PREFACE

Your Excellency：

We are honored by your purchase of GSK980TTi turning machine CNC system manufactured by CNCmakers

The user manual introduces the programming, the operation, the installation and the connection of GSK980TTi turning machine CNC system in details.

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 CNCmakers

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.

- 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.
- The system is with the standard machine operation panel, but the function of each button on the machine panel is defined by the PLC program (ladder diagram). Please pay attention to that the function of the buttons in the manual is described based on the standard PLC programs!
- 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 notic.

Section Ⅰ Programming

Introduce the technical specification, the product introduction, the command code and the program format.

Section Ⅱ Operation

Introduce the operation of CNC system of GSK980TTi turning machine.

Section Ⅲ Installation and Connection

Introduce the installation, connection and setting method of CNC system of GSK980TTi turning machine.

Appendix

Introduce the overall installation dimensions of CNC system of GSK980TTi turning machine and those of its accessories and the alarm information list, etc.

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!

Ⅰ Programming

Ⅱ Operation

Ⅲ Installation & Connection

I Programming

CHAPTER 1 MISCELLANEOUS FUNCTION M\T CODE

Miscellaneous function M code can control the switch about ON and OFF of electric function,it consists of the M code and the 1~4 bit numbers,the following describles only the M code that is solidified in CNC，thest M code is system independent processing.The M code about machine manufacturer design refer to Installation & Connection Manual of the manual of machine manufacturer.

1.1 Miscellaneous Function

Code	Function	Explanation	Remark
M00	Program Stