance (Z1M) of the inspection sensor Z- contact surface

『Modification authority』: Equipment management

『Value Range』: -99999999～99999999

Set the record of each contact surface from measure reference position to inspection sensor. Specify the axis of diameter programming, setting value and diameter value.

5020	Tool compensation measure value is directly input the tool offset number (TSB) in the function B

『Modification authority』: Equipment management

『Value Range』: 0～99

Set the tool offset number when the tool compensation value measure value is directly input to the function B (When the workpiece coordinate system offset value is set).

5021	In the manual tool measure, the memory movement interpolation cycle number before touching the detection sensor

『Modification authority』: Equipment management

『Value Range』: 0～8

Set the memorized movement interpolation cycle number for touching the inspection sensor, it is regarded as 8 when sets to 0.

5043	User the 1st offset axis number (YNSA1)

『Modification authority』: System

『Validate method』: After power-on

『Value Range』: 0～6

『Default Setting』: 0

Set the axis number for compensating the tool offset value of the 1st offset axis, regardless of the 0.

5044	User the 2nd offset axis number (YNSA2)

『Modification authority』: System

『Validate method』: After power-on

『Value Range』: 0～6

『Default Setting』: 0

Set the axis number for compensating the tool offset value of the 2nd offset axis, regardless of the 0.

5045	User the 3rd offset axis number (YNSA3)

『Modification authority』: System

『Validate method』: After power-on

『Value Range』: 0～6

『Default Setting』: 0

Set the axis number for compensating the tool offset value of the 3rd offset axis, regardless of the 0.

5046	User the 4th offset axis number (YNSA)

『Modification authority』: System

『Validate method』: After power-on

『Value Range』: 0～6

『Default Setting』: 0

Set the axis number for compensating the tool offset value of the 4th offset axis, regardless of the 0.

Appendix 1.15 Parameter of Canned Cycle

The setting unit of canned cycle parameter is shown as follows:

	IS-B	IS-C	UNITS
Input in metric system	0.001	0.0001	mm
Input in inch system	0.0001	0.00001	inch

Appendix 1.15.1 Parameter of Canned Cycle

	#7	#6	#5	#4	#3	#2	#1	#0
5101						RTR		

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#2 RTR In the G83 and G87

0: Specify the high-speed peck drilling cycle

1: Specify peck drilling cycle

	#7	#6	#5	#4	#3	#2	#1	#0
5102							**MRC**	

『Modification authority』:Equipment management

『Default Setting』: 0000 0000

#1 MRC The non-monotonic target shape is defined in multi-cycle command (G71 or G72), or non-monotonic Z axis is in G73 cycle and the run-out value is in Z axis or the Finishing allowance X axis is non-monotonic

0: Not alarm

1: Alarm

	#7	#6	#5	#4	#3	#2	#1	#0
5104						**FCK**		

『Modification authority』:Equipment management

『Default Setting』:0000 0100

#2 FCK In combined canned cycles (G71, G72 and G73), the processing appearance is

0: Not checked

1: Checked

	#7	#6	#5	#4	#3	#2	#1	#0
5105						**RF2**		

『Modification authority』: Equipment management

『Default Setting』: 0000 0100

#2 RF2 In the type II of the canned cycle G71, whether perform the rough-machining cutting

0：YES

1：NO

5110	**M code locking C axis in the canned cycle of drilling holes（CMD）**

『Modification authority』:Equipment management

『Value Range』: 3～99

『Default Setting』:35

Set M code, which can lock C axis, during the canned cycle of drilling holes.

5114	**The return value in high-speed peck drilling cycle（HPDCRD）**

『Modification authority』: Equipment management

『Value Range』: 0～99 999 999×（system limit increase）

『Default Setting』: 1000

The return value in G83, G87 high-speed peck drilling cycle is set by the parameter.

5115	The clearance value of peck drilling cycle（PDCRD）

『Modification authority』: Equipment management

『Value Range』： 0～99 999 999×（system limit increase）

『Default Setting』：1000

The clearance value of G83, G87 peck drilling cycle is set by the parameter.

Appendix 1.15.2 Parameter of Thread Cutting Cycle

5130	Chamfering value of the thread cutting cycle（G76，G92） (THD)

『Modification authority』:Equipment management

『Value Range』: 0～99× (0.1 screw pitch)

『Default Setting』: 0

The parameter sets the beveling value of G76 and G92 thread cutting cycle.

5131	Chamfering angle in threading cycle(G92, G76) (CAT)

『Modification authority』:Equipment management

『Value range』:0～89

『Default』:0

The chamfering angle in threading cycle (G76) of the multiple repetitive canned cycle and the thread cutting cycle (G92) of single canned cycle are set by the parameter. When the parameter is set to 0, a value of 45 degree is determined.

Appendix 1.15.3 Parameter of Thread Cutting Cycle

5132	Cutting value of the combined canned cycle G71 and G72 (THC)

『Modification authority』:Equipment management

『Value Range』: 1～99 999 999

『Default Setting』: 1000

Set the cutting value of G71 and G72 combined canned cycle.

5133	Tool retraction amount of G71 and G72 combined canned cycle (MCE)

『Modification authority』:Equipment management

『Value Range』: 0～99 999 999

『Default Setting』: 0

Set the run-out value of G71 and G72 combined canned cycle.

5135	Tool retraction amount of G73 combined canned cycle along X axis direction (G73XE)

5136	Tool retraction amount of G73 combined canned cycle along Z axis direction (G73ZE)

『Modification authority』:Equipment management

『Value Range』: -99 999 999～99 999 999

『Default Setting』: 0

Set the run-out value of G73 combined canned cycle along with X and Z axes direction

5137	Partition times of G73 combined canned cycle (G73DC)

『Modification authority』:Equipment management

『Default Setting』: 1

『Value Range』: 1～999

Set the partition times of G73 combined canned cycle.

5139	Tool retraction amount of G74 and G75 combined canned cycles (G74G75R)

『Modification authority』:Equipment management

『Value Range』: 0～99 999 999

『Default Setting』: 1000

Set the reversal value of G74 and G75 combined canned cycle.

5140	Cut-in amount of G76 compound canned cycle (G76MID)

『Modification authority』:Equipment management

『Value Range』: 0～99 999 999

『Default Setting』: 0

Set the minimum cutting value of G76 combined canned cycle.

5141	Finishing allowance of G76 combined canned cycle (G76FA)

『Modification authority』:Equipment management

『Value Range』: 1～99 999 999

『Default Setting』: 500

Set the finishing allowance of G76 combined canned cycle.

5142	Finishing cycle times of G76 combined canned cycle (G76FC)

『Modification authority』:Equipment management

『Value Range』: 1～99

『Default Setting』: 1

Set the finishing cycle times of G76 combined canned cycle.

5143	Tool nose angle of G76 combined canned cycle (G76TNA)

『Modification authority』:Equipment management

『Value Range』: 0～99 (deg)

『Default Setting』: 60

Set the tool nose angle of G76 combined canned cycle.

5149	Override value for retraction in boring cycles (G85, G89) (BCRDOV)

『Modification authority』:Equipment management

『Value Range』: 0～2000

『Default Setting』: 200

Set the velocity override value (%) of the retraction operation in boring cycle, it is separately enabled to the feedrate. When this speed sets to 0, it equals to the 200% speed override.

Appendix 1.16 Parameter of Rigid Tapping

	#7	#6	#5	#4	#3	#2	#1	#0
5200	SRS	FHD	PCP	DOV		CRG		G84

『Modification authority』:Equipment management

『Default Setting』: 0000 0000

#0 G84 Method of commanding the rigid tapping

0: M code commands the rigid tapping before command G84/G88 (refer to parameter NO.5210).

1: M code doesn't command the rigid tapping. G84/G88 is taken as G code of the rigid tapping, and the common tapping is not used.

#2 CRG After the command of canceling the rigid tapping method, rigid tapping:

0: After the rigid tapping signal RGTAP changes to 0, the method is canceled.

1: Before the rigid tapping signal RGTAP changes to 0, the method is canceled.

#4 DOV Override during the rigid tapping run-out, in the tapping rigid, the override for drawing

0: Invalid

1: Valid, override value is set by parameter 5211

#5 PCP When address Q is commanded in tapping cycle/rigid tapping

0: Used as a high-speed peck tapping cycle

1: Used as a peck tapping cycle

#6 FHD Feed pause and single block running in rigid tapping is:

0: Forbidden

1: Allowed

#7 SRS To select a spindle used for rigid tapping in multi-spindle control:

0:The spindle selection signals SWS1～SWS3 are used

1:The rigid tapping spindle selection signals RGTSP1～RGTSP3

	#7	#6	#5	#4	#3	#2	#1	#0
5201				**OV3**	**OVU**	**TDR**		

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#2 TDR Cutting time constant in rigid tapping

0: Uses a same parameter NO.5261 during cutting and extraction

1: Not use a same parameter during cutting and extraction, parameter NO.5261 for cutting, parameter NO.5271 for extraction

#3 OVU The increment unit of the override parameter（№5211）is

0：1%

1：10%

#4 OV3 The spindle speed for tool extraction is specified by the program (address J). The override during the tool extraction is

0：Invalid

1：Valid

	#7	#6	#5	#4	#3	#2	#1	#0
5202		**OVE**						

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#6 OVE The command range based on the extraction override command (address J) specified by the program during rigid tapping

0：100%～200

1：100%～2000%

	#7	#6	#5	#4	#3	#2	#1	#0
5203				**OVS**				

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#4 OVS In rigid tapping, override by the feedrate override signal and invalidation of override by the override cancel signal is

0：Disabled

1：Enabled

Note1：When the feedrate override is set as valid, the extraction override is invalid。
Note2：The spindle speed override is fixed to 100%, irrelevant with the parameter。

	#7	#6	#5	#4	#3	#2	#1	#0
5209								**RTX**

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#0 RTX In rigid tapping, the drilling axis is

0：Selected by the plane

1：Fixed as Z axis by G84, X axis by G88

Appendix

5210	**M code commanding the rigid tapping（RTMC）**

『Modification authority』:Equipment management

『Value Range』: 0～255

『Default Setting』: 0

M code is set to specify the rigid tapping method. When it is set as 0, CNC takes it as M29.

5211	**Override of extraction during rigid tapping（RTEOV）**

『Modification authority』:Equipment management

『Value Range』: 0～200

『Value Unit』: 1% or 10%

『Default Setting』: 100

The override value of extraction during rigid tapping。

Note 1：When parameter DOV(No.5200#4) is 1, the override value is valid.。
Note 2：When parameter OVU (No.5201#3) is 1, the unit of the setting data is 10%, and the override can be applied to the extraction of 2000%.

5213	Return or clearance in peck tapping cycle (PRTRD)

『Modification authority』: Equipment management

『Value Range』: 0～99999999

『Value Unit』:

SETTING UNITS	IS-B	IS-C	UNITS
linear axis（Input in metric system）	0.001	0.0001	mm
linear axis（Input in inch system）	0.0001	0.00001	Inch

『Default Setting』: 0

The return in high-speed peck tapping cycle or clearance in peck tapping cycle is set by the parameter.

5241	Maximum spindle speed when rigid tapping（RTMS）

『Modification authority』:Equipment management

『Value Range』: 0～9999

『Default Setting』: 1000

Set the spindle maximum speed in rigid tapping.

5261	Time constant of linear acceleration/deceleration when rigid tapping（RTLT）

『Modification authority』:Equipment management

『Value Range』: 0～4000ms

『Default Setting』:100

Time constant of linear acceleration or deceleration for the spindle for the rigid tapping.

5271	Linear acceleration/deceleration time constant when rigid tapping retraction（RTET）

『Modification authority』:Equipment management

『Value Range』: 0～4000ms

『Default Setting』: 100

Set the time constant of linear acceleration or deceleration of the spindle and the tapping axis during the rigid tapping run-out.

Note: The parameter is valid only when parameter TDR (NO.5201 BIT2) is set as 1.

5275	**Actually, the tapping axis lags behind the compensation cycle number (ZBK) sampled by spindle encoder in G84/G88**

『Modification authority』: Equipment management

『Value Range』: 0～10

『Default Setting』: 6

Set in the G84/G88 common tapping (non-rigid tapping), the tapping axis lags behind the compensation cycle number sampled by spindle encoder. Generally, it is better set it to 4~8.

Appendix 1.17 Parameter of Polar coordinate interpolation

	#7	#6	#5	#4	#3	#2	#1	#0
5450							AFC	

『Modification authority』:Equipment management

『Default Setting』: 0000 0000

#0 AFC Whether use the auto override and the auto speed in the polar coordinate interpolation mode.

0: Not use

1: Use

Note: In the polar coordinate interpolation mode, the more closely the tool is near to the work piece center, the bigger the speed vector of the rotary axis is. If the center part exceeds the maximum cutting speed (parameter NO.5462), the servo (NO.411) alarms. Auto feedrate override and auto feedrate limit function auto controls the feedrate, then, the speed vector of the rotary axis doesn't exceed the maximum cutting feedrate.

5460	**Specify the polar coordinate interpolation axis (linear axis)（LAI）**

5461	**Specify the polar coordinate interpolation axis (rotary axis)（RAI）**

『Modification authority』:Machine

『Value Range』: 1～quantity of the control axes

『Default Setting』: NO.5460 is 1; NO.5461 is 5

Set the control axis numbers of the linear axis and the rotary axis for polar coordinate interpolation

5462	**Maximum cutting feedrate of the polar coordinate interpolation（MFI）**

『Modification authority』:Machine

『Default Setting』: 8000

	IS-B	IS-C	UNITS
Machine in metric system	0, 6～24 000	0, 6～10 000	mm/min
Machine in inch system	0, 6～9 600	0, 6～4 800	inch/min

Set the valid maximum feedrate of the polar coordinate interpolation. If the commanded speed is greater than the value, the speed is limited by the maximum one. When the parameter is set as 0, the speed in the polar coordinate interpolation is limited by the maximum cutting feedrate (parameter NO.1422) value.

5463	**Allowable auto override percentage in polar coordinate interpolation（API）**

『Modification authority』:Equipment management

『Value Range』: 0～100 (%)

『Default Setting』: 0

When the polar coordinate interpolation is set, the percentages of the auto override are allowed to limit the cutting feedrate of the rotary axis.

The allowable speed of the rotary axis = Maximum cutting feedrate X override percentage

In polar coordinate interpolation, the more closely the tool is near to the work piece center, the bigger the speed vector of the rotary axis is. When it exceeds the allowable speed, the feedrate automatically multiplies by the override value calculated through the following formula:

Override = Allowable speed of the rotary axis/the speed vector of the rotary axis X 100%

If the revolving speed after timing the override still exceeds the allowable speed, the feedrate is limited in the allowable maximum cutting feedrate (auto speed limit function) .

Note: When the parameter value is set as 0, it is taken as 90%;

To limit the auto speed override and the auto speed, the parameter AFC (NO.5450#1) is set as 1.

Appendix 1.18 Parameter of User Macro Program

	#7	#6	#5	#4	#3	#2	#1	#0
6000			**SBM**					**G67**

『Modification authority』:Equipment management

『Default Setting』: 0000 0000

#0 G67 Macro program mode calling (G66) mode is not set, but mode calling command (G67) is

canceled.

0: P/S alarms (NO.122)

1: Ignore G67

#5 SBM Whether use the single block to stop in the user macro program

0: Not use

1: Use

	#7	#6	#5	#4	#3	#2	#1	#0
6001	**CLV**	**CCV**						

『Modification authority』:Equipment management

『Default Setting』: 0100 0000

#6 CCV After reset, the user macro public variables 100～199 are：

0: Cleared as null

1: Not cleared

Note: In MDI mode, the macro public variables are not cleared after reset.

#7 CLV After resetting, the user macro program part vector 1～33 is

0: Cleared as null

1: Not cleared

	#7	#6	#5	#4	#3	#2	#1	#0
6004							**MFZ**	**NAT**

『Modification authority』:Equipment management

『Default Setting』: 0000 0000

#0 NAT The function command ATAN of the user macro program

0: Result of ATAN is 0～360.0 Result of ASIN is 270.0～0～90.0

1: Result of ATAN is -180.0～0～180.0n Result of ASIN is -90～0～90

#1 MFZ The angles of STN, COS or TAN, which are operation commands of the user macro program, are 1.0×10^{-8} or less, or the operation result is not exact 0

0: Underflow process

1: Reduction to 0

	#7	#6	#5	#4	#3	#2	#1	#0
6008		**GMP**	**TMP**					**F0C**

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#0 F0C The macro variable operation result

0: The alarm occurs when the data range exceeds ±1E308

1: The alarm occurs when the data range exceeds ±1E47

#5 TMP Whether allow the T code to call macro program

0：NO

1：YES

#6 GMP Whether allow M code calling the macro

0：No

1：Yes

6031	The beginning number of the variable to be protected in the common variables （#500~#999）（MPH）

『Modification authority』: Equipment management

『Value Range』: 500～999

『Default Setting』: 0

The beginning number of the variable in the common variables（#500~#999）is protected

6032	The end number of the variable to be protected in the common variables （#500~#999）（MPT）

『Modification authority』: Equipment management

『Value Range』: 500～999

『Default Setting』: 0

The end number of the variable in the common variables（#500~#999）is protected

6060	T code for calling Macro PROG. NO.9010 (TLM1)
6061	T code for calling Macro PROG. NO.9011 (TLM2)
6062	T code for calling Macro PROG. NO.9012(TLM3)
6063	T code for calling Macro PROG. NO.9013 (TLM4)
6064	T code for calling Macro PROG. NO.9014(TLM5)
6065	T code for calling Macro PROG. NO.9015 (TLM6)

6066	**T code for calling Macro PROG. NO.9016 (TLM7)**
6067	**T code for calling Macro PROG. NO.90107(TLM8)**
6068	**T code for calling Macro PROG. NO.90108(TLM9)**
6069	**T code for calling Macro PROG. NO.90109(TLM10)**

『Modification authority』: Equipment management

『Value Range』: 0～99999999

『Default Setting』: 0

T code for calling Macro PROG. NO.9010~9019 is set by the parameter.

6080	**M code for calling Macro PROG. NO.9020 (MLM1)**
6081	**M code for calling Macro PROG. NO.9021 (MLM2)**
6082	**M code for calling Macro PROG. NO.9022 (MLM3)**
6083	**M code for calling Macro PROG. NO.9023 (MLM4)**
6084	**M code for calling Macro PROG. NO.9024 (MLM5)**
6085	**M code for calling Macro PROG. NO.9025 (MLM6)**
6086	**M code for calling Macro PROG. NO.9026 (MLM7)**
6087	**M code for calling Macro PROG. NO.9027 (MLM8)**
6088	**M code for calling Macro PROG. NO.9028 (MLM9)**
6089	**M code for calling Macro PROG. NO.9029 (MLM10)**

『Modification authority』: Equipment management

『Value Range』: 3～99999999

『Default Setting』: 0

M code for calling Macro PROG. NO.9020~9029 is set by the parameter.

Appendix 1.19 Parameter of the Skip Function

	#7	#6	#5	#4	#3	#2	#1	#0
6200	SKF						SK0	

『Modification authority』: Machine

『Default Setting』: 0000 0000

#1 SK0 Set the valid state of the skip signal

0: valid when the input signal is “1”

1: valid when the input signal is “0”

#7 SKF Dry run and override for G31 jumping command are:

0: disabled

1: enabled

	#7	#6	#5	#4	#3	#2	#1	#0
6210		MDC						

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#6 MDC the measured automatic tool compensation value is

0: added to the current offset value

1: subtracted from the current offset value

	#7	#6	#5	#4	#3	#2	#1	#0
6240	IGA							AE0

『Modification authority』: Machine

『Validate method』: After power-on

『Default Setting』: 0000 0000

#0 AE0 Automatic tool compensation signal (X3.6), XAE2 (X3.7) indicates:

0: the measuring position is reached when it is 1

1: the measuring position is reached when it is 1

#7 IGA Automatic tool compensation function is:

0: used

1: not used

6241	Feedrate during automatic compensation (for XAE1 signal)(ATOF1)

Appendix

6242	Feedrate during automatic compensation (for XAE2 signal)(ATOF2)

『Modification authority』: Machine

『Default Setting』: 1000

『Value setting』:

SETTIN UNIT	VALUE UNIT	VALID RANGE		DEFAULT
		IS-B	IS-C	
Metric	1mm/min	6～15000	6～12000	1000
Inch	0.1inch/min	6～6000	6～4800	

These two parameters set the feedrate during automatic tool compensation.

Note: When the setting value of parameter No. 6242 is valid, the setting value of parameter No. 6241 is valid too.

6251	The γ value of X axis during automatic tool compensation (ATOR1)

6252	The γ value of Z axis during automatic tool compensation (ATOR2)

『Modification authority』: Equipment management

『Value range』: 1～99999999

『Default Setting』: 1000

These two parameters set the γ value in tool compensation function in sequence.

Note: It is always set based upon the radius value regardless of the diameter or radius specification

6254	The ε value of X axis during automatic tool compensation (ATOE1)

6255	The ε value of Z axis during automatic tool compensation (ATOE2)

『Modification authority』: Equipment management

『Value range』: 1～99999999

SETTING UNIT	IS-B	IS-C	unit
Linear axis (metric input)	0.001	0.0001	mm
Linear axis (inch input)	0.0001	0.00001	inch
Rotary axis	0.001	0.0001	deg

These two parameters set the ε value in tool compensation function in sequence.

Note: The value is set in radius no matter diameter or radius programming is specified

Appendix 1.20 MPG Retraction Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
6400		**MGO**						**RPO**

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#0 RPO In the retraction function, the feedrate at the rapid traverse rate:

0：Clamped at the 10% of its equivalent override

1：Clamped at the 100% of its equivalent override

#6 MCO In the retraction function, perform the relative G code with measurement:

0：MPG pulse enabled

1：MPG pulse disabled, it always performs below the 100% override

	#7	#6	#5	#4	#3	#2	#1	#0
6401								**CRH**

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#0 CRH Whether forbid the MPG retraction in the hand MPG retraction method:

0：YES

1：NO

6405	**Clamp the override value (MLF) of the MPG retraction function at the rapid traverse rate**

『Modification authority』: Equipment management

『Value Range』: 0～100

『Default Setting』: 0

Set the override value for clamping at the MPG retraction function at the rapid traverse rate, when the 0 is set, this function is disabled, and the RPO(No.6400#0) is enabled (0～100).

6410	**The movement value of MPG per one pulse (MPM)**

『Modification authority』: Equipment management

『Value Range』: 0～100

『Default Setting』: 0

Set the movement value (0～100) of the MPG per one pulse by the override conversion

The mechanical movement value when actually rotates the MPG, which can be calculated according to the following method:

[Command speed] × [MPG override] × ([Parameter setting value]/100) ×8/60000 (mm or inch)

For example: The command speed is 30mm/min; the MPG override is 100; the movement value caused by MPG per one pulse in the case of the parameter No.6410 sets to 1, refer to the following formula:

[The movement value per one pulse]=30[mm/min] ×100×(1/100) ×(8/60000)[min]=0.004mm

Appendix 1.21 Parameter of Graphic Display

	#7	#6	#5	#4	#3	#2	#1	#0
6500					**DPA**			

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#3 DPA Current position display on the graphic display screen

0: Display the actual position including the tool nose radius compensation and tool offset

1: Display the programming position without tool compensation and offset

Appendix 1.22 Parameter of Run Hour and Parts Count Display

	#7	#6	#5	#4	#3	#2	#1	#0
6700							**PRT**	**PCM**

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#0 PCM M codes counting the total quantity of the processing parts and the quantity of the processing parts

0: M codes specified by M02 and M30 and parameter NO.6710

1: M codes only specified by parameter NO.6710

#1 PRT During setting, the signal PRTSF (F62.7) of the sufficient quantity of the processing parts is

0: Cut off

1: Not cut off

6710	M codes counting the total quantity of the processing parts and the quantity of the processing parts (MPC)

『Modification authority』: Machine

『Value Range』: 0～9999

『Default Setting』: 0

The machine program executes M codes set by the parameter, total quantity of the processing parts and quantity of the processing parts plus 1, respectively.

Note:When the setting value is 0, it is invalid (M00 can't count the parts). And it can't be set as 98 and 99, 198, neither.

6713	Quantity of the required parts (RPM)

『Modification authority』: Machine

『Value Range』: 0～9999

『Default Setting』: 0

When the quantity of the processing parts equals to that of the parts required being processed, the signal PRTSF (F62.7) of the enough quantity of the required parts outputs to PLC. However,

Note: If the quantity is 0, it is regarded as infinitely great, not output to PRTSF.

Appendix 1.23 Parameter for Tool Life Span Administration

	#7	#6	#5	#4	#3	#2	#1	#0
6800			GRC	GPS	SIG	LTM	GS2	GS1

『Modification authority』: Equipment management

『Validate method』: After power-on

『Default Setting』: 0000 00000

#0 GS1 The registered group numbers and the tool numbers of each 1 group can be changed by setting the parameters GS1, GS2 based upon the Max. group number in the parameter 6813.

#1 GS2 The registered group numbers and the tool numbers of each 1 group can be changed by setting the parameters GS1, GS2 based upon the Max. group number in the parameter 6813.

The relationships between GS1, GS2 and tool numbers are shown below:

GS2	GS1	Group Number	Tool Number
0	0	The 1/8 from the 1 to the Max. group number (No.6813)	1~16
0	1	The 1/4 from the 1 to the Max. group number (No.6813)	1~8

1	0	The 1/2 from the 1 to the Max. group number (No.6813)	1~4
1	1	The Max. group number (No.6813)	1~2

#2 LTM The specification of tool life span count type

0：Specify based upon the times

1：Specify based upon time

#3 SIG In the tool skip based on the signal, whether select the signal input group number by the tool group number

0：Do not input

1：Input

#4 GRS When inputting the tool-change resetting signal TLRST:

0: The clearing group is specified by the GRC of parameter 6800#5

1: Clear the registered executing data of the overall groups

#5 GRC When inputting the tool-change resetting signal TLRST, the specified group:

0: Automatically inspect the group used up of the life span by CNC

1: Select the signal specification by external tool group number

	#7	#6	#5	#4	#3	#2	#1	#0
6801						**LVF**	**TSM**	

『Factory type』： Equipment

『Modification authority』: Equipment management

『Default Setting』： 0000 00000

#1 TSM In the tool life span administration function, the life span count exists in the case of multi-offset command

0: The counting is performed based upon the each same tool number

1: The counting is performed based upon the each cutter

#2 LVF Use the time count life span value in the tool life span administration function, the tool life span count override signal *TLVO～*TLV9<G049.0～G050.1> places at:

0：Disabled

1：Enabled

	#7	#6	#5	#4	#3	#2	#1	#0
6802	**RMT**							**T99**

『Default Setting』： 0000 00000

『Modification authority』: Equipment management

『Validate method』：Immediately

#0 T99 When the tool group of the life span is used up, perform the M99 in the main program:

0: Do not output the tool-change signal

1: Output the tool-change signal, and then enter to the auto operation stop state.

#7 RMT Tool life span predicted signal TLCHB

0: The residual value of life-span (life-span value — life-span counter), ≤ the remainder value of the ON life-span when resetting the counting value > It is OFF when resetting the counting value

1: The surplus of life-span = ON during resetting counting value, the residual value of life-span ≠ OFF during the resetting counting value

	#7	#6	#5	#4	#3	#2	#1	#0
6804		**LFI**						

『Default Setting』: 0000 00000

『Modification authority』: Equipment management

『Validate method』: Immediately

#6 LFI The selected tool life-span counting in the tool life administration

0: Enabled

1: Count the disabled signal LFCIV (G48.2) by tool life-span, the shifting is performed between enabled or disabled.

	#7	#6	#5	#4	#3	#2	#1	#0
6805							**FGL**	

『Default Setting』: 0000 00000

『Modification authority』: Equipment management

『Validate method』: Immediately

#1 FGL life-span counting type is registered based upon the life data of G10 in the case of the specified time

0: Unit by 1 minute

1: Unit by 0.1 second

6810	**Tool life-span administration ignore number (TLC)**

『Default Setting』: 0

『Modification authority』: Equipment management

『Value Range』: 0～9999 9999

『Validate method』: Immediately

When the figure exceeds the set value by using the T code, some value deducted from the set

value based upon the T code numerical value becomes the tool group number of the tool life-span administration.

6811	The M code is used by tool life-span counting restart (MRN)

『Default Setting』: 0

『Modification authority』: Equipment management

『Value Range』: 0～127

『Validate method』: Immediately

In this case, the life-span existence is set by times, the tool group when the tool life-span counting restarting specifies the M code is used up.

The tool-change signal (TLCH) may also be output even if only one signal; when it is set to 0, the parameter will then be ignored.

6813	The Max. group number of the tool life-span administration (MTN)

『Default Setting』: 0

『Modification authority』: Equipment management

『Value Range』: 0, 8, 16, 32, 64, 128

『Validate method』: After power-on

Set the used top group number of each path, after this parameter is set, the power should be temporarily turned off.

6844	The residual span using times of tool (TLP)

『Default Setting』: 0

『Modification authority』: Equipment management

『Value Range』: 0~65535

『Validate method』: Immediately

In the case of the tool life-span is specified, cutter output span reaches to the tool residual span of the predictive signal (Using times).

6845	The remainder span using time of tool (TLR)

『Default Setting』: 0

『Modification authority』: Equipment management

『Value Range』: 0~4300

『Validate method』: Immediately

In the case of the tool life-span is specified, cutter output span reaches to the tool residual span

of the predictive signal (Using time).

Appendix 1.24 Parameter of MPG Feed

	#7	#6	#5	#4	#3	#2	#1	#0
7100				**HPF**				**JHD**

『Modification authority』: Machine

『Default Setting』: 0000 0000

#0 JHD MPG feeding in JOG mode or increment feeding in MPG feed mode

0: Invalid

1: Valid

	JHD=0		JHD=1	
	JOG MODE	MPG MODE	JOG MODE	MPG MODE
JOG feeding	O	×	O	×
MPG feeding	×	O	O	O
Increment feeding	×	×	×	O

#4 HPF When MPG feedrate exceeds the manual rapid movement speed

0: The speed is limited in the manual rapid movement speed, the pulse exceeding the manual rapid movement part is ignored (The scale of MPG does not comply with the movement amount)

1: The speed is limited in the manual rapid movement speed; the exceeding part isn't ignored but saved in CNC. (Although MPG is stopped, the machine still moves the pulse value saved in CNC and then stops.)

	#7	#6	#5	#4	#3	#2	#1	#0
7102								**HNGx**

『Modification authority』: Machine

『Parameter Type』: Bit axis

『Default Setting』: 0000 0000

#0 HNGx Revolving direction of each axis movement direction and that of MPG

0: Same

1: Opposite

	#7	#6	#5	#4	#3	#2	#1	#0
7103						**HNT**		

『Modification authority』: Machine

『Default setting』: 0000 0000

#2 HNT Movement amount override of the incremental feed/MPG feed is set to the one that is selected by the MPG feed movement selection signal

0：1

1：10 times

7110	Number of MPG (NMP)

『Modification authority』: Machine

『Value Range』: 1~2

『Default Setting』: 1

Set the quantity of MPG.

7113	MPG feed override M(MFM)

『Modification authority』: Machine

『Value Range』: 1～127

『Default Setting』: 100

Set the override when MPG feeding movement value selection signals MP1=0, MP2=1.

7114	MPG feed override N(MFN)

『Modification authority』: Machine

『Value Range』: 1～1000

『Default Setting』: 1000

Set MPG feeding override when MPG feeding movement value selecting signals MP1=1, MP2=1.

MOVEMENT VALUE SELECTING SIGNAL		MOVEMENT VALUE (MPG FEEDING)
MP2	MP1	
0	0	Minimum setting unit * 1
0	1	Minimum setting unit * 10
1	0	Minimum setting unit * M
1	1	Minimum setting unit * N

7117	Allowable pulse cumulative value in MPG feed (APM)

『Modification authority』: Machine

『Value Range』: 0～1000

『Default Setting』: 1000

When MPG feeding instance exceeds the rapid movement speed, the pulse exceeding the rapid movement is not canceled but saved. The parameter sets the allowable value of the memory capacity.

Note: When overrides, such as X100 or more than it, are selected, MPG rapidly turns round. MPG feeding is more than the rapid movement speed; the speed is limited by the rapid movement speed. The pulse exceeding the rapid movement speed is ignored; therefore, the scale value of MPG doesn't comply with the actual movement value. Then, If the allowable value is preset in the parameter, the pulse exceeding the rapid movement speed is not canceled, but saved in CNC temporarily (the part exceeding the allowable value is ignored). When MPG revolving speed becomes slower or the revolving stops, the saved pulse changes into the movement command and outputs. Pay attention to it if the allowable value is set too big, even MPG is stopped revolving, CNC won't stop until the remaining pulse is completed.

Appendix 1.25 Parameters of Program Restart

	#7	#6	#5	#4	#3	#2	#1	#0
7300	**MOU**	**MOA**						

『Modification authority』: Machine

『Default Setting』: 0000 0000

#6 MOA In program restart operation, before movement to a machining restart point

0:The last M, S, T and B codes are output

1:All M codes and the last S, T and B codes are output

#7 MOU In program restart operation, before movement to a machining restart point after restart block search

0: The M, S, T and B codes are NOT output

1: The M, S, T and B codes are output

	#7	#6	#5	#4	#3	#2	#1	#0
7301								**ROF**

『Modification authority』: Machine

『Default Setting』: 0000 0000

#0 ROF In the restart coordinate display on the program restart screen, whether display the various tool compensation values

0: Display the tool compensation and offset

1: Set by DAL, bit 6 of parameter No.3104 or DAC, bit 7 of parameter No.3104

7310	The axis sequence by dry run after a program is restarted (ROAX)

『Modification authority』: Machine

『Value Range』: 1～quantity of the control axes

『Default Setting』: 1

The axis sequence when the machine moves to the restart point by dry run and is specified by the dedicated axis after a program is restarted

Appendix 1.26 Polygon Machining Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
7603		PQS		PSM	PLR			

『Modification authority』: Machine

『Default Setting』: 0000 0000

#3 PLR The tool rotation axis with each movement value of the polygon machining

0: Round off by the setting value of the parameter 7620

1: Round off based upon 360

#4 PSM The workpiece rotation axis working mode of the polygon machining

0: Speed mode

1: Position mode

#6 PQS The PQ value of the polygon machining is:

0: The rotation ratio between the tool rotation axis and workpiece rotation axis

1: The ratio value between the polygon number and tool number

7610	The controllable axis number (PCA) for using the tool rotation axis of the polygon machining

『Modification authority』: Machine

『Value Range』: 0～quantity of the control axes

『Default Setting』: 0

Set the controllable axis number of the tool rotation axis for using the polygon machining, when it is set to 0, which means that this function does not work.

7620	The movement amount (PEM) per each rotation for using tool rotation axis of polygon machining

『Modification authority』: Machine

『Value Range』: 0～3600000

『Default Setting』: 0

Set the movement amount per each cycle of the tool rotation axis

7621	The upper-limit speed (PSM) for using the tool rotation axis of the polygon machining

『Modification authority』: Machine

『Value Range』: 0～99999999

『Default Setting』: 0

Set the upper-limit speed of the tool rotation axis

Appendix 1.27 Parameter of PLC Axis Control

	#7	#6	#5	#4	#3	#2	#1	#0
8001		AUX	NCC		RDE	OVE		MLE

『Modification authority』: Machine

『Default Setting』: 0000 0000

#0 MLE Whether the locking machine signal MLK of PLC control axis is valid

0: Valid

1: Invalid

#2 OVE Signals relative with the dry run and the override controlled by PLC axis

0: Same signals controlled by CNC

1: Signals especially used in PLC

#3 RDE In PLC axes control, whether the dry run is valid for the rapid feeding commands

0: Invalid

1: Valid

#5 NCC For PLC control axes (the control axes select the axes chosen by the signal), command the program to command the movement

0: According to the axis control command, PLC controls the axis, P/S (No.139) alarms; the axis is not controlled, CNC command is valid.

1: P/S (No.139) alarms.

#6 AUX The number of bytes for the code of an auxiliary function to be output is

0：One

1：Two

	#7	#6	#5	#4	#3	#2	#1	#0
8002	FR2	FR1	PF2	PF1	F10		DWE	RPD

『Modification authority』: Machine

『Default Setting』: 0000 0000

#0 RPD The rapid movement speed of PLC control axis

0: Feedrate set by parameter No.1420

1: In axis control command, feedrate set by feedrate Value

#1 DWE When use the increment system IS-C, the minimum time specified by the pause command during PLC axis control

0: 1ms

1: 0.1ms

#3 F10 In PLC axis control, the minimum increment units of the cutting feedrate (per min)

F10	Input in metric system	Input in inch system
0	1mm/min	0.01inch/min
1	10mm/min	0.1inch/min

#4, #5 PR1, PR2 In PLC axis control, the least increment unit of cutting feed

PF2	PF1	Speed
0	0	1/1
0	1	1/10
1	0	1/100
1	1	1/1000

#6, #7 FR1, FR2 The feedrate units of per revolution feeding during PLC axis control

FR2	FR1	Input in metric system	Input in inch system
0	0	0.0001mm/rev	0.000001inch/rev
1	1		
0	1	0.001mm/rev	0.00001inch/rev
1	0	0.01mm/rev	0.0001inch/rev

	#7	#6	#5	#4	#3	#2	#1	#0
8004		**NCI**	**DSL**			**JFM**		

『Modification authority』: Machine

『Default Setting』: 0000 0000

#2 JFM Feedrate units of continuous feeding (06h) of PLC control axis

INCREMENT SYSTEM	JFM	INPUT IN METRIC SYSTEM	INPUT IN INCH SYSTEM	ROTARY AXIS
IS-B	0	1mm/min	0.01inch/min	1deg/min

	1	200mm/min	2.00inch/min	200deg/min
IS-C	0	0.1mm/min	0.001inch/min	0.1deg/min
	1	20mm/min	0.200inch/min	20deg/min

#5 2DSL When selecting the axes controlled by PLC is forbidden, if the axes are tried to exchange

0: Failed and P/S No.139 alarms

1: Axes, without commanding the channel, are executed exchanging

#6 NCI During decelerating the axes controlled by PLC, in-position check is

0: Executed

1: Not executed

	#7	#6	#5	#4	#3	#2	#1	#0
8005							**CDI**	

『Modification authority』: Machine

『Default Setting』: 0000 0000

#1 CDI When PLC control axis selects the diameter programming, under PLC axis control

0: Radius programming specifies the movement distance

1: The diameter programming specifies the movement distance

#2 R10 When the RPD parameter(No.8002#0) is set to "1", the unit for specifying a rapid traverse rate for the PLC axis is

0：×1

1：×10

	#7	#6	#5	#4	#3	#2	#1	#0
8006	**EAL**			**EFD**				

『Modification authority』: Machine

『Default Setting』: 0000 0000

#4 EFD In axis control by PLC, the unit for specifying feed cutting for PLC axis is

0：×1

1：×100

#7 EAL In axis control by PLC, the function that allows the alarm signal to be reset by a CNC reset operation is

0: Not release the alarm of PLC control axis

1: Release the alarm of PLC control axis

8010	**Selecting each axis DI/DO group controlled by PLC（EPAS）**

『Modification authority』：Machine

『Parameter Type』: Word axis type

『Value Range』: 0～4

『Default Setting』: 0

Each DI/DO group controlled by each PLC axis, which is shown as the following list:

NUMERICAL VALUE	**REMARK**
0	The axis is not controlled by PLC
1	DI/DO in group A is used
2	DI/DO in group B is used
3	DI/DO in group C is used
4	DI/DO in group D is used

8022	**Maximum feedrate of feeding/per revolution controlled by PLC axis（EPMF）**

『Modification authority』：Machine

『Parameter Type』: Word axis type

『Value Range』:

INCREMENT SYSTEM	VALUE UNITS	VALID VALUE RANGE	
		IS-B	IS-C
Machine in metric system	1mm/min	6~15000	6~12000
Machine in inch system	0.1inch/min	6~6000	6~6000
Rotary axis	1deg/min	6~15000	6~12000

『Default Setting』: 6

Set the maximum feedrate of feeding/per revolution controlled by PLC axis.

8028	**For each PLC control axis, the linear acceleration or deceleration time constant specified by speed command during JOG feeding（EPAT）**

『Modification authority』：Machine

『Parameter Type』: Word axis type

『Value Range』: 0~3000ms

『Default Setting』: 100

Specify the linear acceleration or deceleration time constant during JOG feeding

Note: If it is set to "0", the system doesn't control the acceleration and deceleration.

8030	Shift of reference position for PLC controlled axes (RPS)

『Modification authority』: Machine

『Parameter Type』: Word axis type

『Value Range』: -99999999～99999999

『Default Setting』: 0

Set the shift of reference position for PLC controlled axes

Appendix 1.28 Parameter of the Basic Function

8130	Total quantity of the controlled axes (TCA)

『Modification authority』: System

『Validate method』: After power-on

『Value Range』: 2～6

『Default Setting』: 2

Set the total quantity of the axes controlled by CNC system.

	#7	#6	#5	#4	#3	#2	#1	#0
8131								HPG

『Modification authority』: Machine

『Validate method』: After power-on

『Default Setting』: 0000 0001

#0 HPG Whether use MPG feeding

0: Not use

1: Use

	#7	#6	#5	#4	#3	#2	#1	#0
8132							YOF	TLF

『Modification authority』: Machine

『Validate method』: After power-on

『Default Setting』: 0000 0000

#0 TLF Whether use the tool work life management function

0: Not use

1: Use

#1 YOF The Y-axis offset is

0: Not used

1: Used

	#7	#6	#5	#4	#3	#2	#1	#0
8133						**SCS**		**SSC**

『Modification authority』: Machine

『Validate method』: After power-on

『Default Setting』: 0000 0001

#0 SSC Whether use the function of the constant surface speed (G96)control

0: Not use

1: Use

#2 SCS Whether use CS outline control function

0: Not use

1: Use

	#7	#6	#5	#4	#3	#2	#1	#0
8134						**CCR**		

『Modification authority』: Machine

『Validate method』: After power-on

『Default Setting』: 0000 0000

#2 CCR The chamfering/corner R is

0: Not use

1: Use

	#7	#6	#5	#4	#3	#2	#1	#0
8135	**RPTH**					**NSQ**		

『Modification authority』: Machine

『Validate method』: After power-on

『Default Setting』: 0000 0100

#2 CCR The program restarting function is

0: Used

1: Not used

#7 RPTH Whether use the thread recovery function

0: NO

1: YES

#7	#6	#5	#4	#3	#2	#1	#0

8136					**NOP**			

『Modification authority』: Machine

『Validate method』: After power-on

『Default Setting』: 0000 0000

#3 NOP Whether use the soft machine tool panel

0: NO

1: YES

Appendix 1.29 Parameter for Slopping Axis Control

	#7	#6	#5	#4	#3	#2	#1	#0
8200						**AZR**		**AAC**

『Modification authority』: Machine

『Validate method』: After power-on

『Default Setting』: 0000 0000

#0 AAC Whether perform the slopping axis control

0: NO

1: YES

#2 AZR When performing the slopping axis manual reference point return in its axis control method

0: Perpendicular axis is also moved at the same time

1: Perpendicular axis does not move

	#7	#6	#5	#4	#3	#2	#1	#0
8209								**ARF**

『Modification authority』: Machine

『Validate method』: After power-on

『Default Setting』: 0000 0000

#0 ARF Move to the reference point from the intermediate pont specified by G28/G30 based upon the slopping axis control:

0: The motion of the slopping coordinate system

1: The motion of Cartesian coordinate system

8210	**The slopping angle (INA) in the slopping axis control**

『Modification authority』: Machine

『Validate method』: After power-on

『Value Range』: -1800000~1800000

『Default Setting』: 0

This parameter sets the slopping axis angle in its axis control

Setting unit: IS-B 0.001deg; IS-C 0.0001deg.

8211	**The slopping axis number (ANS) for performing the slopping axis control**
8212	**The rectangular axis number (ANC) for performing the slopping axis control**

『Modification authority』: Machine

『Validate method』: After power-on

『Value Range』: 0~6

『Default Setting』: 0

This parameter sets the slopping axis number when the slopping axis is controlled. When one of any parameters sets to 0, alternatively, either the same numbers are set or non-control axis number is set, which means that the function is disabled.

Appendix 1.30 Parameter of GSKLink Communication Function

	#7	#6	#5	#4	#3	#2	#1	#0
9000							**GCRC**	**GNET**

『Modification authority』: Machine

『Validate method』: After power-on

『Default Setting』: 0

#0 GNET Whether the system GSKLink communication function is enabled

0: NO

1: YES

#1 GCRC Whether the system GSKLink communication data is performed the verification

0: NO

1: YES

Appendix 2 Standard PLC Function Configuration

Appendix 2.1 Standard Panel on the Machine Tool

Appendix 2.1.1 GSK988TA1 Standard Panel on Machine Tool

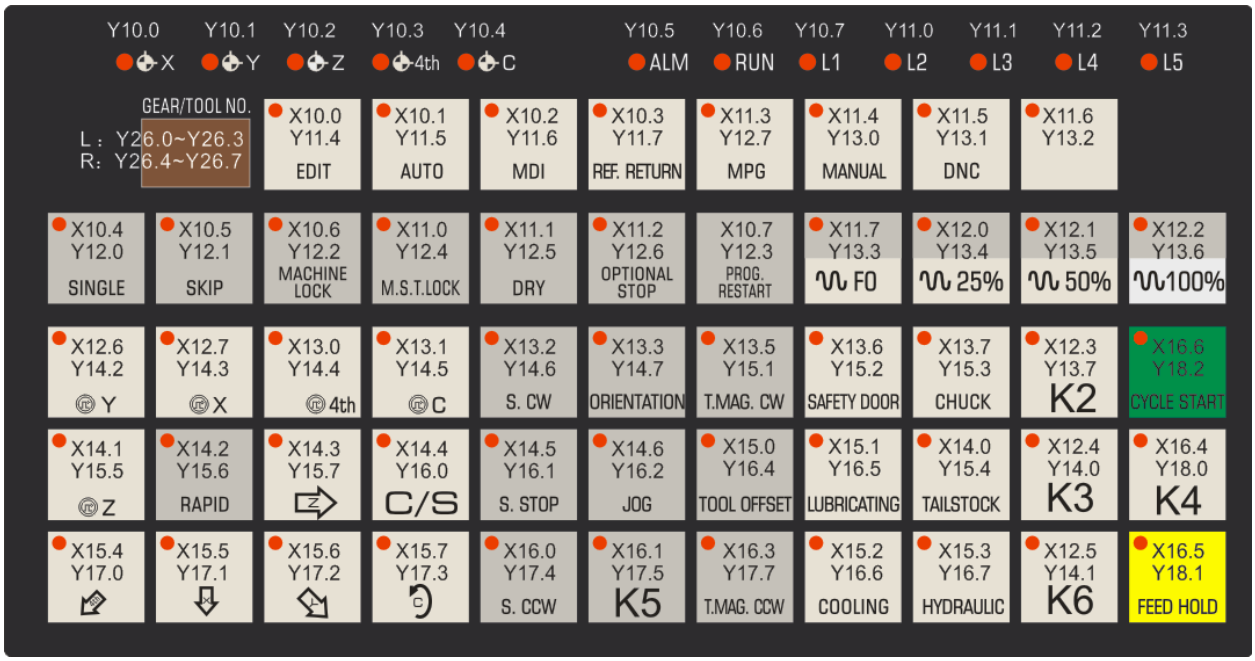


Fig 2-1 GSK988TA1 Standard layout of operation panel

Note: It is the same size between GSK988TA1-H and GSK988TA1 about the address of Standard Panel

Appendix 2.1.2 GSK988TA Standard Panel on Machine Tool

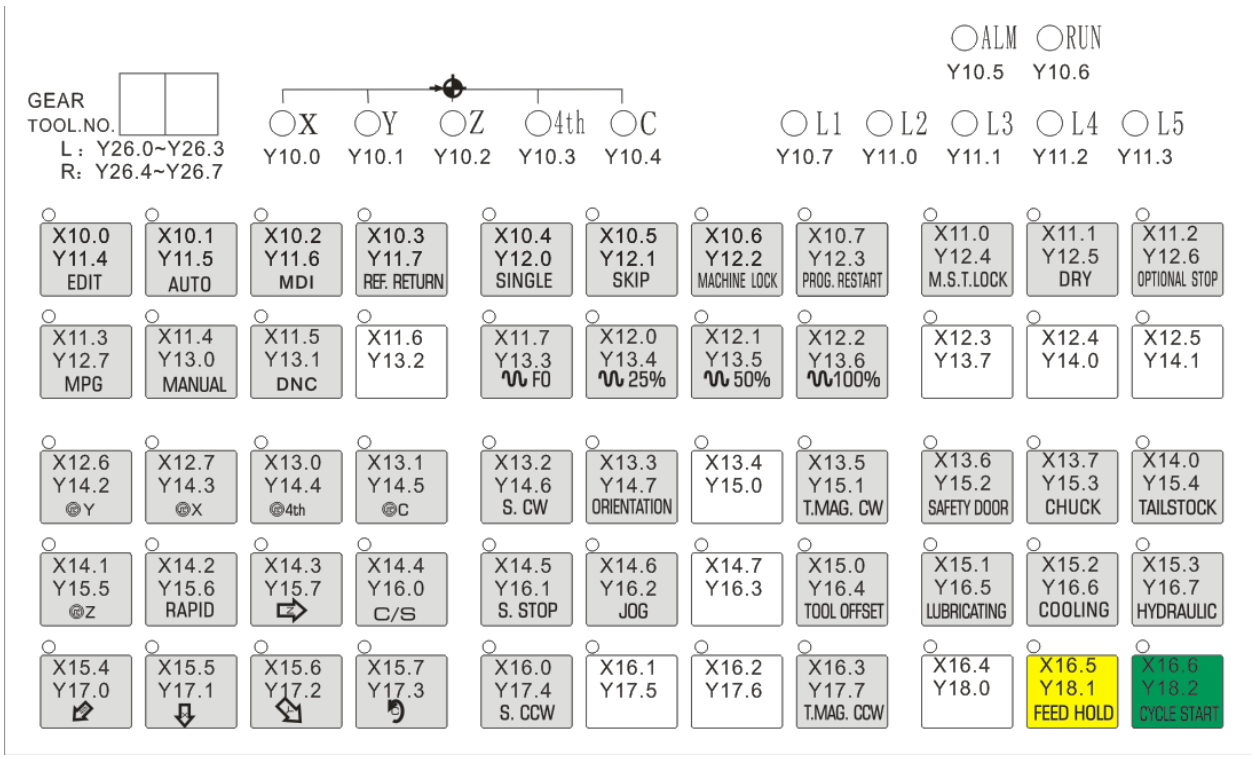


Fig.2-2 GSK988TA Standard layout of operation panel

Appendix

Appendix 2.1.3 GSK988TA-H Standard Panel on Machine Tool

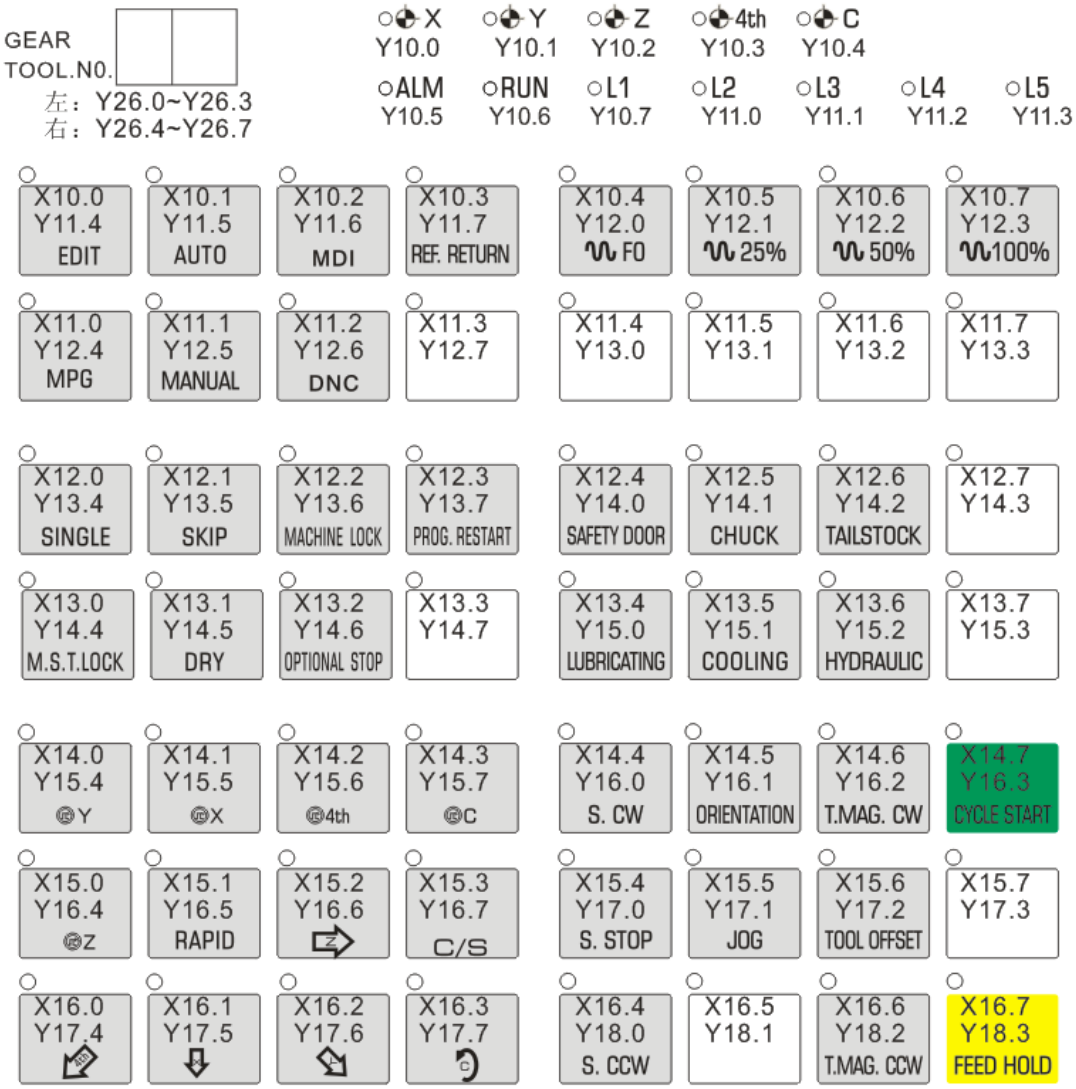


Fig. 2-3 GSK988TA-H Standard layout of operation panel

Appendix

Appendix 2.1.4 GSK988TB Standard Panel on the Machine Tool

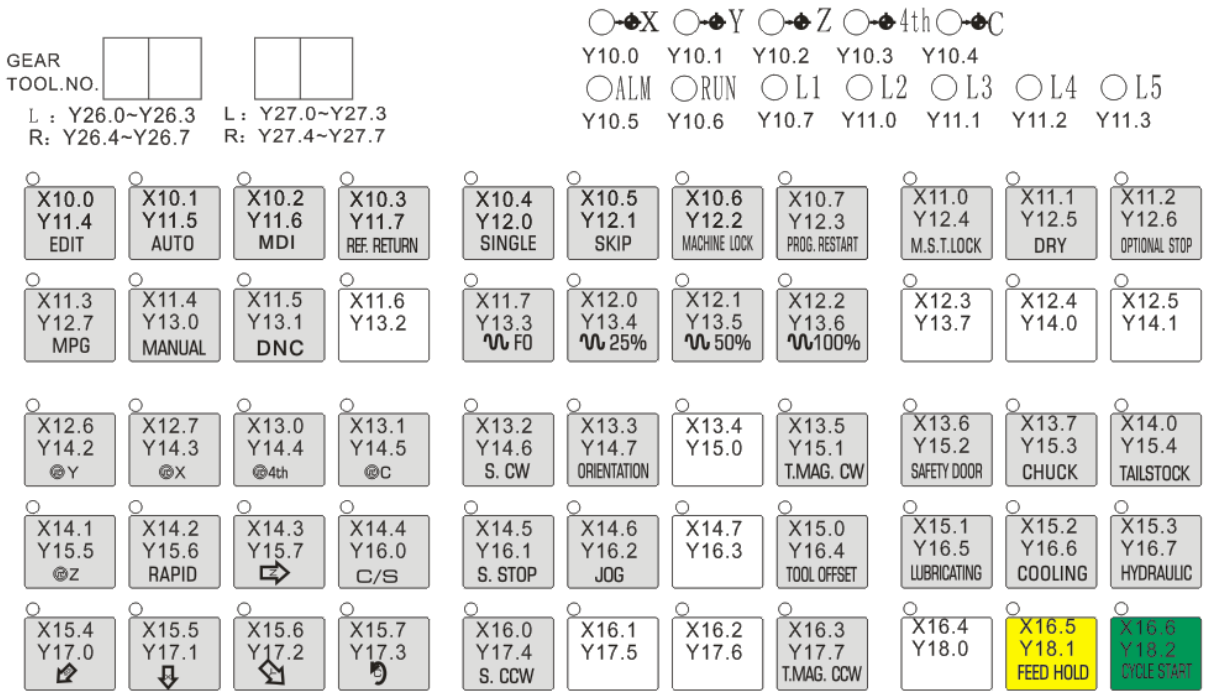


Fig. 2-4 988TB Standard layout of operation panel

Note: It is the same size between GSK988TB-H and GSK988TB about the address of Standard Panel

Appendix 2.2 Definitions of X and Y Addresses of the Ladder Diagram

I/O of GSK988TA/988TA1/988TB is classified into high speed I/O signal and the common I/O one. The high speed I/O signals are those of CN61 on CNC back cover. The common I/O signal is the extension signals of the remote I/O unit. The function of I/O signal of CNC (except for the signal of the marked fixed address) is defined by the system internal PLC program (the ladder diagram). When GSK988TA/988TA1/988TB turning machine CNC is configurated with the machine, I/O function is set by the machine manufacturer; please refer to the user manual for the machine manufacturer about the details.

In this chapter , please pay attention that the common I/O singals（X and Y addresses）function is mainly for the starndard PLC program of GSK988TA/988TA1/988TB.

Appendix 2.2.1 High speed I/O interface

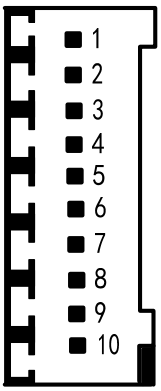


Fig. 2-5 Pins of CN61

Corresponding connector pins	PLC address	Function defined by the standard PLC addresses		Note
CN61.1	0V			
CN61.2	X0.0		Reserved	
CN61.3	X0.1		Reserved	
CN61.4	X0.2		Reserved	
CN61.5	X0.3		Reserved	
CN61.6	X0.4	SKIP	G31 skip signal	
CN61.7	X0.5	ESP	Emergency stop input signal	
CN61.8	X0.6	G36	G36 skip signal	
CN61.9	X0.7	G37	G37 skip signal	
CN61.10	0V			

Appendix 2.2.2 Common machine I/O interface

The all-purpose output/input of the GSK988TA/988TA1/988TB is distributed by connecting the I/O unit with the GSKLink. Up to 4 I/O units can be connected by GSKLink, up to 48 input points and 32 output points of each I/O unit, and the address use range is X80~X127 and Y80~Y127 of which the user should configure the system address by herself/himself.

An I/O unit with 48 input points and 32 output points is configured in the standard configuration of the GSK988TA/988TA1/988TB system. The configuration address of standard ladder diagram in the system is X100~X105 and Y100~Y103. The overall I/O introduced in this User Manual, however, if the difference condition occurs, refer to the User Manual offered by machine tool manufacturer. The signal addresses are subject to it.

PLC address	Function defined by standard PLC address		Remark
X100.0	SAGT	Protection door detection signal	
X100.1		Reserved	
X100.2	DIQP	Chuck input signal	
X100.3		The 1st axis deceleration signal	
X100.4	DITW	Tailstock control signal	

PLC address	Function defined by standard PLC address		Remark
X100.5		Emergency stop input signal	
X100.6	PRES	Pressure detection signal	
X100.7	T05	Tool position signal 5/ tool post pre-indexing signal (Yantai AK31)/Sensor E (Liuxin Tool Post)	
X101.0	T06	Tool position signal 6/ tool post pre-indexing signal (Yantai AK31)/Sensor F (Liuxin Tool Post)	
X101.1	T07	Tool position signal 7/ tool post overheat signal (Yantai AK31)	
X101.2	T08	Tool position signal 8	
X101.3		Reserved	
X101.4		Reserved	
X101.5	M41I	The 1st gear stage in-position	
X101.6	M42I	The 2nd gear stage in-position	
X101.7	T01	Tool position signal 1/T1 (Yantai AK31)/Sensor A (Liuxin Tool Post)	
X102.0	T02	Tool position signal 2/T2 (Yantai AK31)/ Sensor B (Liuxin Tool Post) Sensor A (Liuxin Tool Post)	
X102.1	T03	Tool position signal 3/T3 (Yantai AK31)/Sensor C (Liuxin Tool Post)	
X102.2	T04	Tool position signal 4/T4 (Yantai AK31)/Sensor D (Liuxin Tool Post)	
X102.3		Reserved	
X102.4		Reserved	
X102.5		Reserved	
X102.6	TCP	Tool post lock signal Tool post proximity switch signal (Yantai AK31)	
X102.7		Reserved	
X103.0	LMI1+	The 1st axis + side overtravel signal	
X103.1	LMI2+	The 2nd axis + side overtravel signal	
X103.2	LMI3+	The 3rd axis + side overtravel signal	
X103.3	WQPJ	Chuck in-position signal (outer chuck clamping and inner chuck unclamping)	
X103.4	NQPJ	Chuck in-position signal (inner chuck clamping and outer chuck unclamping)	
X103.5		Reserved	
X103.6		Reserved	
X103.7		Reserved	
X104.0	LMI1-	The 1st axis – direction overtravel signal	
X104.1	LMI2-	The 2nd axis – direction overtravel signal	
X104.2	LMI3-	The 3rd axis – direction overtravel signal	

PLC address	Function defined by standard PLC address		Remark
X104.3	LMI4+	The 4th axis + direction overtravel signal	
X104.4	LMI4-	The 4th axis - direction overtravel signal	
X104.5	LMI5+	The 5th axis + direction overtravel signal	
X104.6	LMI5-	The 5th axis - direction overtravel signal	
X104.7		Reserved	
X105.0		Reserved	
X105.1		Reserved	
X105.2		Reserved	
X105.3		Reserved	
X105.4		Reserved	
X105.5		Reserved	
X105.6		Reserved	
X105.7		Reserved	
Y100.0	M08	Cooling output signal	
Y100.1	M32	Lubrication output signal	
Y100.2		the hydraulic station output signal	
Y100.3	M03	Spindle CCW signal	
Y100.4	M04	Spindle CW signal	
Y100.5	M05	Spindle stop signal	
Y100.6	M35	Spindle hold output signal	
Y100.7	SPZD	Spindle braking output signal	
Y101.0	M41	Spindle gear 1 output signal	
Y101.1	M42	Spindle gear 2 output signal	
Y101.2	M43	Spindle gear 3 output signal	
Y101.3	M44	Spindle gear 4 output signal	
Y101.4	M12(DOQPJ)	Outer chuck clamping output / Inner chuck unclamping output signal	
Y101.5	M13(DOQPS)	Outer chuck unclamping output /inner chuck clamping output signal	
Y101.6	TL+	Tool post forward rotation output signal	
Y101.7	TL-	Tool post reverse rotation output signal	
Y102.0		Tool post motor braking signal (Yantai AK31)/ tool post unclamping output (Liuxin Tool Post)	
Y102.1		Tool post pre-indexing electromagnet signal (Yantai AK31)/ Tool post lock output (Liuxin Tool Post)	
Y102.2	YLAMP	Tri-colored lamp – yellow (normal state, non-running, non-alarm)	
Y102.3	GLAMP	Tri-colored lamp – green (running state)	
Y102.4	RLAMP	Tri-colored lamp – red (alarm state)	

PLC address	Function defined by standard PLC address		Remark
Y102.5	M10	Tailstock advancing output signal	
Y102.6	M11	Tailstock retracting output signal	
Y102.7		Reserved	
Y103.0	M37	Chip	
Y103.1	M38	Chip	
Y103.2		Reserved	
Y103.3		Reserved	
Y103.4	SORI	Spindle orientation signal	
Y103.5	SEC0	Spindle orientation selection signal 1	
Y103.6	SEC1	Spindle orientation selection signal 2	
Y103.7	SEC2	Spindle orientation selection signal 3	

Note 1: The addresses of X100.0~X105.7 are the high level input valid, that is to say, when the input signal is connected with +24V, X address signal status is 1, otherwise, the status is 0.
Note 2: When Y address signal status output by low level is 1, the output signal is connected with 0V (0V output); when Y address signal status is 0, the output signal is high resistance.
Note 3: When Y address signal status output by high level is 1, the output signal is connected with 24V (24V output); when Y address signal status is 0, the output signal is high resistance.

Appendix 2.2.3 Interface of the Handhold Box

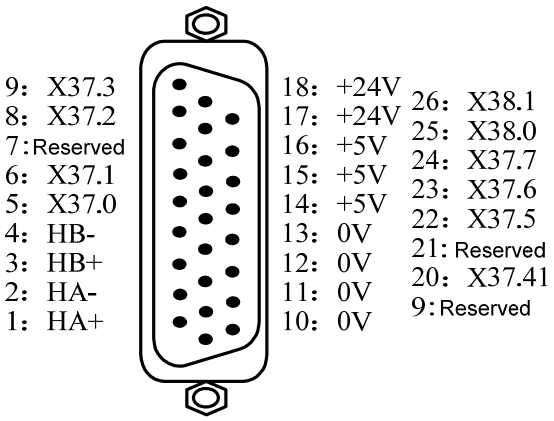


Fig.2-6 CN32 MPG interface
（Pin socket of type D in 26 cord）

DB Pin	Signal Definition	Signal Instruction	Function defined by standard PLC address
CN32.1,CN32.2	HA+，HA-	MPG phase A signal input	/
CN32.3,CN32.4	HB+，HB-	MPG phase B signal input	/
CN32.5	X37.0	PLC signal address, switch amount input	External MPG box X axis selection signal
CN32.6	X37.1	PLC signal address, switch amount input	External MPG box Y axis selection signal
CN32.8	X37.2	PLC signal address, switch amount input	External MPG box Z axis selection signal
CN32.9	X37.3	PLC signal address, switch amount input	External MPG box ×1 gear signal
CN32.22	X37.5	PLC signal address, switch amount input	External MPG box ×10 gear signal
CN32.23	X37.6	PLC signal address, switch amount input	External MPG box ×100 gear signal
CN32.24	X37.7	PLC signal address, switch amount input	External MPG box ×X1000 gear signal
CN32.25	X38.0	PLC signal address, switch amount input	External MPG box the 4th axis selection signal
CN32.26	X38.1	PLC signal address	External MPG box the 5th axis selection signal
CN32.20	X37.4	PLC signal address	External MPG box the 6lx axis selection signal
CN32.10，CN32.11 CN32.12，CN32.13	0V	0V	/
CN32.14，CN32.15 CN32.16	+5V	+5V	/
CN32.17,CN32.18	+24V	+24V	/

Note: When X37.0~X38..0 as high level input are valid, that is to say, when the input signal is connected with +24V, the input is valid, and X address status is 1; otherwise, X address status is 0.

Appendix 3 Interface Explanation

Appendix 3.1 CNC Rear Cover Interface Layout

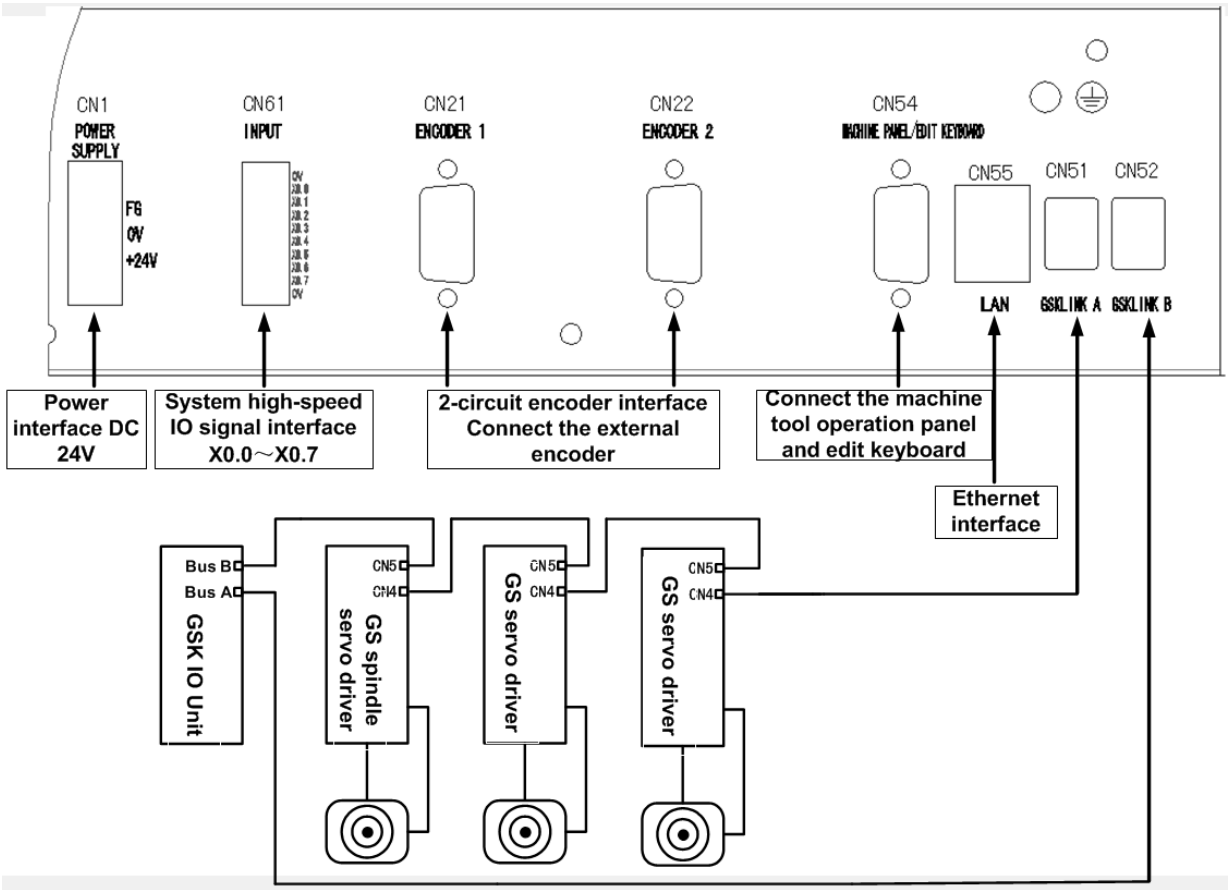
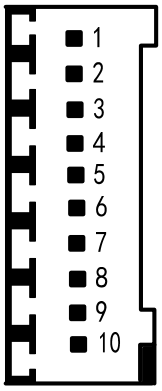


Fig.3-1

Appendix 3.1.1 High Velocity Input Interface CN61

GSK988TA/988TA1/988TB system equips with the high velocity I/O interface CN61 of 1 input signal, its address is X0.0~X0.7

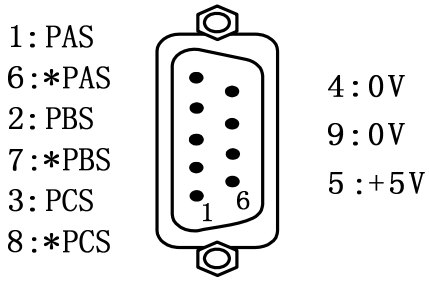
Input port CN61	CN61 pin No.	PLC add.
	1	GND
	2	X0.0
	3	X0.1
	4	X0.2
	5	X0.3
	6	X0.4
	7	X0.5
	8	X0.6
	9	X0.7

Appendix

CN61	10	GND

Appendix 3.1.2 Encoder Interface CN21 and CN22

GSK988TA/988TA1/988TB owns two-circuit encoder input interface (N21, CN22), refer to the Fig. 3-2.。



（9-core D-male socket）

Fig.3-2

Appendix 3.1.3 Communication Interface CN54

GSK988TA/988TA1/988TB system and machine operation panel are connected with the communication. Refer to the Fig.3-3 for the interface pin

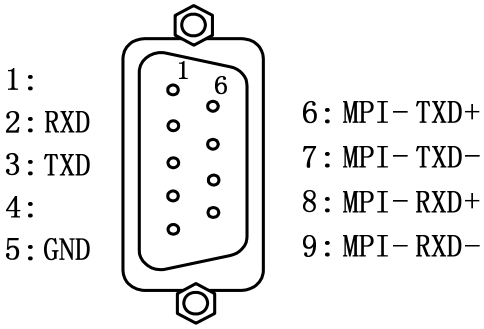


Fig.3-3

Appendix 3.1.4 Network Interface CN55

The system uses standard general cable interfaces.

Appendix 3.1.5 Standard interface

The bus interfaces of GSK988TA/988TA1/988TB are CN51 and CN52 (GSKLinkA and GSKLinkB), the interfaces are possessed feed servo drive unit with GSKLink bus communication function, spindle drive unit and extension I/O unit communication connection.

GSKLink bus communication connection cable is shown below:

A1	TX−		A1	TX−
A3	TX+		A3	TX+
A2	RX−		A2	RX−
A4	RX+		A4	RX+
	Outmost layer shielding cable			Outmost layer shielding cable

Fig.3-4 GSKLink communication connection

Appendix 3.2 Rear Cover Interface of Machine Tool Operation Panel

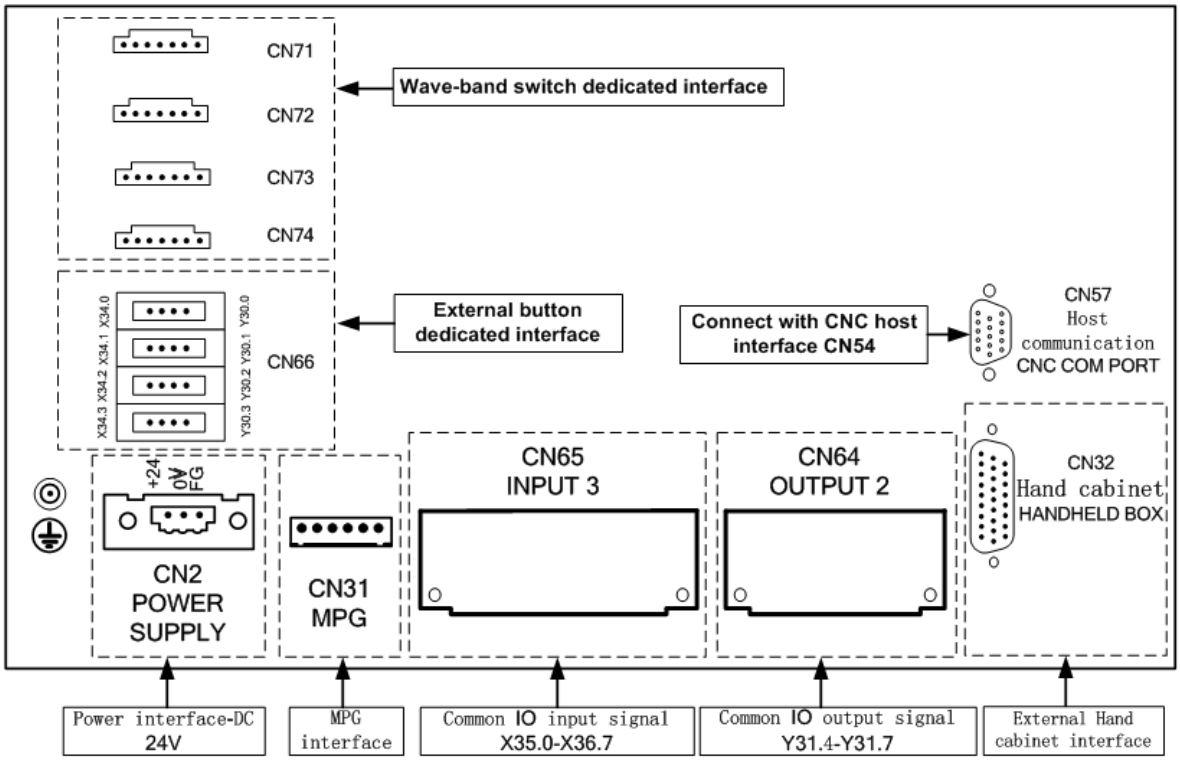


Fig.3-5

Appendix 3.2.1 Dedicated Wave Band Switch Interface

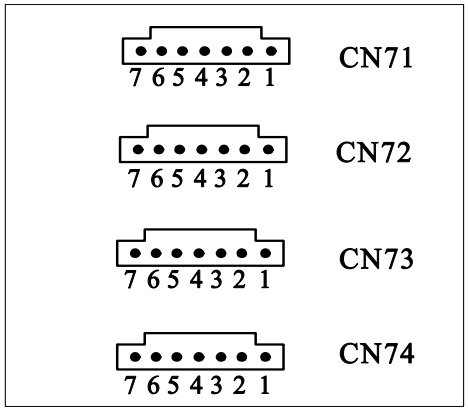


Fig.3-6

Appendix 3.2.2 Dedicated Interface of The External Button CN66

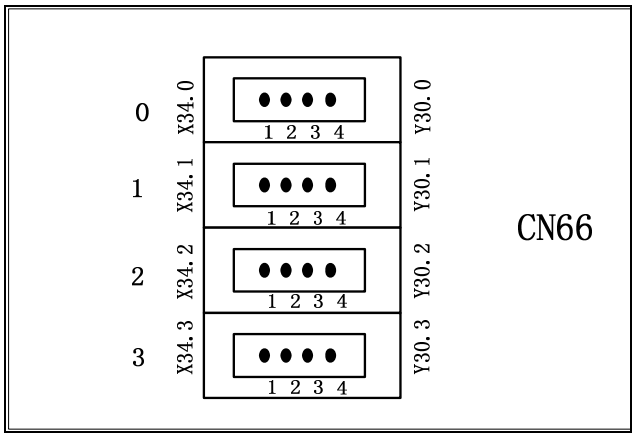
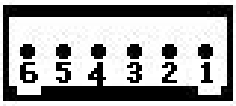


Fig.3-7

Appendix 3.2.3 MPG Interface CN31 and CN32

MPG Interface CN31



6：VCC 5：HA- 4：HA+ 3：HB- 2：HB+ 1：GND

Fig.3-8

Hand cabinet interface CN32

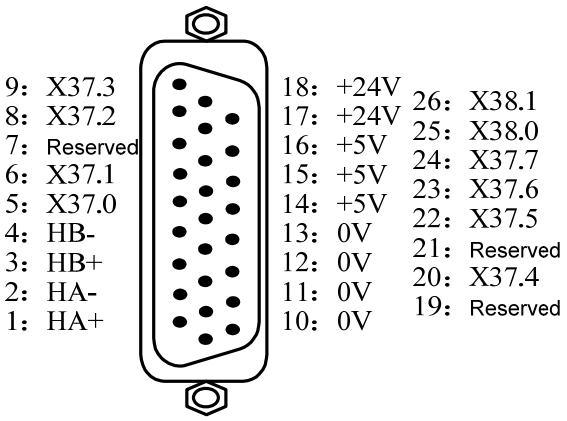


Fig.3-9 CN32 MPG interface（26-core type D pin socket）

Appendix 3.2.4 Communication Interface CN57

GSK988TA/988TA1/988TB system and machine tool operation panel are adopted the communication connection method.

Pin No.	Signal	IN/OUT	Explanation
1	RXDA	IN	Accept the data difference signal
2	RXDB	IN	Accept the data difference signal
4	TXDA	OUT	Deliver the data difference signal
5	TXDB	OUT	Deliver the data difference signal

Appendix 4 Alarm Troubleshooting

Appendix 4.1 CNC Common Alarm Remedy

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
000	Emergency stop, ESP open circuit	1. Whether the ESP button is controlled	Modify the parameter or check the connection
		2. Incorrect wiring	
		3. The setting of bit 7 of parameter 3003 (ESP) is inconsistent with the actual connection.	
		4. The setting of parameter K10.7 is inconsistent with the actual connection.	
001	Part program open failure	Program is not downloaded before the running in AUTO mode.	Reset to clear alarm and re-execute the program
002	Part prog. segment loading failure	Error found in loading segment when executing MDI prog. or checking syntax	
010	Single block exceeds 256 characters	Characters excessive in single block	
011	Data exceeds permissive range	Input data exceeds permissive range or the specified data exceeds 8 digits	
012	Address not found	With number or symbol other than address at the beginning of a block	
013	No data follows address	No data follows address or expression format following address checks error, without brackets	
014	Illegal use of negative sign	Sign "-" was input after an address with which it can't be used, or two or more "-" was input	
015	Illegal use of decimal point	Decimal point"." was input after an address with which it can't be use, or two or more "." was input	
016	Input illegal address	Input unusable address in significant area	
017	Incorrect G code	Specify improper G code or that with functions not provided	
018	Address repetition error	Specify the same address twice or more in a block Or specify two or more G codes in same group in a block. Refer to para. 3403#6 AD2.	

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
019	End of record	Specify end symbol(%) of record, or not specify end of program, referring to para.3404#6 EOR	
040	Too many M codes	Count of M codes specified in a segment exceed value of para.3404#7 M3B or 3	
041	Number followed M code out of range	Digits of M code exceed the value of para.3030 MCB	
042	G code specified error with M99	G28, G30, G53, G36, G37 can't be specified with M99 in a segment	
050	Illegal tool No.	Specify a tool No. which doesn't exist, or exceeds the value of para. 3032 TCB	
051	Compensation No. not found	Tool offset compensation number exceeds range by T code (0~99)	
052	Illegal T code in block	G10, G04 doesn't work with G50(Group A) or G92(Group B)in same block. Refer to the parameter 5006#1 TGC	
060	Feedrate exceeds range	Feedrate was not specified or exceeds range:1.Check G98 or G99 state for feedrate difference of usage in Group A 2.Check G94 or G95 state for feedrate difference of usage in Group B	
062	Illegal G96 code was found	G96 was specified while const-surface-speed control function is not performed with reference to param.8130#0	
063	Axis specified error in constant surface speed control	In G96 modal, the specified axis by parameter 3770 is wrong	
070	Command can't run in MDI mode	Command cannot run in MDI mode, for example, G36/G37, G70~G73	
071	DNC time out	DNC transmission failure, please check	
075	Axes type specified error	Axis type is invalid for specifying; Check the setting of the corresponding parameters 1006#0 and 1006#1 by this axis	

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
076	Illegal rotation axis for interpolation instruction	An rotation axis can't be specified except for in polar coordinates, cylinder interpolation, G00, G01 mode	
077	The specified axis is the simple controllable synchronization axis	The specified axis is set to simple synchronization controllable axis in parameter 8311.	
080	Property error for basis axes of plane	Property set error for basis axis of plane in radius interpolation	
081	Illegal rotation axes specified in circular interpolation	Modify the program or check the setting of parameters 1006#0 and 1006#1	
082	No radius and I/J/K commanded	In circular interpolation, R, I, J, K has not been specified, referring to para. 3403#5 CIR	
083	Illegal radius	In circular interpolation, Destination is not on the arc specified by R, referring to para.3403#4 RER	
084	Over tolerance of radius	In circular interpolation, difference of the distance between start point and the center of an arc and that between end point and the center of an arc exceeded setting value ,referring to para.3410 CRE	
085	Axes too much specified in circular interpolation	In circular interpolation, more than 3 axes specified	
086	Three-point arc command data error	There is no instruction in the three-point arc intermediate point instructions, or mid-point instruction can not constitute an arc.	
087	Three-point arc command data can't used to determine full arc.	Three-point arc command can not process full circle, the instruction must be specified end	
088	three-point arc command data error	1.The start, end, mid point shouldn't be on the same line, or start, end point is the same 2. The radius is 0 determined by end, mid point.	
096	Address P or X out of range	Dwell time specified by P exceeds 0~99999999, or X exceeds -9999~9999	

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
100	Chamfering amount, J was specified error in thread cutting commands	Value of J address exceeds permissive range, or the number followed J is less than zero in G92, G76	
101	Chamfering amount K was specified error in thread cutting commands	Chamfering amount specified by K is less than zero, or exceeds permissive range	
102	Value of L out of range in multi-threading	The value specified in L exceeds the range(1~99) in Multi-threading	
103	Illegal lead command	Lead specified by F is out of range	
104	Value of R out of range in variable threading	In variable threading, the lead incremental and decremental specified by R exceeded permissive range	
105	Chamfering amount too large of long axis in threading	Chamfering amount of long axis was greater than thread length; alternatively, the long axis end-retraction value calculated by leading F and parameter 5130 is excessive big in the G92/G76 command.	
106	Chamfering amount too large of latitude axis in threading	Chamfering amount of latitude axis in G92 was greater than the distance between start point and end point	
107	Axes not in selected plane in threading	Specify the axis out of the selected panel in thread command.	
108	Illegal axes for interpolation in threading	In threading, basis and parallel axes are both specified, or more than 2 parallel axes are specified	
109	C axis not exist in rigid threading	C axis is not set in parameter No.1020 for rigid threading	
110	C axis is not rotation type in rigid threading	C axis is not rotation type in rigid threading, Refer to parameter No.1006#0 and 1006#1.	
111	Spindle speed S not specified in rigid threading	S address was not specified for rigid threading command G32.1	
130	Illegal plane select	In the plane selection command, two or more axes in the same direction are selected	
131	Illegal basic axis specified for selected plane		

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
140	Metric/inch conversion command not at the beginning of the program	Metric/inch conversion command must at the beginning of the program	
141	Metric/inch conversion command not at a single block	G20/G21 metric/inch conversion can not be shared a same block with other G commands, which should be specified separately.	
142	G20/G21 metric/inch conversion can not be specified in sub-program	The metric/inch conversion is performed during calling the sub-program.	
150	Improper code in the same block with G22	The G and MSTF code can't be in the same block with G22	
151	For G22 Data exceeds permissive range	Input data exceeds permissive range,\nOr the specified data exceeds 8 digits	
152	For G22 A stroke limit check inhibited area error	The coordinate of the plus side inhibited area is not greater than that of the minus side inhibited area Or the difference is not greater than 2000 output increment, referring to No.1322 & No.1323	
153	G22 command contains an illegal axes instruction	In G22 instruction , axes other than the basic is commanded or U/V/W is used	
154	Axes specified for G22 property error	Instructions the X/Y/Z axis corresponds to the basic property is set to 0 or parallel to the axis	
160	G code in the same block with G25/G26	Specify G code of other group with G25/G26	
165	Reference point not established of axes	Reference point haven't been established before cycle start with reference to param.1005#0 ZRNx	
166	The axis does not turn to reference point while G28 was specified	The reference point does not set up before performing G28, it is better to modify the program or the parameter 1002#3 AZR.	
167	The axis does not turn to reference point while G30 or G53 was specified	The axis does not turn to reference point while G30 or G53 was specified，Please establish reference point	
168	Illegal reference	Address P specifies other values	

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
	point by P address in G30	than 2~4 in G30	
169	Mid point of axis out of range for reference position return in G28/G30	In G28/G30, the position of mid-point is out of range	
180	Illegal offset value L/P for G10	1.In setting an offset amount by G10, neither L nor P is specified 2. In setting an offset amount by G10, the offset value specified by P is excessive or not specified.	
181	Command address not match in programmable parameter input function	G10/G11 is not match,G10 is specified duplicated, or G11 is specified while G10 is not specified	
182	Programmable parameter input function not canceled	Programmable parameter input function is not canceled by G11 before program ends	
183	Illegal command in Entering data from program	The NC commands, such as the axis address, G code or MSF, etc. are specified in programmed data input.	
200	P value out of range for G31	P value of G31 is beyond 1～4	
201	G31 not allowed in G99	Both basic axes and parallel axes are specified, or more than 2 parallel axes for a basic axis are specified	
202	G31 not allowed in tool radius compensation mode	In tool nose radius compensation mode, specify skip cutting command	
210	Illegal G36/G37 specified in auto tool compensation	Illegal G36/G37 specified in auto tool compensation	modify param.6240#7 IGA or modify the program
211	Offset number not found in G36/G37	Auto tool compensation(G36\G37) was specified without T code	
212	T code not allowed in G36/G377	T code and auto tool compensation(G36, G37) was specified in the same block	
213	Illegal axis command in G36/G37	In auto tool compensation function(G36,G37),an invalid axis is specified	
214	Illegal axis command in G36/G37	Axis specified to move is not the corresponding axis in G36, G37 or the command is incremental	

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
215	Para.error in G36/G37	γ is less than ε.Refer to param. 6251 ATOR1, 6254 ATOE1, 6252 ATOR2, 6255 ATOE2	
216	ATC not allowed in tool radius compensation mode in G36/G37	Auto tool compensation(G36, G37) was specified in tool radius compensation mode	
230	Illegal command in the same block with G7.1	In circular interpolation G7.1（G107）block，other group of G codes or MST is specified	
231	None-rotation axis specified to start circular interpolation	None-rotation axis is specified within G7.1(G107) block	
232	Too many rotation-axes specified in circular interpolation	Too many rotation-axes is specified with G7.1(G107) in block	
233	Illegal negative sign of radius specified in circular interpolation	Illegal negative sign of radius is specified with G7.1(G107) in block	
234	Illegal G12.1, G51.2 found in circular interpolation	In circular interpolation mode, it is illegal to specified polar interpolation command G12.1,or polygon processing command G51.2	
235	Illegal change-plane command found in circular interpolation	In circular interpolation, it is illegal to specified G17~G19 to change plane	
236	Illegal change-workpiece-coordinate command in circular interpolation	In circular interpolation, it is illegal to specify G54~G59 to change workpiece coordinates	
237	Illegal multi-cycle command specified in circular interpolation	In the column interpolation method, G54~G59 can not be specified to perform the workpiece coordinate system selection	
238	Illegal tapping, drilling command specified in circular interpolation	Illegal tapping, drilling command is specified in circular interpolation by G84~G89	
239	Illegal canned-cycle command specified in circular interpolation	Illegal canned-cycle command specified in circular interpolation G90~G94	

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
240	Illegal threading command specified in circular interpolation	Illegal threading command specified in circular interpolation by G32 or G34	
241	Illegal axis of other plane specified in circular interpolation	Illegal axis of other plane is specified in circular interpolation	
242	In circular interpolation mode, G code in Group 00 other than G04 is specified	In circular interpolation mode,G code in Group 00 other than G04 is can't be specified, including G27~G30,G31,G36/G37 ,G52 ,G 53	
243	Illegal G code or T code specified in circular interpolation	In circular interpolation, it is illegal to specify G00, or T code	
244	Specify radius by I/J/K in cylinder circular interpolation	Specify radius by I/J/K in cylinder circular interpolation	
245	Improper code in cylinder circular interpolation in tool compensation C type	When in G41 or G42:specify improper G-code to start or end circular interpolation	
250	Para.error for polar interpolation	When the polar coordinate interpolation command G12.1 (G112) is performed, the setting of the corresponding parameter 5460 (linear axis) or parameter 5461 (revolving axis) by polar coordinate interpolation axis is detected.	
251	C-type tool compensation error in polar interpolation	When C-type compensation is performed(None-G40-Modal),it is illegal to specify polar interpolation by G12.1/G13.1	
252	Illegal axes specified for selected plane in polar interpolation	In the polar coordinate interpolation method, the axis of the arc command is out of the selected panel.	
253	Other group of G code in the same block with G12.1/G13.1	Specify the G command of other groups in the polar coordinate interpolation command G12.1/G13.1 (G112/G113) block.	
254	Repetition of G12.1 command.	G112 already performed while command another G112	
255	Illegal T instruction found for polar interpolation	Specify T code which can't be used in polar interpolation.	
256	Illegal G code specified in polar	In polar interpolation, only G code below is proper: 1.Group	

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
	interpolation	00:G04 or G65. 2.Group 01:G01, G02 or G03. 3. Group 03(G98/G99 in A-type-g code, G94/G95 in B-type-g code). 4. Group 05:G40~G43, and Group 09:G66/G67.	
270	G50 is invalid	Alarm when specify coordinate set command(G50 in A-type-G code,G92 in B-type-G code),refer to Para.1202#2	
271	Value to set the coordinate system is out of range	Refer to the valid range of G50(G92 in B-type-G code)/G52/G53/G54~G59	
272	Address P out of range for Append coordinate	Address P to select append coordinate is out of range in G54.1	
280	Address P not defined	Address P(program number) was not commanded in block including M98, G65 or G66	
281	Subprogram nesting error	The subprogram call exceeds 12 folds	
282	Program number or sequence number not found	The program number was not found specified by P in M98, M99, G65 or G66.	
283	Fail to open the program during sub-program calling	Sub-program calling; fail to open the sub-program when the internal preread sub-program occurs.	
284	Fail to init nc buffer of sub program	Fail to buffer the initial sub-program when sub-program calls.	
285	Subprogram call error	A program can't call main program or itself in M98,G65 or G66	
286	Subprogram in use	Sub program can't be called because it is in edit state or unsaved state	
287	Program call statement can't run in MDI&DNC operation	Marco program and subprogram call in MDI &DNC operation isn't supported	
300	Illegal G code specified with multi-spindle control	When select spindle in multi-spindle control function, illegal G code is specified with S_P_;S_ and G25/G26, S_P_ and G96/G97 can't be specified in same block	

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
301	P address error in multi-spindle selection	When select spindle in multi-spindle control function, P address assigns an illegal value beyond the range set in Para.No.3781	
302	Absence of P address in multi-spindle selection	In multi-spindle selection, P address with spindle speed command S is absence. The alarm is release when Para.No.3706#2 is set to 0	
303	Error multi-spindle function disabled	1.Para.MPP （ No.3703#3 ） =0,while specify P address to select spindle\n2.Para.MPS(No.3781)= 0，while specify P address to select spindle	
310	Illegal polygon processing command in C-type tool compensation mode	In C-type tool compensation mode(None-G40 mode), while specify polygon processing commandG51.2(251)	
311	G51.2 repetition in polygon processing	In the G51.2 polygon machine mode, the command, G51.2, is specified again.	
312	Error found in P address of G51.2	P is not specified or set by a value out of range in G51.2	
313	Error found in Q address of G51.2	Q is not specified or set by a value out of range in G51.2	
314	PQ ratio incorrect	P/Q is not equal to 0 refer to param7603#1.	
315	G51.2 is not performed in polygon processing	When Para.No.7610 is set to 0,polygon processing command G51.2 is not used	
316	Rotation axis set error	The shaft axis of rotation is not set	
317	Conflict code specified in the same block with G51.2 or G50.2	G51.2 or G50.2 is specified in the same block with other G or MT instructions	
318	polygon processing instructions of the screw mosquito instructions	Polygon processing, command a mosquito Lo instructions	
319	polygon processing command axis command is illegal	Polygon processing, command the tool rotation axis move command	
320	polygon rotation axis machining work is no instruction S instruction position mode	Polygon rotation axis of the workpiece machining way to position mode (para. 7603 #4), there is no instruction S command.	

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
321	Illegal command specified in polygon processing	In polygon processing, circular interpolation command G7.1, polar interpolation command G12.1 and tapping/drilling cycle command G84~G89 can't be used	
400	Parameter switch is ON	Press【RESET】to cancel alarm.	
401	Fail to open the component program	The current program is removed or there is no current program.	
402	Parameters back up failure	Check the memory or power-on again	
403	Parameters recover failure.	Check whether parameters are being written in, or power-on and retry	
405	Recovery of system Para. successfully	Please power on again	
406	Recovery of PLC Para. successfully	Please power on again	
407	Recovery of servo Para. successfully	Please power on again	
408	Recovery of units Para. successfully	Please power on again	
409	Fail to import Para.from extern file	Data in imported Para.file detected invalid, and old data was recovered	
430	More than 3 parameters fount out of range	More than 3 parameter data are exceeded the setting range, which are already used the default value.	
450	Parameter is already modified.	.	A parameter which requires the power off was input, turn off power
451	Servo parameter is already modified.		A servo parameter was modified which requires the system & the servo restart
452	Bus communication logic ID number is modified.		Before the device has been restarted, the internal station address will not be effective
453	I/O unit parameter is already modified.		I/O parameter which requires the power off was input, turn off power
460	Number of CNC controllable axes exceeds the total number		Check para. No.1010 and 8130

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
461	Duplicated axis attribution were set		Modify para. No. 1022
462	Duplicated axis name were set	The possible reasons: 1) The same axes names are set; 2) The forbidden axis name is specified at the current G code system.	Modify the parameter No.1020 or No.3401#6.
463	Disabled rotation axis setting	The setting of rotation axis of parameter No.1006 is disabled	
465	Duplicated servo command was set for control-axis		Modify para.No.1023
470	Duplicated servo comm id or analog address was set for spindle		Modify para.No.3717
471	Invalid AO address of I/O unit was set for spindle		Modify para.No.3717
472	Duplicated spindle encoder number was set		Modify para.No.3723
474	Logic id was not set for spindle using Cs contour function		Modify para.No.3717 or 8133#2, 3704#6, 3704#7.
475	,Logic id was not the same between axis and spindle using Cs contour function		Modify para.No.1023, 3717, 8133#2, 3704#6, 3704#7
476	Logic id was the same between axis and spindle while Cs contour function is not used		
477	AO address of spindle beyond the maximum address of the correspond I/O unit		Modify para.No.3717.
480	Duplicated comm id of I/O unit was set for spindle		Modify para.No.3051, No.3052, No.3053, No.3054
490	None of valid comm It was set		Modify para.No.9000#0, or No.1023, No.3717, No.3050, No.3051, No.3052, No.3053, No.3054, or No.3060
491	Custom macro config file changed, restart the system to become effective		The custom macro config file has changed, restart the system to make it effective

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
500	Reference position not established		Manually move to reference return and press "axis move" key on operation panel under "reference return" mode to establish it
501	Encode data error		It is necessary to set up the machine reference point; it may be the reason of the encoder data read error.
504	Servo battery voltage too low		Please replace the servo battery , and then reestablished the reference position
510	The alarm occurs due to the extensive error on the power-on machine coordinate	The error value between the machine tool coordinate establishement and memory with power-on exceeds the tolerance. Reason: 1) The carriager position moves during the machine tool is power-off; 2) The setting value of parameter No.1206 is excessive small.	It is necessary to reset the machine tool reference point or turn the power on again.
511	Machine coordinate initialized error too large	The error value between the machine coordinate establishment and memory with power-on is excessive big. Reason: 1) The carriager position moves during the power-off, turn on the power again. 2) The motor encoder is changed.	
512	The system parameter relevant to the reference point has been altered.	The numerical value is different when the system parameter and the machine reference point establishment are inspected with power-on, which includes parameter No.1811#2 or No.1816 or No.1820	
513	The servo parameter relevant to the reference point has been altered.	The numerical value is different when the servo and the machine reference point establishment are inspected with power-on, which contains of the command reverse parameter or gear ratio parameter.	
514	Fail to read the parameter record file relevant to the reference point	The parameter value can not be checked the previous established machine zero when the power is turned on.	

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
520	PC signal detected error because of servo dislink		Please check communication connection of gsklink, and power on again
604	Servo alarm		Check the servo
650	Power supply to the servo is turned off	The coordinate system became inaccurate when the control command to the servo is interrupted	Please return to the reference position
700	stored stroke limit1 : +	Exceeded the + sides stored stroke limit 1	
701	stored stroke limit1 stored stroke limit1 : -	stored stroke limit1	
702	stored stroke limit2 : +	Exceeded the + sides stored stroke limit; 2. The internal/external inspection is determined by bit 0 of parameter No.1300.	
703	stored stroke limit2 : -	Exceeded the - sides stored stroke limit; 2. The internal/external inspection is determined by bit 0 of parameter No.1300	
704	stored stroke limit3 : +	Exceeded the + sides stored stroke limit 3	
705	stored stroke limit3 : -	Exceeded the - sides stored stroke limit 3	
706	Over travel : +	Exceeds + side overtravel limit	
707	Over travel: -	Exceeds - side overtravel limit	
710	Spindle speed alteration inspection alarm	Actual spindle speed exceeds the allowable range of commanded value	Check the machine tool cutting state, or refer to Para.No.4912 and 4913
720	Error in manual tool offset measurement	Illegal operation including: 1. A couple or more of axes shift has been detected, or no axis shift has been detected while the complete signal input; 2. Direction of axis shift and complete signal detected reversal; 3. Direction of axis shift was not fixed;	

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
		4. Another signal inputted while the last measurement was not completed; 5. At present, the manual tool-setting operation is being operated	
721	Reference position not established for manual tool measurement	Reference position of the axis is not established	Establish the REF. position first and then re-measure
722	Data written error in manual tool measurement	Data write error in the manual tool measurement	Please check the ram
723	Error selecting tool offset number by PLC in manual tool measurement	Tool offset number selection is incorrect by PLC in the manual tool measurement	Please check the PLC
905	Append coordinate established failure	G54.1 append coordinate failed to established	Press 【RESET】to cancle alarm, or power-on again or send it to the factory for inspecting.
910	Initial parameter failure	User parameter file does not exist or data is damaged. Default parameters become effective	
911	Initial CNC configuration failure	CNC config file does not exist or data is damaged. Default configuration becomes effective	
912	Initial tool offset data failure	Tool offset file does not exist or data is damaged. Initial data becomes effective	
913	Initial tool life data failure	Tool life file does not exist or data is damaged. Initial data becomes effective	
914	Initial pitch error compensation data failure	Pitch error compensation file does not exist or data is damaged. Initial data becomes effective	
915	Initial PLC program failure	Read file failure in registering program, or compile failure.	
930	Tool compensation initialization failure	The tool compensation file is not available or the verified error. The backup value is loaded	
931	Parameter file initialization failure	The Parameter file is not available or the verified error. The backup value of parameters is loaded	
940	Data in NVRAM changed	Data version in nvram detected inconsistent with the new version	Data version in nvram detected inconsistent with

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
		in used	the new version in used.\nFor ABS. encoder is used, please re-establish the REF. position; For INC. encoder, please return to REF. position again. Please re-execute the PLC Program and using the default data in plc to recover the data in NVRAM
941	Power off when accessing NVRAM	The system powered off as running	Perform REF. position return, and check the coordinates, tool offset values
942	Data area 1 of NVRAM detected abnormal	Verification of data area 1 of NVRAM is wrong or ruined	Perform REF. position return, and check the coordinates, tool offset values. If this alarm frequently occurs, it is better to maintain it by factory.
943	Data area 2 of NVRAM detected abnormal	Verification of data area 2 of NVRAM is wrong or ruined	If the system uses the absolute encoder, rebuild the reference point. If the system uses the incremental encoder, the zero return operation should be operated again. If the alarm frequently occurs, it is better to maintain by sending to the factory.
950	TRYOUT limit timed out. System functions are restricted.		Please contact the dealer
990	Too many alarm and info	The number of alarm exceeds 20 or number of info exceeds 30.	
991	Undefined alarm No.	Missing alarm content for alarm No.	
992	Format error in alarm content	Part of data in alarm content and operation info was incorrect	
993	PLC alarm information table error	The alarm num specified is not found in PLC alarm information table or out of the range 1000~2999	Modify the PLC alarm information table
4000	Syntax check cancelled because of reset	Reset causes incompleted the syntax check, referring to para 3401#2 NCK	Please check again

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
4001	Same sequence number found in syntax check	There are duplicated sequence number, which might cause error	Modify the program
4010	Value of some parameter out of range	This may be caused by system updating, or switching of Para.ISB\ISC, or switch of linear axis and rotation axis	
4011	GSKLink function is not used,while valid station address was set		Modify para No.9000#0,or No.1023, No.3717, No.3050, No.3051, No.3052, No.3053, No.3054, or No.3060
4012	Custom macro config data absence	The custom macro displaying function is used,while the config file is not found, or no config data in it	Modify the Para.No.8132#6, or execute the config file custom macro window
4013	Custom macro config data error	The custom macro displaying function is used,while error configuration found in the config file,Please execute the correct config file	
4020	Default encoder communicates through GSKLink while none of valid communication station address has been set for the spindle		Modify para.No.3717, or set a none-zero value to para.3723
4100	Fail to set FPGA param	The FPGA Para.was not set successfully which may cause abnormal in threading or spindle control	Please power on again or contact the manufacturer
4110	Servo battery voltage low	The position will lost when power off the servo	Troubleshooting: Change the battery. Note: It is important to change the battery in the state of (that is, the servo driver is power-on) the driver alarm displays. And then turn it off, the alarm eliminates after the power is turned on. Explanation: User can continue machining after this alarm occurs, but it is essential to change the battery as soon as possible.
4120	Gateway GSK-Link-PA terminal slave has been altered.		The gateway and its corresponded communication link need to be restart。

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
4200	Machine panel devID error		Check the connection between machine panel and CNC
4201	Machine panel device information error		Check the connection between machine panel and CNC.
4202	Machine panel continuous communication error detected		Check the connection between machine panel and CNC
4205	Soft panel enabled, and machine panel stopped		Refer parameter No.8136#3(NOP) to enable machine panel
4210	MDI panel communication error	MDI panel communicated with CNC failure	
4300	System enters level 1	Never attempt to modify the parameter when the system is the level one authority.	
4304	Default password for try-out function detected	Please change the password for try-out function for security	
4305	The system will reach the try-out time limit	The system will stop soon. Please contact the salsman to get release code	
5000	Communication disconnect physically	Disconnection or interference on cable may cause the alarm	Power on again
5001	Check of ring devices overtime	Disconnection or interference on cable may cause the alarm	The system has re-tried to fix the problem.Even so,it may fail at last.Please refer to final state of communication to [RESET] the alarm, or power on again
5002	Handshake of ring B failure	Disconnection or interference on cable may cause the alarm	The system has re-tried to fix the problem.Even so,it may fail at last.Please refer to final state of communication to [RESET] the alarm, or power on again
5003	Check of time-delay failure	Disconnection or interference on cable may cause the alarm.	The system has re-tried to fix the problem.Even so,it may fail at last.Please refer to final state of communication to [RESET] the alarm, or power on again
5004	Communication configuration Para.error	Disconnection or interference on cable may cause the alarm.	The system has re-tried to fix the problem.Even so,it may fail at last.Please refer

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
			to final state of communication to [RESET] the alarm, or power on again
5005	GSKLink initial error		Please power on again
5006	Devices number detected unequal to value set in system params		Refer to PAR.No.1023, No.3717, No.3050, No.3051, No.3052, No.3053, No.3054, No.3060, and comm station address of each device. Power on again.
5007	None of valid comm station address of device detected		Refer to PAR.No.1023, No.3717, No.3050, No.3051, No.3052, No.3053, No.3054, No.3060, and comm station address of each device. Power on again
5008	Communication returned to CP0 on master station		Power on again
5010	Incorrect parameter for GSKLink		Please refer to PAR.No.1023, No.3717, No.3050, No.3051, No.3052, No.3053, No.3054, No.3060, and comm station address of each device. Power on again
5011	GSKLink disconnected		Please check the connection to each device. Power on again
5020	MDT lost		Please check the device
5021	MST lost		Please check the device
5022	MDT data verified error		Please check the device
5023	GDT data verified error		Please check the device
5030	C1D device alarm		Please check the device
5031	C2D device alarm		Please check the device
5040	Communication of slave device stopped		Please check the device
5100	IDN16,24 error		Refer to corresponding system parameters, and check the work state of the device
5101	IDN32,35 error		Refer to corresponding system parameters, and

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
			check the work state of the device
5102	IDN5030,5031,5033 error		Refer to corresponding system parameters, and check the work state of the device
5103	Fail to config I/O unit		Please check the device.
5132	C3D device alarm	This may be caused by：(1)Modify servo Para.on servo side;(2)Re-load\recover servo parameters on servo side;(3)Save servo parameters on servo side	Press [RESET] to release the alarm
5133	Please check the device. Press [RESET] to release the alarm		
5198	Initializing GSKLink	Please wait	
5199	GSKLink communication error.		
5200	Fail to load servo property		Power on again.
5201	Fail to load servo information lists		Power on again
5210	Fail to load servo parameters		Power on again
5211	Import of servo parameters	Please select the import Para.to be effective by【SELCT EFF. PAR】softkey. The parameters could not be saved before selecting.	
5220	Inconsistent of servo Para.read from servo-para-file saved in cnc and that loaded from servo		Please enter [servo param] layer and then select the effective servo Para.with【SELCT EFF. PAR】softkey.
5400	I/O unit Para.file not exist	I/O unit Para.file not exist, And the file failed to be created automatically	Press [RESET] to release the alarm
5401	Mapping table does not record the correct read configuration	It is already automatically written the current read configuration.	Press [RESET] to release the alarm
5402	The mapping table detects the incorrect address or parameter setting		The error has been corrected automatically Press [RESET] to release the alarm.

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
5403	Fail to load property	Fail to read the information from remote equipment, check whether the I/O unit is on the normal working state.	Power on again
5404	Inconsistent	The property of I/O unit loaded is different from before.Current property is recorded automatically	Press [RESET] to release the alarm.
5406	Fail to config the device	Fail to send the parameter to the remote equipment, check whether the I/O is on the normal working state.	Power on again
5500	Fail to load property of gateway		Power on again
6000	Data exceeds permissive range of extern coordinate origin offset	Data exceeds permissive range	Modify the program
6001	Data exceeds permissive range of tool offset	Data exceeds permissive range	Modify the program
6005	Data exceeds permissive range of additional workpiece coordinate origin	Data exceeds permissive range	Modify the program.
6006	Error loading Append coordinate offset	Fail to read the additional workpiece coordinate system offset value when the workpiece coordinate system is updated.	Please check the cnc flash
6007	Data exceeds permissive range of workpiece coordinate origin	Data exceeds permissive range	Modify the program.
6010	Toollife data run error	Tool Group No. exceeds the maximum allowable value,or the tool group commanded in the machine program is not set	Modify the program or modify the tool life data
6015	Auto tool compensation signal Not detected	In auto tool compensation mode(G36, G37),when enter the area assign in parameters, the measurement arrival signal is not detected (XAE\EAE)	Refer to settings or operation
6020	Over-speed of spindle in threading	In threading, the spindle speed specified is too fast for the threading axis	Modify the program
6021	Spindle speed too low in threading	S command was not specified or is set to zero, Spindle encoder feedback is abnormal	Modify the program

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
6022	Signal of 1-rotation not detected	The specified speed by spindle is lower when the thread machining is performed, which causes the feed axis abnormal.	Modify the program
6023	Increase/decrease amount of lead error in threading	Increase/decrease amount of lead is too large, which causes the feed axis abnormal	Modify the program
6024	Spindle encoder lines out of 100~5000	This type of encoder is not supported in taping	Check the parameter setting (NO.3720 and NO.3723) or change the spindle encoder
6025	Spindle rotation signal(SFR,SRV) detected error in taping	Check the output of G signal SFR, SRV or encoder connection.	Modify the program or PLC
6026	The spindle speed is excessive high or low during the common tapping, so that the tapping axis can not be performed the feed normally.	The possible reasons: 1) S command specified a value equal to zero or out of range; \n2)Encoder feedback abnormal 3) Abnormal spindle encoder feedback.	Modify the program or check the status of encoder
6027	One-revolution signal Not detected in tapping	Refer to pram NO.3723 for correct encoder setting of spindle	Check the work-state or connection of the encoder
6028	M code execution abnormal for spindle start	stop in tapping\nRefer to PLC for M code processing with spindle start/stop,CW/CCW rotation	Check whether the M code has become effective
6029	Incorrect spindle encoder selection	Incorrect spindle encoder selection, so that the thread or tapping can not be performed.	Modify the program or check the parameter NO.3723.
6030	Rigid tapping signal is off	The probable reasons: 1) Fail to detect the RGTAP signal or do not specify (By the M29 or other M codes) the rigid tapping mode before the tapping is performed; 2) Fail to correctly send the rigid tapping spindle selection signal.	Modify the program or check the ladder diagram
6031	C-axis commanded in spindle mode when execute rigid tapping	The program specified a movement along the Cs-axis when the signal CON (G27#7) is OFF.	Correct the program, or consult PLC program to find the reason the signal is not turned on
6032	Spindle selection error in rigid tapping	Refer to following possible caution: 1) Check the Plc for correct spindle selection signal assignment. 2) Refer to Para.for correct Cs contour setting	
6035	C-axis commanded in spindle mode	The program specified a movement along the Cs-axis when the signal CON (G27#7) is OFF.	Correct the program, or consult PLC program to find the reason the signal is not turned on
6036	C-axis commanded	The program specified a	Correct the program, or

Appendix

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
	in spindle mode when execute G28	movement along the Cs-axis when the signal CON (G27#7) is OFF.	consult PLC program to find the reason the signal is not turned on
6037	Spindle speed arrival signal Not detected	Spindle speed arrival signal(SAR) is not detected in cutting	Modify the program or PLC
6038	Error detecting the release signal for rigid tapping (RGTAP)	RGTAP signal error in cancel rigid tap	Refer to PLC or Para.No.5200#2
6040	spindle	In polygon processing, the rotation axis for workpiece spindle is not set, or Cs contour setting is incorrect, or the rotation axis for workpiece conflicts with the axis for tool rotation.	Modify the Plc for spindle selection, or Refer to Para.No.7610
6041	synchronous instruction is illegal in polygon processing	In the polygon processing, synchronous operation, the synchronous axis movement command is issued by the NC program	please modify the program
6042	synchronous mode command is illegal in polygon processing	Polygon processing, while trying to synchronize the operation and CS contour control	please modify the program
6043	Not in position control of workpiece rotation axis in polygon processing	The workpiece rotation axis doesn't enter position control mode in polygon processing	Modify the program or Check the PLC for the reason why the signal doesn't become ON
6044	Polygon cutting spindle speed error	In the polygon processing method can not maintain the rotational speed ratio command value, since the spindle speed or faster than the polygonal shaft clamp synchronous or low	please modify the program
6045	Not in position control of tool rotation axis in polygon processing	The tool rotation axis doesn't enter position control mode in polygon processing.	Modify the program or Check the PLC for the reason why the signal doesn't become ON
6050	Illegal variable number in macro program	A value not defined as a variable number is designated in the custom macro	Modify the program
6051	Macro variables are protected	The macro variable is protected from modification	Refer to PARA. 6031 and 6032
6052	Macro variables modification is forbidden	The macro value is read-only	Modify the program
6053	Null value not allowed for system macro	Null value is not allowed to set to system macro	Modify the program
6054	Data exceeds permissive range of macro value	Data exceeds permissive range	Modify the data

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
6060	The same axis was commanded by PLC and CNC	Axis control command was given by PLC to an axis controlled by CNC	Modify the program or check the PLC.
6061	Cannot change PLC control mode	Select an axis which is in commanding by PLC control	Modify the PLC program
6070	Encoder data error because of PC signal Not detected	The servo is not applied, or Gsklink communication failed, or the spindle failed to rotate to the precise position for correct encoder data	Check the Gskling communication state or the work state of spindle
6075	Error return to reference point of slant-controlled axis	In the state of the manual reference point return in the slopping axis control and no reference point return operation after power-on for the automatic reference point return, try to perform reference point return operation of the perpendicular axis in the state of the slopping axis reference point return does not execute.	Perform the reference point return operation of the perpendicular axis after its operation along the slopping axis is completed.
6080	Illegal G code specified in handwheel retraction block	Illegal G code specified in handwheel retraction block was found	Modify the program
6200	Canned cycle cmd in non ZX plane	Canned cycle can't command in non ZX plane	Modify the program
6201	Specify other axes not included in ZX plane	Specify other axes not included in ZX plane	Modify the program
6202	In G90/G77,G92/G78 commands, absolute value of R is greater than that of address U(radius assigned)	In block using G90/G92 command（G code Group B:G77/G78）,When sign of address R and U is opposite, absolute value of R is greater than that of U(Radius assigned)	Modify the program
6203	In G94/G79 commands, absolute value of R is greater than that of W.	In blocks using G94（G code Group B:G79）,When the sign of address R and W is opposite, the absolute value of R is greater than that of W	Modify the program
6210	Illegal plane select in multiple repetitive cycle	When specifying the multi-cycle G70~G76, its plane does not XZ (modal regards as G18), or specify the plane shifting commands G17~G18 in G70~G76 blocks.	Modify the program
6211	Specify other axes not included in ZX plane in G70~G76	Specify other axes not included in ZX plane in G70~G76	Modify the program.
6212	Illegal G code in G70~G73	In the G70~G73, unused G code is specified between two blocks based upon the addresses P and	Modify the program

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
		Q; the G0~G3, G96/G97, G98/G99, G40~G42can only be specified in the blocks of NS~NF.	
6213	G70~G73 cannot operate in MDI mode	G70~G73 with P & Q was specified in MDI mode	Modify the program
6214	Illegal macro statement in G70~G73	Macro statement is unallowable in G70~G73 command	Modify the program
6215	Call the sub-program in the G70~G73 cycle	Fail to call the sub-program in the G70~G73 cycle	Modify the program
6216	Illegal subprogram call in G70~G73	Subproram call is unallowable in G70~G73 command	Modify the program
6217	Incorrect address P command in G70~G73	Fail to specify the P or the command exceeds its range in the G70~G73.	Modify the program
6218	Incorrect address Q command in G70~G73	Fail to specify the Q or the command exceeds its range in the G70~G73.	Modify the program
6219	Fail to search the address P or Q in the G70~G73	The sequence number specified by P & Q was not found in G70,G71,G72 or G73	Modify the program
6220	The commands between P and Q are same in the G70~G73	The number specified by address P & Q the same in G70~G73	Modify the program
6221	The two blocks as components of G71~G73 command are discontinuous	The two blocks as components of G71~G73 command are discontinuous, which is possible to cause error	Modify the program
6222	Blocks between Ns & Nf exceeds 100 in G70~G73	Too many blocks of the Ns—Nf in G70~G73, it exceeds the Max. allowable 100 blocks.	Modify the program
6223	Cutting direction determined by Ns-Nf blocks is the same with track direction in G71~G73	Direction of cutting conflicts with track direction, so that the track will not close	Modify the program
6224	Direction of cutting(in Ns-Nf blocks) and finishing allowance is the same in G71~G73	Direction of cutting(in Ns-Nf blocks) and finishing allowance is the same in G71~G73	Modify the program
6225	Shape specified in Ns-Nf blocks Not monotonous in	Arc specified in Ns-Nf blocks of G70~G73 is long arc	Modify the program

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
	G71~G73		
6226	Arc shape specified in Ns-Nf blocks Not monotonous in direction of X axis in G71~G72	Arc shape specified in Ns-Nf blocks is not monotonous in direction of X axis in G71type-I or G72	Modify the program
6227	Arc shape specified in Ns-Nf blocks Not monotonous in direction of Z axis in G71~G72	Arc shape specified in Ns-Nf blocks is not monotonous in direction of Z axis in G71type-I or G72	Modify the program
6228	Arc shape specified in Ns-Nf blocks Not monotonous in direction of X axis in G71 type-II	Arc shape specified in Ns-Nf blocks is not monotonous in direction of X axis in G71 type-II	Modify the program
6229	Arc shape specified in Ns-Nf blocks Not monotonous in direction of X axis in G73	Arc shape specified in Ns-Nf blocks is not monotonous in direction of X axis in G73	Modify the program
6230	Arc shape with finishing allowance specified in Ns-Nf blocks Not monotonous in direction of Z axis in G73	Arc shape with finishing allowance specified in Ns-Nf blocks is not monotonous in direction of Z axis in G73	Modify the program
6231	Arc shape with retraction amount specified in Ns-Nf blocks Not monotonous in direction of Z axis in G73	Arc shape with retraction amount specified in Ns-Nf blocks is not monotonous in direction of Z axis in G73	Modify the program
6233	X axis of start point was on cutting path in G71/G72	Start point was on cutting path in G71/G72, which may cause interfere of tool and workpiece, referring to para. 5104#2 FCK	Modify the program
6234	Z axis of start point was on cutting path in G71/G72	Start point was on cutting path in G71/G72, which may cause interfere of tool and workpiece, referring to para. 5104#2 FCK	Modify the program
6235	X axis of start point was on cutting path in G73	Start point was on cutting path in G73, which may cause interfere of tool and workpiece, referring to para. 5104#2 FCK	Modify the program.
6236	Z axis of start point was on cutting path in G73	Start point was on cutting path in G73, which may cause interfere of tool and workpiece, referring to para. 5104#2 FCK	Modify the program
6237	Too many concaves in G71 type II	More than 10 concaves are specified in G71 type II	Modify the program

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
6238	Direction of chamfering and finishing allowance along X axis is inconsistent in G73	Direction of chamfering and finishing allowance along X axis is inconsistent in G73	Modify the program
6239	Direction of chamfering and finishing allowance along Z axis is inconsistent in G73	Direction of chamfering and finishing allowance along Z axis is inconsistent in G73	Modify the program
6240	Too many cutting blocks per cycle in G71 II cycle	Too many cutting blocks per cycle are specified in G71 II cycle	Modify the program
6241	Finishing allowance U of X axis in G70~G73 beyond proper range	Finishing allowance U of X axis in G70~G73 is beyond proper range	Modify the program
6242	Finishing allowance W of Z axis in G70~G73 beyond proper range	Finishing allowance W of Z axis in G70~G73 is beyond proper range	Modify the program
6243	G00 or G01 move command not found in first block of G71~G72	G00 or G01 move command should include in first block of G71~G72	Modify the program
6244	G00-G03 move command not found in first block of G73	G00-G03 move command not found in first block of G73	Modify the program
6245	Over tolerance of radius in G71~G73	The radius D-value calculated by arc command in G71~G73 exceeds its range.	Modify the program
6246	X axis motion in the first block of G71	X axis increment was not commanded in first block of G71, or X axis increment is zero	Modify the program
6247	Z axis motion in the first block of G72	Z axis increment was not commanded in first block of G72, or Z axis increment is zero	Modify the program.
6248	Depth of cutting is less than zero or more than maximum in G71 or G72	Single tool infeed value is less than or equals to 0 in G71 or G72 command, alternatively, it is more than the top tool in-feed value.	Modify the program
6249	Escaping amount(R(e)) is less than zero in G71 or G72	Escaping amount(R(e)) is less than zero in G71 or G72	Modify the program
6250	Increment cutting amount out of range in G73	Increment cutting amount out of range in G73	Modify the program
6251	The number of division R(d) in G73 out of range	The rounding number of division is less than 1 or more than 999	Modify the program
6252	Direction of cutting and finishing	Cutting direction determined by position point and NS block	Modify the program

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
	allowance is the same	conflicts with the finishing allowance direction	
6253	Para. modified failure in G71~G76	Para. modified failure in G71~G76. Check that the para. file be abnormal	Modify the program
6254	Part prog. segment loading failure in G71~G76	Fail to read the program in NS~NF from G70~G76 during operation	Modify the program
6260	Value of address Q beyond the proper range in G74G75	Value of address Q is beyond the proper range in G74G75	Modify the program
6261	Value of address P beyond the proper range in G74G75	Value of address P is beyond the proper range in G74G75	Modify the program.
6262	Value of address R(e) is beyond the proper range in G74 or G75	Retraction amount specified by address R(e) in G74 or G75 is less than zero, or greater than maximum.	Modify the program.
6263	Value of address R(△d) is beyond the proper range in G74 or G75	Retraction at the cutting end specified by R(△d) in G74 or G75 is less than zero, or greater than maximum	Modify the program
6270	X or Z axis increment is 0 in G76	X or Z axis increment is 0 in G76	Modify the program
6271	Repetitive count in finishing is less than 1 or greater than 99 in G76	Repetitive count in finishing is less than 1 or greater than 99 in G76	Modify the program
6272	G76 thread chamfering width P(r) exceeds the permit range.	G76 thread chamfering width P(r) exceeds the permit range.	Modify the program
6273	Angle of tool tip out of range in G76	Angle of tool tip out of range in G76	Modify the program
6274	Q(△dmin) out of range in G76	Minimum cutting depth Q (△dmin) out of range in G76.	Modify the program
6275	Finishing allowance R(d) out of range in G76	Finishing allowance R(d) is less than least increment in G76	Modify the program
6276	G76 thread taper R(i) beyond the proper range	G76 thread taper value specified by address R(i) exceeds the proper range	Modify the program
6277	R and U is inconsistent for taper thread cutting in G76	Machining start position is between thread beginning point and end point in G76	Modify the program.
6278	Thread height not specified by P in G76	Thread height not specified by P in G76	Modify the program.
6279	Incorrect thread height in G76	Thread height is less than Finishing allowance in G76	Modify the program

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
6280	Thread height is less than Finishing allowance or minimum cutting depth in G76	Thread height is less than Finishing allowance or minimum cutting depth in G76	Modify the program
6281	Thread height is larger than the destination to end point in G76	The tooth specified by G76 is more than the distance between the positioning point and thread end.	Modify the program
6282	Number followed address Q is out of range in G76	depth of cut in 1st cut Q was out of range, or not specified	Modify the program
6283	Taper of thread is bigger than 45 in G76	Taper of thread is bigger than 45 in G76	Modify the program
6284	Taper is parallel to the tool in G76	The thread taper specified by G76 is parallel with the cutter, and it can not be performed the cutting.	Modify the program
6285	Incorrect thread taper or pointed angle specified by G76	G76 can not be specified the correct pointed angle or thread taper, so that the normal cutting can not be performed.	Modify the program
6300	Illegal S code command in rigid tapping	In rigid tapping, an S value is out of range or not specified	Modify the program
6301	S code not found in rigid tapping by G84 or G88	In G84 or G88 rigid tapping(parameter 5200#0 is set to 1),an S value is not specified	Modify the program
6302	Beyond the range of address J for spindle extraction in rigid tapping	Value of address J exceeds the range for spindle extraction in rigid tapping	Modify the program
6303	Illegal K in tapping	The specified repeated times K value in the tapping or drilling canned cycle does not within the 1~99.	Modify the program
6304	Lead specified in address F beyond the range in tapping	Lead or speed specified in address F beyond the range in tapping	Modify the program
6305	Inch Lead value specified in address I beyond the range in tapping	Inch Lead value specified in address I is beyond the range in tapping	Modify the program
6306	Incorrect program command in the rigid tapping	In the rigid tapping, the M code and S value based upon the rigid tapping mode does not share with a same block.	Modify the program
6307	Illegal axes-motion command in rigid tapping mode	In rigid tapping, a motion block is specified between M code(start rigid tapping) and G84 command	Modify the program
6308	Invalid axis in rigid tapping or drill cycle command	An invalid axis is specified in G83~G89 command	Refer to Para.1022 for axis-property setting, or

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
			modify the program.
6309	Tapping axis or drilling axis changed in tapping	Command G84/G88 when tapping， or switch G83/G87, G85/G89 command in drilling cycle. For example: Specify G87 when in G83 state, or specify G88 when in G84 state, or specify G89 when in G85 state	Modify the program.
6310	Plane changes while tapping	The plane shifting commands G17~G19 are specified in rigid tapping mode.	Modify the program
6311	Tapping distance too short in rigid tapping	In rigid tapping command G84/G88, tapping distance (distance from R plane to bottom of hole) is less than lead	Modify the program
6312	Cutting depth less than retraction depth in deep-hole tapping	In peck tapping command(Q is not set to 0),Value of Q is less than retraction depth setting in Para.No.5213	Modify the program of param
6313	Unusable data specified in tapping	Specify other M code or S code between rigid tapping M code block and G84 block	Modify the program
6314	Illegal M code specified in rigid tapping or drilling cycle	The M code without sharing a same block is specified in the G83~G89 blocks or in its modal. The M code can be shared with a same block: 1. G83/G87 and G85/G89 are the C axis clamping M code (parameter 5110 setting value). 2. The M codes of G84/G88 can be shared with a same block: Specify the rigid tapping mode M code (Parameter 5210 setting value; the M code is M29 if the parameter is set to 0); C axis clamping M code (Parameter 5110 setting value).	Modify the program
6315	M code to clamp C axis error in drilling cycle	M code to clamp C axis error, referring to Para.No.#5110, and M30 can't be used	Modify the program
6316	G84/G88 tapping specified in G96 mode	G84/G88 tapping can't be specified in G96 mode	Modify the program
6317	Illegal address specified in rigid tapping or drilling cycle	In G83~G89 mode, G7.1/G107, G12.1/G112, G13.1/G113 is specified, alternatively, the polygon machines the G51.2 command	Modify the program
6318	Illegal T code specified in rigid tapping or drilling cycle	Illegal T code is specified in G83~G89	Modify the program
6330	Improper command	A function which can't be used in	Modify the program

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
	in custom macro program	custom macro program is commanded	
6331	Brackets not match in custom macro program	The '['AND']' does not match in the user macro program	Modify the program
6332	Condition command error in custom macro program	Condition command doesn't exist in custom macro program.	Modify the program
6333	Format error in macro program	There is an error in other format than <Formula>	Modify the program
6334	Illegal variable number in macro program	A value not defined as a variable number is designated in the custom macro	Modify the program
6335	Unallowable macro program call	A program in G66 modal specified M98, G65 or G66	Modify the program
6336	The nesting of bracket exceeds the upper limit	The nesting of bracket exceeds the upper limit(5 quintuple)	Modify the program
6337	Illegal argument	The SQRT argument is negative, the arguments BCD and BIN are negative or BIN argument value can not be shifted into the correct BCD code.	Modify the program
6338	Divided by zero	Divisor was 0(including tan90°)	Modify the program
6339	Quadruple macro modal call	A total of four macro call and macro modal calls are nested	Modify the program
6340	Macro control command can't be used in DNC and MDI program	Macro control command was specified in DNC and MDI mode	Modify the program
6341	Missing end statement	DO-END does not correspond to 1：1.\nOr has other illegal cmd exists in END block, incorrect format	Modify the program
6342	assignment operation of custom macro not allowed	User's authority is too low to execute assignment operation of custom macro	Modify the program
6343	Illegal loop number	in DOn, 1≤n≤3 is not established	Modify the program
6344	NC and macro statement in same block	NC and custom macro coexist	Modify the program
6345	Illegal macro sequence number	The sequence number specified in the branch statement was not 1~99999, or, it can't be searched	Modify the program
6346	Illegal argument address	An unallowable argument address was used which is not in <Argument Designation>	Modify the program
6347	Tool radius direction data error	The custom macro data used for tool radius direction input should be in the range of 0~9 after rounded	Modify the program

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
6348	Illegal argument	The argument is incorrect, or the argument is illegal	Modify the program
6349	Operand of logical operation statement error	Operand of logical operation statement OR, XOR, AND are negative.	Modify the program
6350	G67 custom macro cancel	G67 was commanded while corresponding G66 command was not found. Please check the program whether G66 should be added, and referring to para 6000#1 G67.	Modify the program
6351	Macro variables are protected	The macro variable is protected from modification	Modify the program
6352	Macro variables modification is forbidden	The macro value is read-only	Modify the program
6353	Overflow of float data	Float data exceeds the allowed range in macro calculating(±1E47 when ARA.6008#0 is set to 1, or else is ±1E308)	Please modify the program or PARA. 6008#0
6354	Macro prog. shouldn't be called by M code	Macro prog. shouldn't be called by M code	Modify the program
6355	Null value not allowed for system macro	Null value is not allowed to set to system macro	Modify the program
6356	Not proper T code to call custom macro program	Not proper T code to call custom macro program	Modify the program, or refer to Para.No.6008#5, 6060~6069
6357	Illegal G code in same block with G66/G67	G66/G67 doesn't work with G code of other group in same block.	Modify the program
6370	No solution at NRC	A point of intersection can't be determined for tool nose radius compensation.	Modify the program
6371	Not allowed to start & cancel NRC in arc command	Start or cancel tool nose radius compensation in circular interpolation	Modify the program
6372	Can't change plane in NRC	The offset plane is switched in tool nose radius compensation	Modify the program
6373	Interference in circular block	The arc start point or end point coincides with arc center, or destination point is not on arc.	Modify the program
6374	Interference in G90 or G94 block	Overcut will occur in tool nose radius compensation in canned cycle G90 and G94	Modify the program
6375	Interference in arc concluded from checking	Overcut is possible to occur in tool nose radius compensation	Modify the program
6376	Inconsistent of direction of tool path in NRC and on drawing	Inconsistent of direction of tool path in NRC and on drawing(if exceeds range between 90 and 270 degree)possibly result in part	Modify the program

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
		overcut	
6377	G41 or G42 not allowed in MDI mode	G41 or G42 was specified in MDI mode(tool nose radius compensation),referring to para 5008#4 MCR	Modify the program
6378	Inner whole circle cutting overcut	In inner whole circle cutting, overcut possibly occur, referring to para 5008#5 CNF	Modify the program
6379	undercut in machining step being less than tool radius	undercut in machining step being less than tool radius, search 5008#6 CNS	Modify the program
6380	Radius of arc is less than that of tool in inner surface arc cutting	Radius of arc is less than that of tool in inner surface arc cutting, which might cause overcut	Modify the program
6381	Arc cmd exists when cancel temporarily or create NRC	While NRC is canceled temporarily as a result of a non-NRC G code, an arc command was specified	Modify the program
6382	Over tolerance of radius in tool compensation calculation	In the cutter compensation, the radius D-value calculated from arc command exceeds its range.	Modify the program
6383	NRC detected error	Detect error in tool nose radius compensation. This is due to program or operator	Modify the program
6384	Tool offset not executed before polar coordinate interpolation	The tool offset is not executed of the linear axis in Polar coordinate interpolation	Modify the program
6385	Error found when cancelling cylindrical interpolation	Cylindrical interpolation can't be cancelled in C tool compensation mode	Modify the program
6386	Property error for axes of plane of C tool compensation	Property set error for two axis of plane of C tool compensation	Modify the program
6400	HF/CNR function or CHF/CNR measurement-program-inputting function disabled	If block contains address', R'or',C', refer Para.No.8134#2 for setting to 0 2. Please refer to parameter No.3453 and parameter No.8134	Modify the program
6401	CHF/CNR measurement-program-inputting function Only used in automatic mode.	CHF\CNR measurement-programming function do not work in MDI or DNC mode	Modify the program
6402	Code than G01G02/G03 after CHF/CNR	Improper movement other than G01G02/G03 is specified next to chamfer/corner R block	Modify the program

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
6403	Code is not G01G02/G03 after CHF/CNR	The block next to the chamfer/corner R is not G01G02/G03	Modify the program
6404	Illegal axis after CHF/CNR	An axis not selected in the plane is specified in the block next to the chamfer/corner R	Modify the program
6405	Program end block after CHF/CNR\nIn auto-meme	DNC mode ,the next block is program end block after CHF/CNR R specified; Or in MDI mode, the block containing CHF/CNR R is the last block	Modify the program
6406	CHF/CNR R address specified in the NF block of G71~G76	CHF/CNR R address was specified in the NF block of G71~G76	Modify the program
6407	Plane selection not allowed after CHF/CNR	It's not allowed to specified a command to select axis plane in the block next to the chamfer/corner R	Modify the program
6408	Improper movement after CHF/CNR\nA move distance less than the value of chamfering , chamfering point is not on the tool track	The movement value along axis in the specified block of chamfering or corner R is smaller than the chamfering value or corner R value, alternatively, the chamfering point calculated does not at the path.	Modify the program
6409	Data error in CHF/CNR	Invalid data of chamfer/corner R is specified	Please modify the program
6410	Multiple G04 dwells are specified after the chamfering or corner R	In the block, after specifying the block of the chamfering or corner R, two or more G04 dwell commands are specified.	Please modify the program
6411	None-motion block after CHF/CNR	Blank block or M/S/T/F block without motion command after CHF/CNR.	Please modify the program
6412	End position or angel not specified in CHF/CNR measurement-program-inputting function	For blocks after Address (Aa), coordination or angle value should be specified	Please modify the program
6413	End position calculates error	In the drawing dimension direct input, the specified angle is less than 1 degree. The E-O-B can not be correctly calculated.	Please modify the program
6414	Address ',A' specified in the last block in direct drawing dimension programming	In direct drawing dimension programming Address, ',A' specified in the last block, so that the destination of the block can't be auto calculated	Please modify the program

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
6415	Address ',A' specified in NF block of G71~G76 command in direct drawing dimension programming	In direct drawing dimension programming, Address ',A' specified in NF block of G71~G76 command, , so that the destination of the block can't be automatically calculated	Modify the program
6416	Fail to calculate destination point	In direct drawing dimension programming, Fail to calculate destination point	Modify the program
6417	Illegal G code specified in direct drawing dimension programming	Illegal G codes are as follows:\nG code of Group 00 (G04 excluded). G code of Group 01(G00,G01,G32 excluded).\nG code of Group 07 (G22/G23 excluded), G code of Group 11(G17~G19 excluded).Group 08(G83~G89 excluded)	Modify the program
6418	Fail to employ direct drawing dimension programming for G code modal state	G code modal state doesn't not work with direct drawing dimension programming, included G codes are as follows: G7.1 circular interpolation, Group 01 (G00, G01, G32 excluded), Group 08 G83~G89	Modify the program
6419	Illegal address in CHF/CNR measurement-program-inputting function	This may be caused by: More than 2 continuous blocks without motion command specified	Modify the program
6430	Illegal tool group number	Tool life group number is less than 1; alternatively, it exceeds the Max. allowable value set by parameter 6813; when parameter 6813 sets to 0, up to 128 groups can be performed.	Modify the program
6431	Tool group number not found	Tool group number commanded in machining program is not set.	Modify the program or parameter
6432	T code not found	In tool life registeration , a T code was not specified where is should be	Modify the program
6433	Illegal tool life data	The tool life to be set is too excessive or not set. When count with time, refer to Para.No.6805#1 for time unit.	Modify the program
6434	Tool life management command not matched	T[][]99 not specified or specified error when using T[][]88\nModify the program	Modify the setting value.
6440	Block sequence NO. not found	In program restart operation, the sequence NO. is not found	Modify the setting value
6441	Illegal assign of G71~G73 cycle blocks for program restart operation	In restart operation, the block assigned block is included in NS~NF blocks of G71~G73	Please assign alter start block.

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
6442	Illegal G code in start block of program restart operation	Illegal G code is specified in start block of program restart operation	Please assign alter start block
6443	Unallowed restart from threading or rigid tapping block	The restart block contains threading commands(G32, G33, G34), thread cycle(Group A:G92, Group B:G78), thread canned cycle(G76), and raid tapping cycle(G84/G88)	Please assign alter start block
6444	System variables operation found in program restart search	In program restart search, operation on system variables is not allowed	Please assign again
6445	,Illegal command specified in MDI mode after program restart search	Only M, S, T commands will execute in MDI mode after program restart search	Modify the program
6446	G28/G30 command found in program restart search	G28/G30 command found in program restart search, while command the start of program and has not returned to the ref. point	Modify the program

Appendix 5 Installation Layout

Appendix 5.1 Installation Dimension of GSK988TA/988TA1/988TB and its Accessory

GSK988TA/988TA1/988TB divide into GSK988TA1 (Vertical), GSK988TA1-H (Horizontal), GSK988TA (Vertical), GSK988TA-H (Horizontal), GSK988TB (10.4 inch vertical) and GSK988TB-H (10.4 inch horizontal), and its configured operation panels are also different, refer to the following table for the detailed types.

Table 5-1

Production type	Panel name	Structure	Name
GSK988TA1 （vertical-type）	Machine operational panel	With MPG	MPU-08E
		Without MPG	MPU-09E
GSK988TA1-H （horizontal-type）	Machine operational panel	With MPG	MPU-10E
		Without MPG	MPU-11E
GSK988TA （vertical-type）	Machine operational panel	With MPG	MPU-08
		Without MPG	MPU-09
GSK988TA-H （horizontal-type）	Machine operational panel	With MPG	MPU-10
		Without MPG	MPU-11
GSK988TB （10.4 inch screen vertical-type）	Editing keyboard		EDU-01
	Machine operational panel		MPU-20
	Machine operational panel	With MPG	AP04
		Without MPG	AP05
GSK988TB-H （10.4 inch screen horizontal-type）	Editing keyboard		EDU-02
	Machine operational panel		MPU-20
	Machine operational panel	With MPG	AP06
		Without MPG	AP07

Appendix

Appendix 5.1.1 GSK988TA1 and its Accessory

Appendix 5.1.1.1 GSK988TA1 Host Figure Installation Dimension

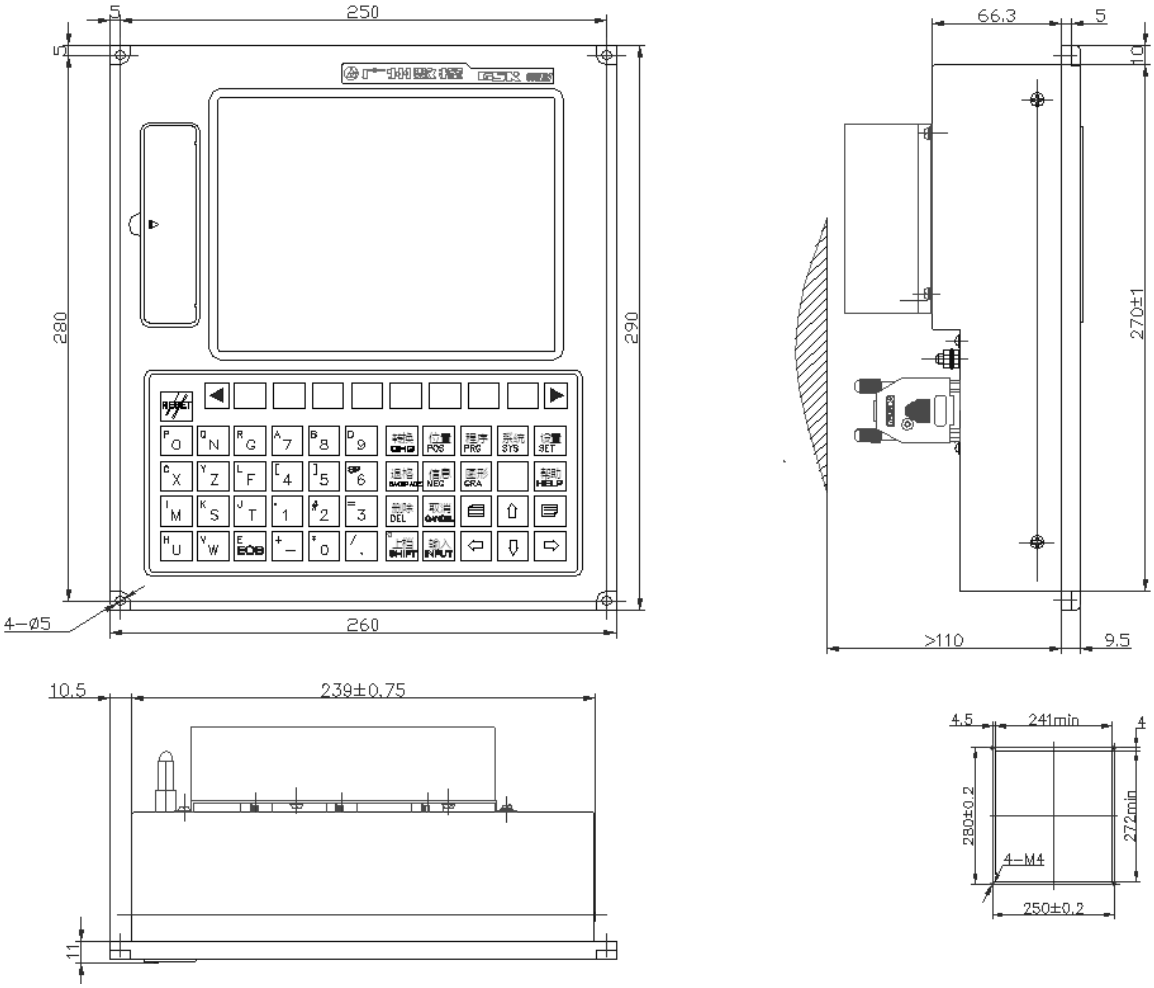


Fig. 5-1 GSK988TA1 appearance installation dimension

Appendix 5.1.1.2 Outline Installation Dimension of GSK988TA1 Operation Panel MPU-08E

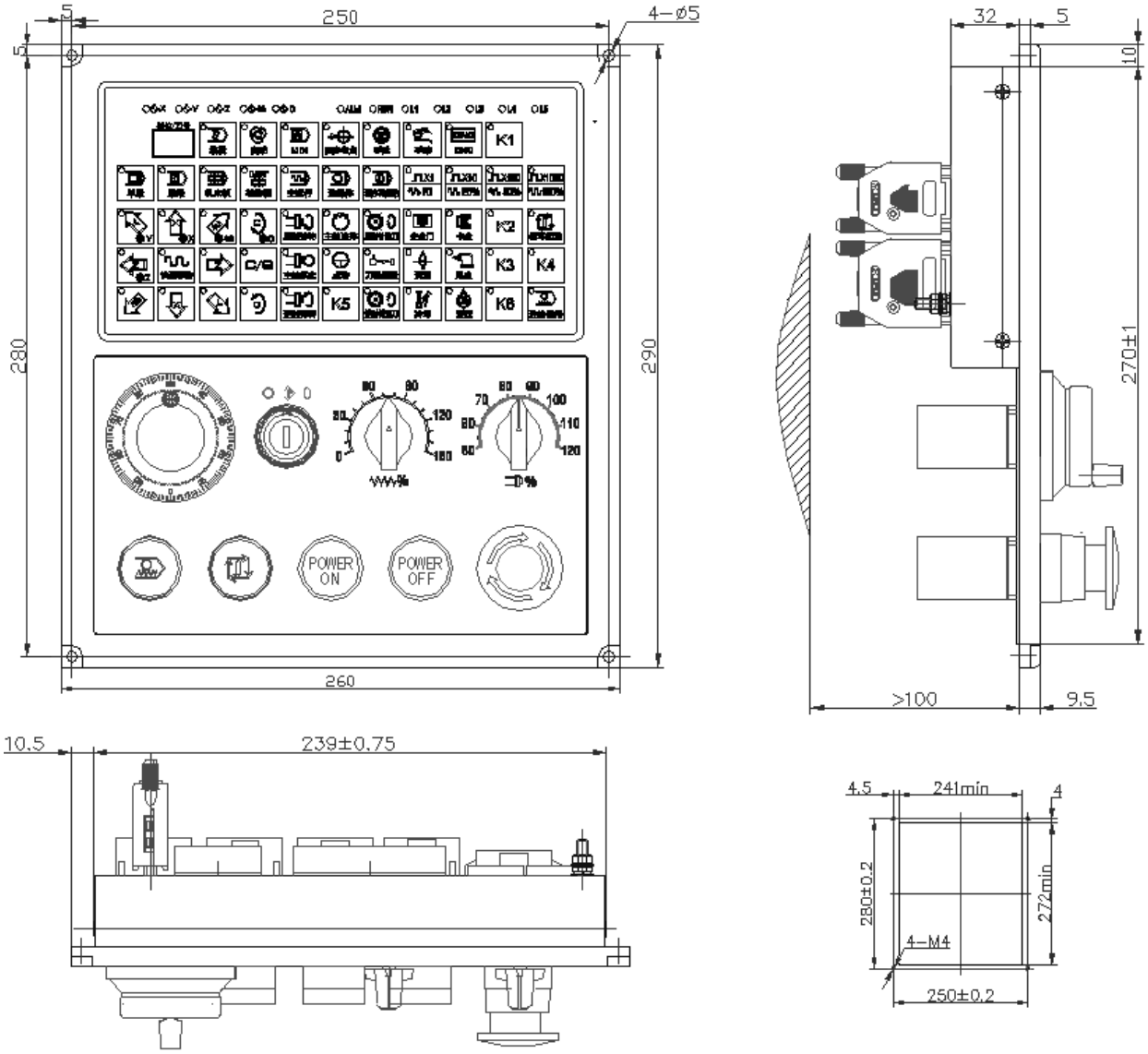


Fig. 5-2 The installation dimension of machine operation panel MPU-08E

Note: The installation dimension of the operation panel MPU-09E is identical with the one of the MPU-08E, which is the different between them is with or without MPG.

Appendix 5.1.2 GSK988TA1-H & Accessory

Appendix 5.1.2.1 GSK988TA1-H Host Appearance Installation Dimension

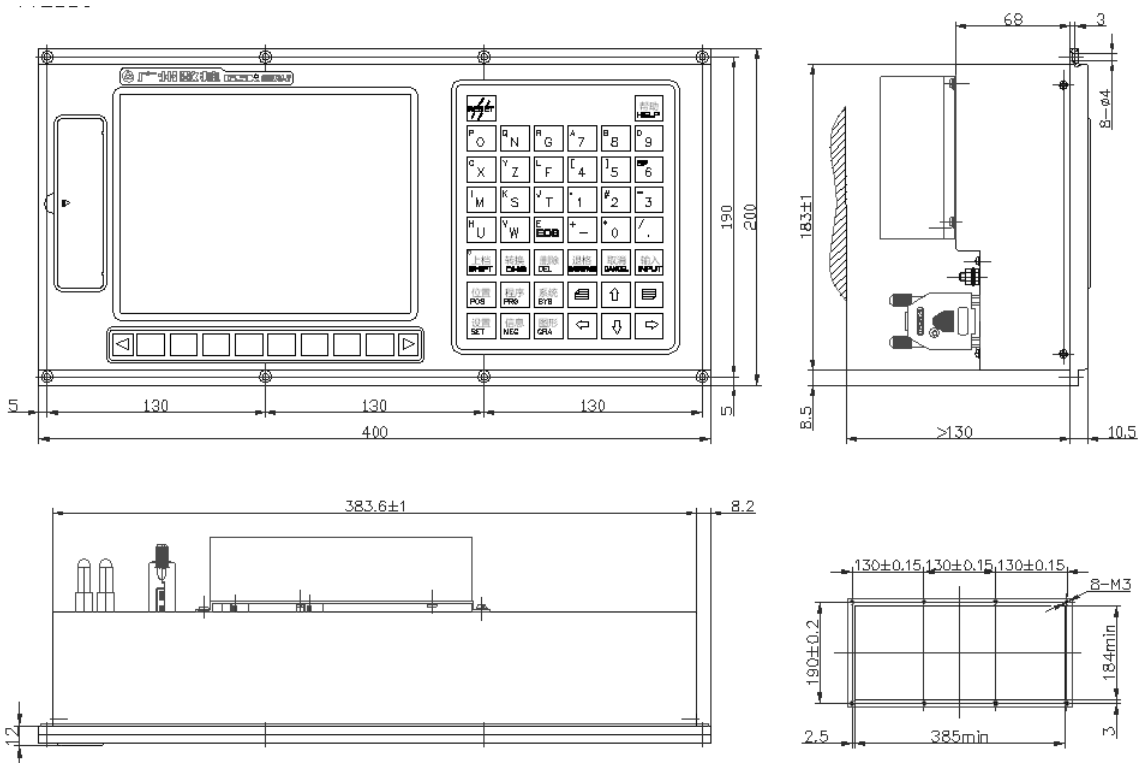


Fig.5-3

Appendix 5.1.2.2 MPU-10E Appearance Installation Dimension of GSK988TA1-H Operation Panel

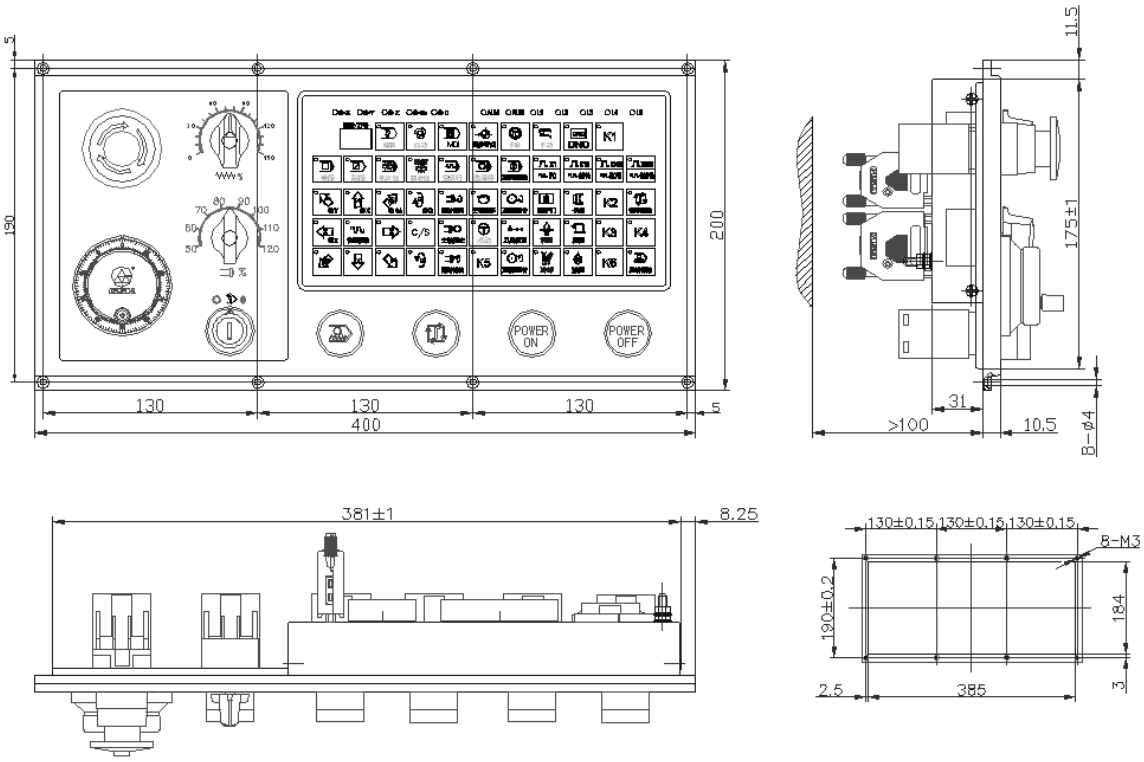


Fig.5-4

Note: The installation dimension of the operation panel MPU-10E is identical with the one of the MPU-11E,

Appendix

which is the different between them is with or without MPG.

Appendix 5.1.3 GSK988TA and its Accessory

Appendix 5.1.3.1 GSK988TA Host Figure Installation Dimension

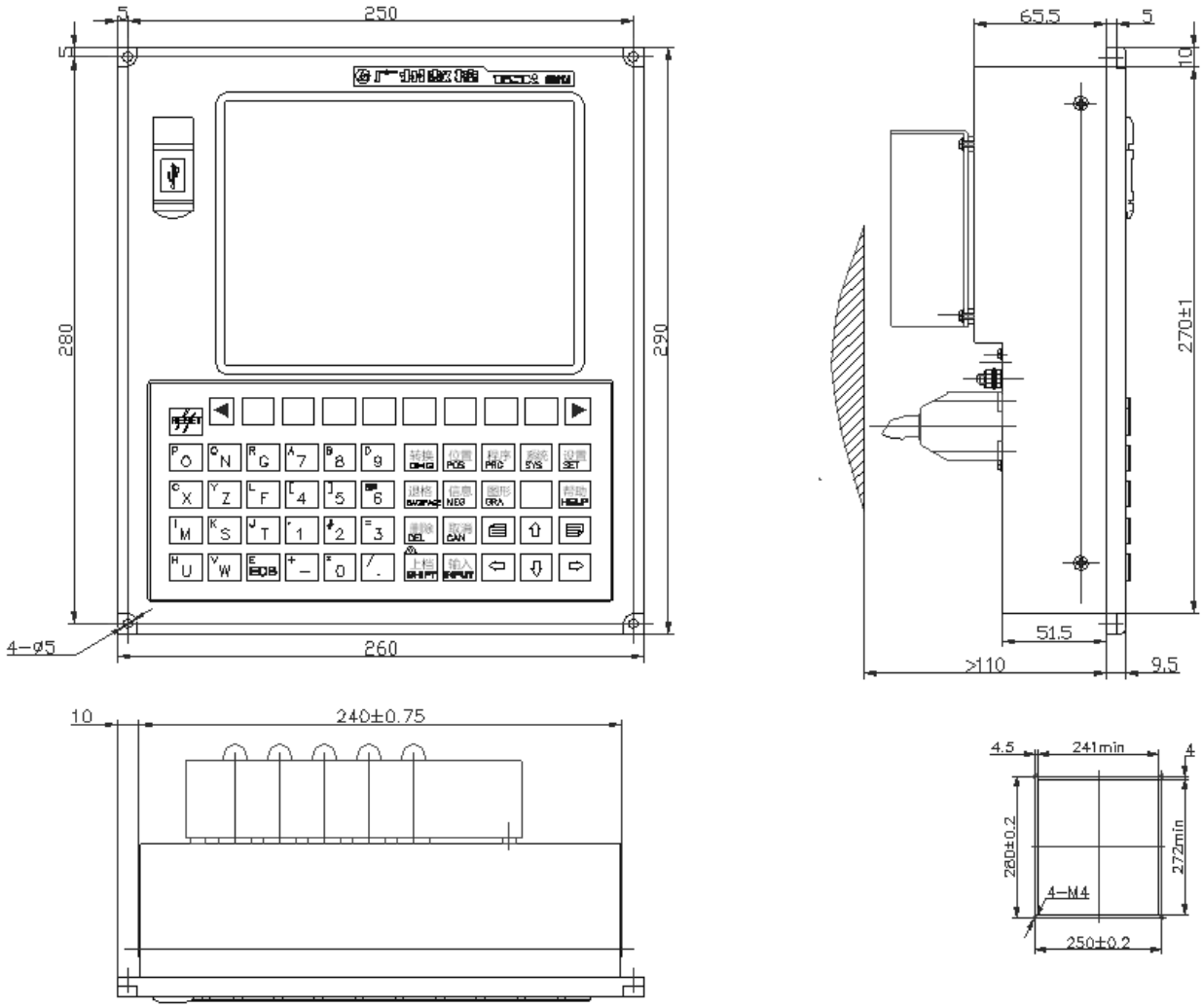


Fig.5-5

Appendix 5.1.3.2 Appearance Installation Dimension of GSK988TA Operation Panel MPU-08

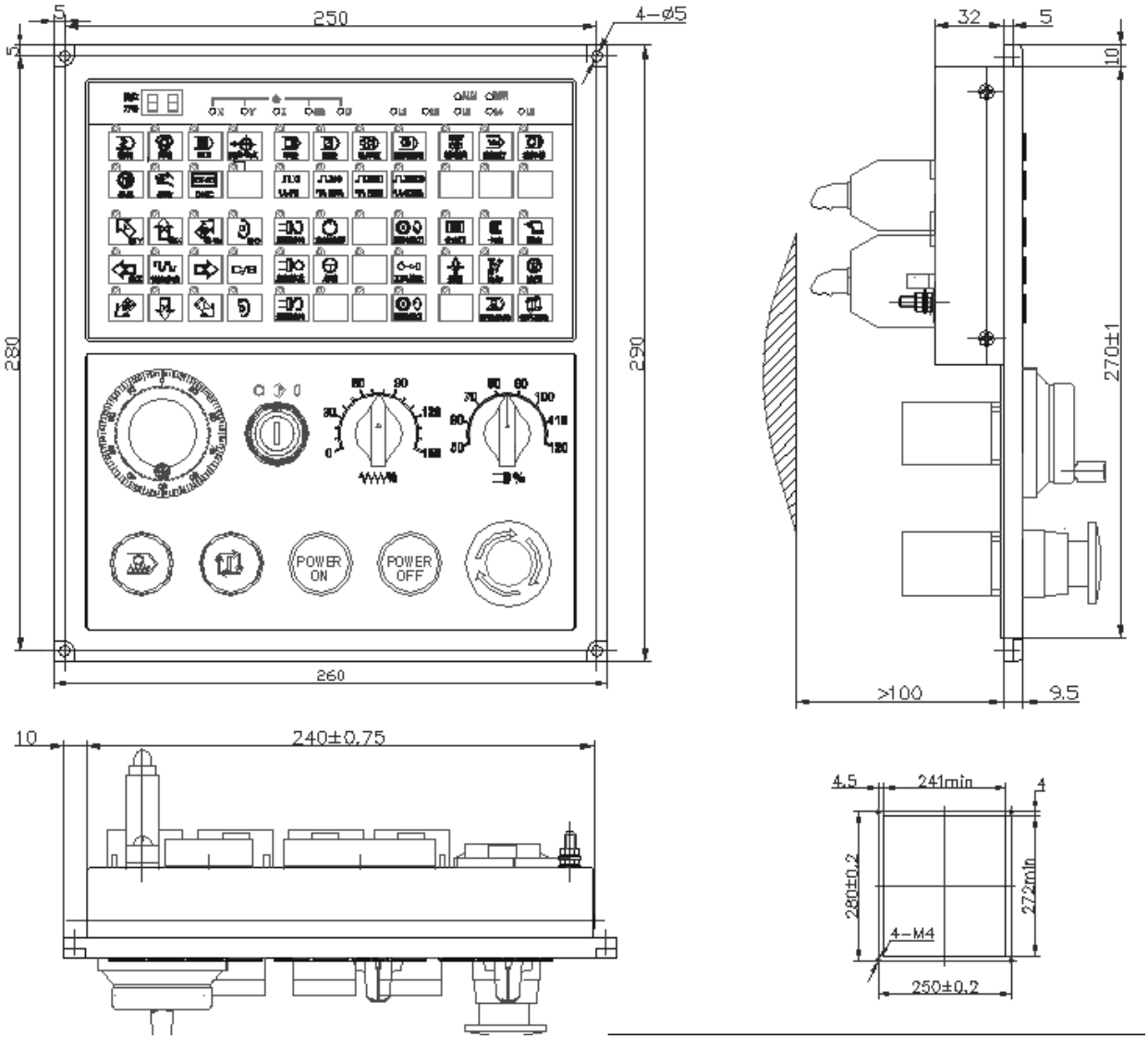


Fig.5-6

Note: The installation dimension of the operation panel MPU-09 is identical with the one of the MPU-08, which is the different between them is with or without MPG.

Appendix 5.1.4 GSK988TA-H & Accessory

Appendix 5.1.4.1 GSK988TA-H Host Appearance Installation Dimension

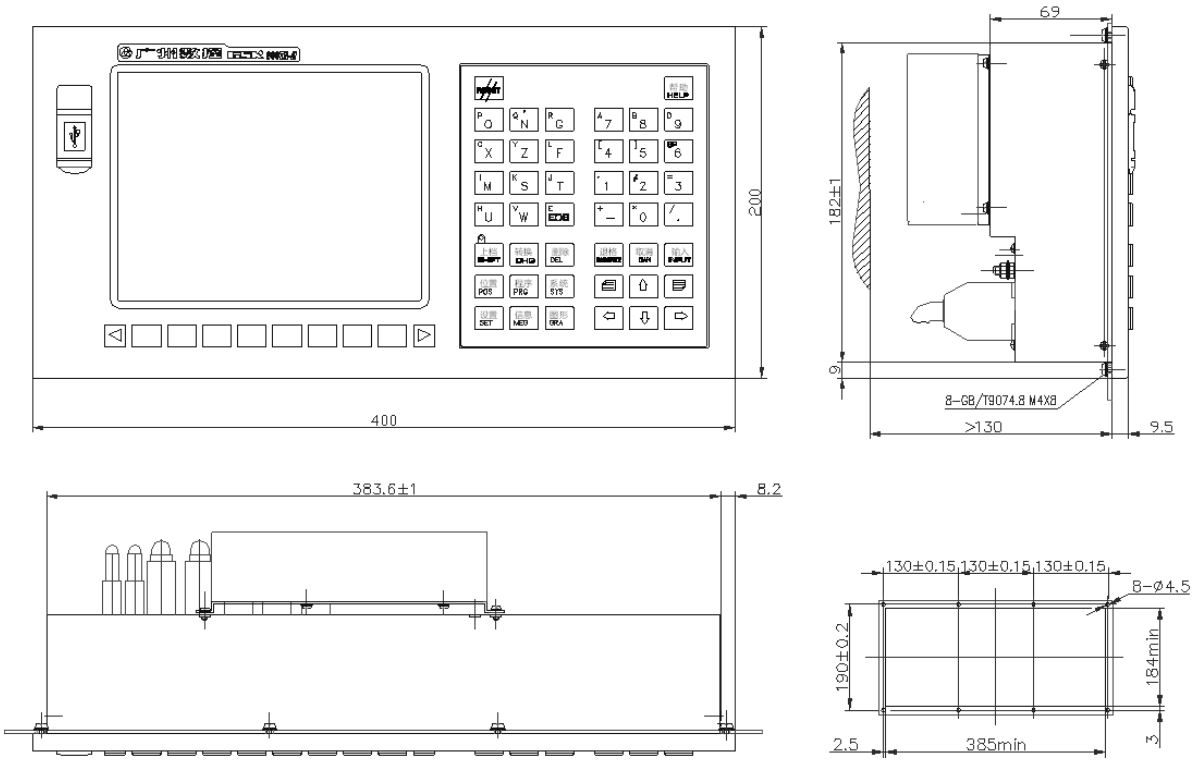


Fig.5-7

Appendix 5.1.4.2 MPU-10 Appearance Installation Dimension of GSK988TA-H Operation Panel

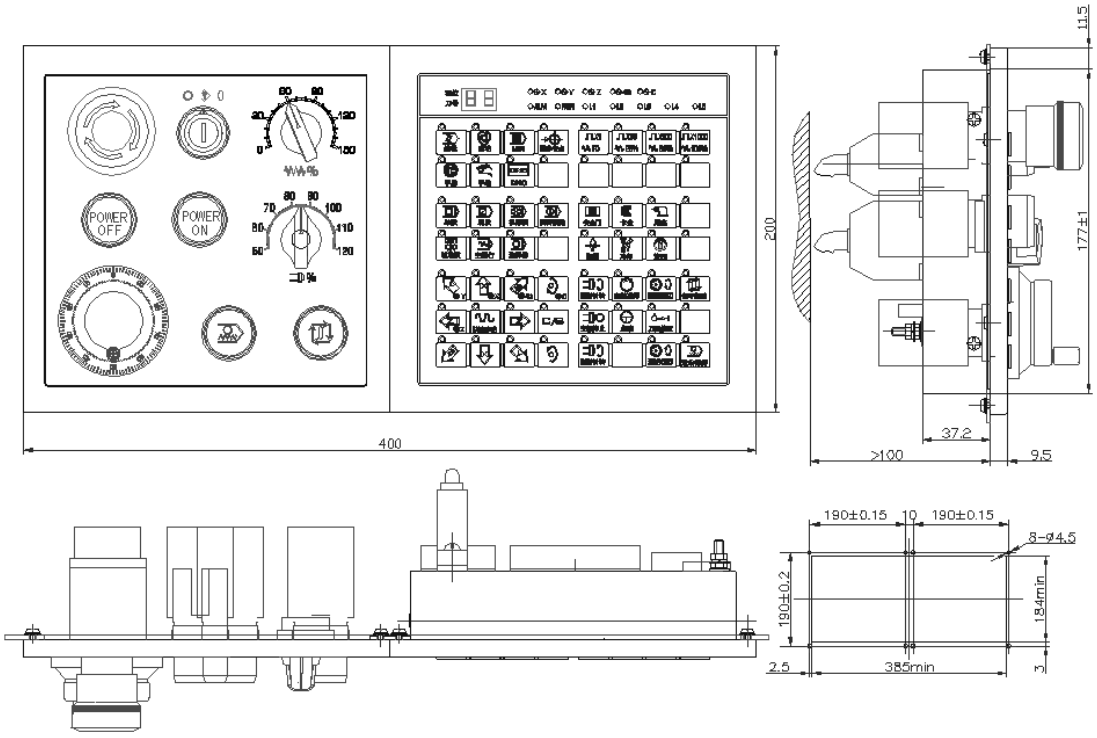


Fig.5-8

Appendix

Note: The installation dimension of the operation panel MPU-10 is identical with the one of the MPU-11, which is the different between them is with or without MPG.

Appendix 5.1.5 GSK988TB and its Accessory

Appendix 5.1.5.1 GSK988TB Host Outline Installation Dimension

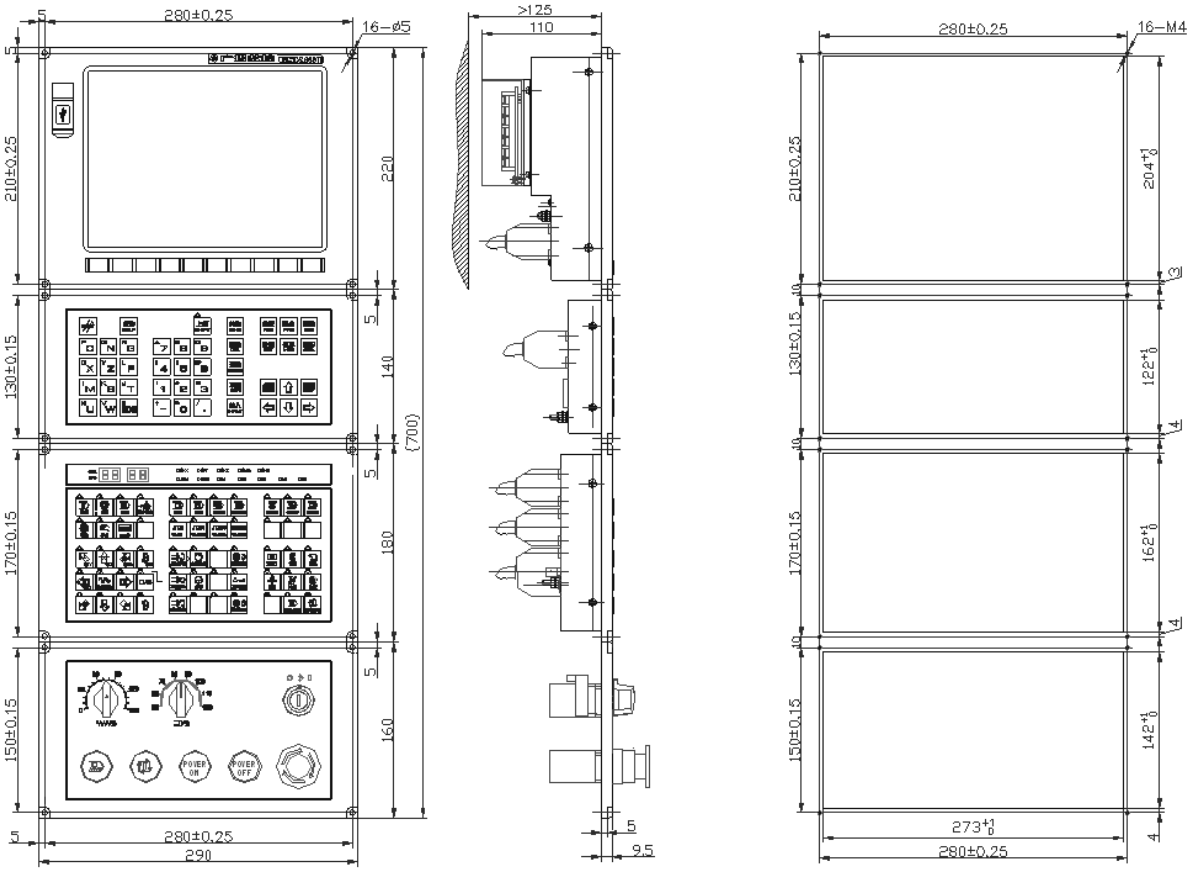


Fig. 5-9

Appendix 5.1.5.2 GSK988TB-H Host Outline Installation Dimension

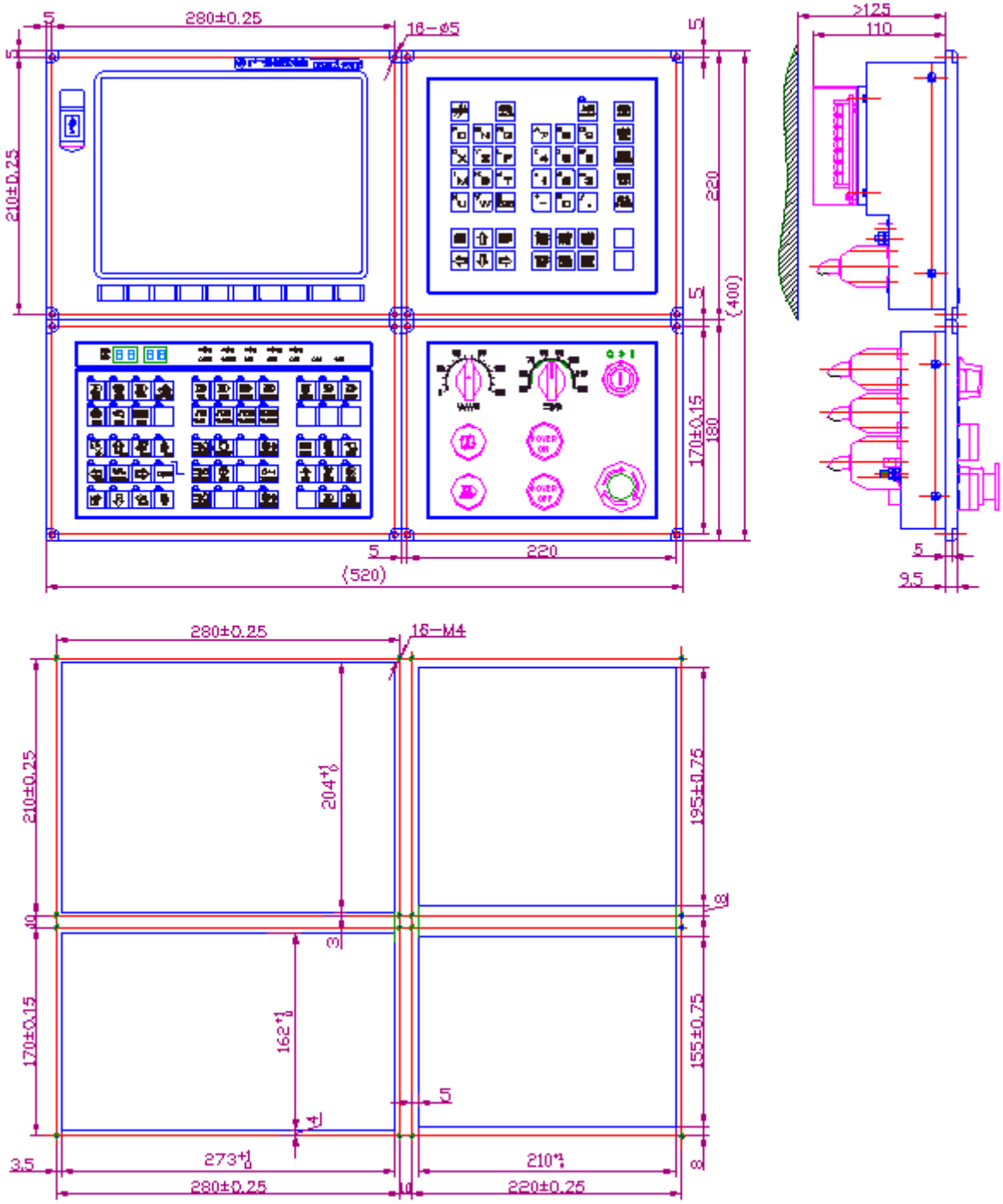


Fig. 5-10

Appendix 5.1.6 I/O Unit Appearance Dimension

Appendix 5.1.6.1 IOL-01T Appearance Dimension

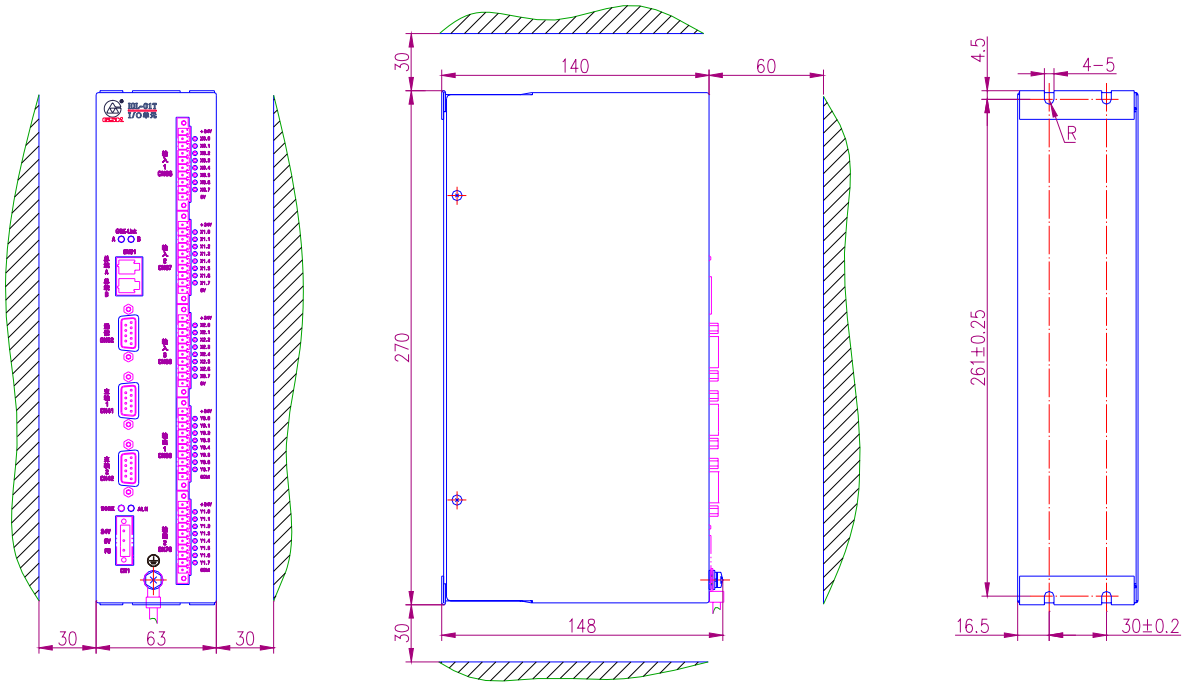


Fig. 5-11

Appendix 5.1.6.2 IOL-02T Appearance Dimension

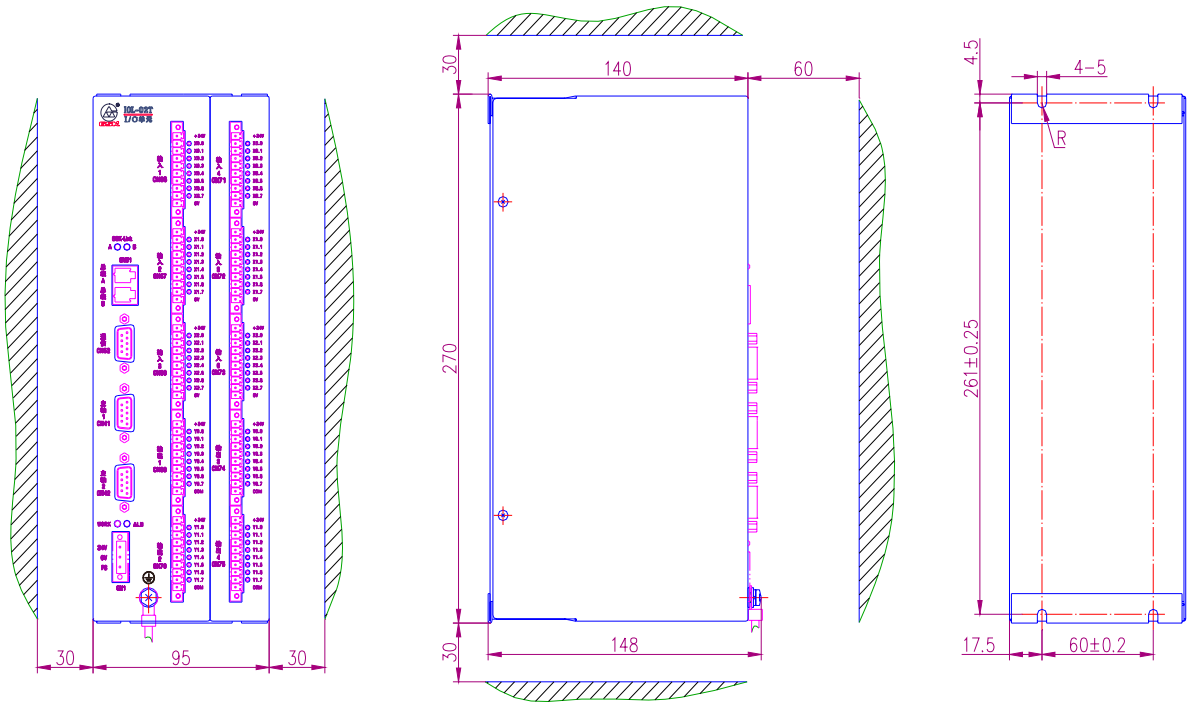


Fig.5-12

Appendix 5.1.6.3 IOL-02F Appearance Dimension

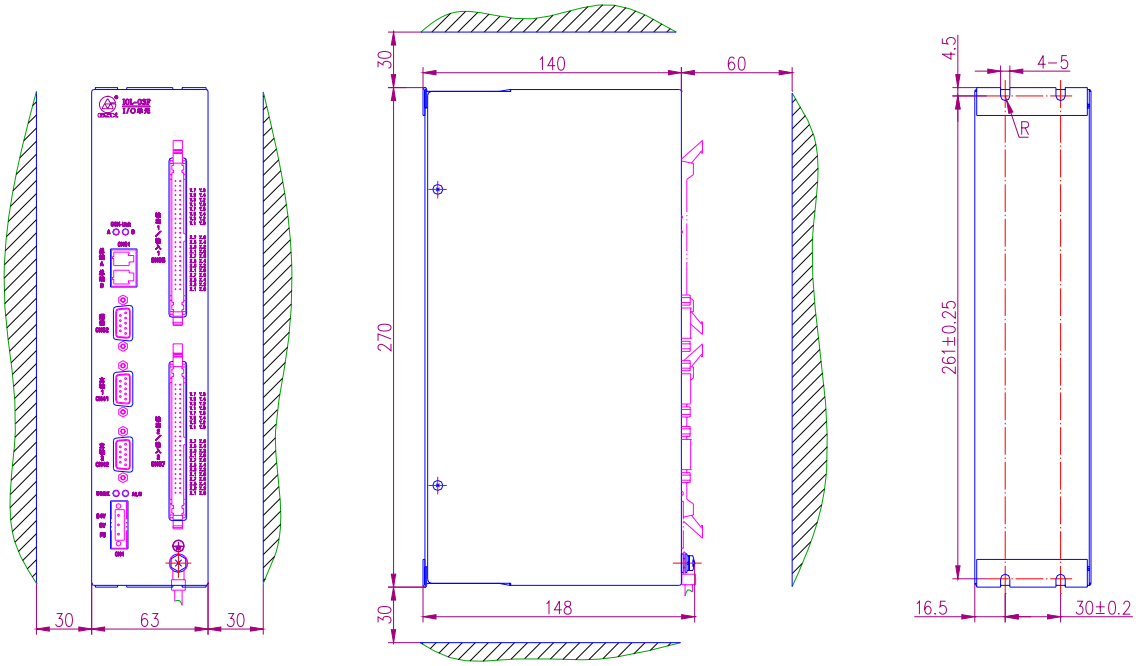


Fig.5-13

Appendix 6 List of Normal Operation

Classification	Function	Operation	Mode	Display page	Password level	Program switch	Parameter switch	Note
Zero clear	X axis relative coordinate zero clear	SET REL, numerical value 0, INPUT		Position				
	Z axis relative coordinate zero clear	SET REL, ⇩, numerical value 0, INPUT		Position				
	Number of the machining work pieces zero clear	CLEAR PART CNT		Position				
	X axis tool offset value zero clear	CLEAR		Setting tool offset	Level 2, 3 and 4			
	Z axis tool offset value zero clear	CLEAR		Setting tool offset	Level 2, 3 and 4			
Setting the data	Word axis type parameter	INPUT, parameter value, INPUT	MDI mode	Parameter	Level 2 and 3		ON	
	Bit axis type parameter	INPUT, parameter value, INPUT	MDI mode	Parameter	Level 2 and 3		ON	
	Macro variable	MEASURE, X axis measured value, OK		Tool offset	Level 2 and 3			
	X axis tool offset input	MEASURE, X axis measured value, OK		Tool offset	Level 2, 3 and 4			
	Z axis tool offset input	MEASURE, Z axis measured value, OK		Tool offset	Level 2, 3 and 4			

Appendix

Appendix 6 List Of Normal Operation

Classification	Function	Operation	Mode	Display page	Password level	Program switch	Parameter switch	Note
	Tool wearing value input	+ INPUT, wearing value, OK		Tool offset	Level 2, 3 and 4			
Search	Search from the program line	LOCATE, line number, OK	Edit mode	Program content	Level 2, 3 and 4			
	Search from the initial line to the bottom	SEARCH, character, FROM TOP			Level 2, 3 and 4			
	Search from the current program to the top	SEARCH, character, PREV			Level 2, 3 and 4			
	Search from the current program to the top	SEARCH, character, NEXT			Level 2, 3 and 4			
	Search for the specified program	SEARCH, program name, OK	Edit mode and Auto mode	Program content	Level 2, 3 and 4			
	Search for the system parameter, servo parameter or the pitch error compensation parameters	SEARCH, Parameter number, OK		System page				
Delete	Delete the characters behind the cursor	DELETE	Edit mode	Program content	Level 2 and 3	ON		
	Delete the characters before the cursor	BACK SPACE	Edit mode	Program content	Level 2 and 3	ON		
	Delete single block	DEL BLK	Edit mode	Program content	Level 2 and 3	ON		
	Delete many blocks	Select many block, DELETE	Edit mode	Program content	Level 2 and 3	ON		

Appendix

Classification	Function	Operation	Mode	Display page	Password level	Program switch	Parameter switch	Note
	Delete the single block	Select the block to be deleted, DELETE	Edit mode	Program directory	Level 2 and 3	ON		
Normal shortcut key	Cursor moved to the file ahead	CHANGE + ⇧	Edit mode	Program content	Level 2 and 3	Cursor moved to the file ahead		
	Cursor moved to the file end	CHANGE + ⇩	Edit mode	Program content	Level 2 and 3	Cursor moved to the file end		
	Cursor moved to the line ahead	CHANGE + ⇦	Edit mode	Program content	Level 2 and 3	Cursor moved to the line ahead		
	Cursor moved to the line end	CHANGE + ⇨	Edit mode	Program content	Level 2 and 3	Cursor moved to the line end		
	Select the arbitrary block	SHIFT + ⇧, ⇩, ⇦, ⇨	Edit mode	Program content	Level 2 and 3	Select the arbitrary block		
	Copy the arbitrary block	CHANGE + P O	Edit mode	Program content	Level 2 and 3	Copy the arbitrary block		
	Cut the arbitrary block	CHANGE + C X	Edit mode	Program content	Level 2 and 3	Cut the arbitrary block		
	Paste the arbitrary block	CHANGE + Q N	Edit mode	Program content	Level 2 and 3	Paste the arbitrary block		
Create	Create a new program	NEW, program name, OK	Edit mode and Auto mode	Program content	Level 2 and 3	ON		
Rename	Rename a program	RENAME, program name, OK	Edit mode	Program directory	Level 2 and 3	ON		
Save as	Save the program as	SAVE AS, program name, OK	Edit mode	Program directory	Level 2 and 3	ON		
Execute	Execute the program	Select the program, LOAD	Edit mode and Auto mode	Program directory	Level 2 and 3	ON		

Appendix

Appendix 6 List Of Normal Operation

Classification	Function	Operation	Mode	Display page	Password level	Program switch	Parameter switch	Note
Setting the switch	ON and OFF of the program switch	ON: ⇦ OFF: ⇨	MDI mode	CNC setting	Level 2 and 3			
	ON and OFF of the parameter switch	ON: ⇦ OFF: ⇨	MDI mode	CNC setting	Level 2 and 3			
	ON and OFF of the automatic sequence number	ON: ⇦ OFF: ⇨	MDI mode	CNC setting	Level 2 and 3			
	Input unit	Metric system: ⇦ Inch system: ⇨	MDI mode	CNC setting	Level 2 and 3			

Note 1: "," in the "operation" column means the operation of two keys should be executed in order, "+" means the two keys should be pressed meanwhile.

Example: INPUT, parameter value, INPUT: It means firstly press INPUT key, and then input the parameter value, finally press INPUT;

CHANGE + Q N: **It means the two keys should be pressed meanwhile.**

Note 2: The blank in the row of the operation mode, the display page, the password level, the program switch or the parameter switch means the corresponding function is irrelative with the ite.

Appendix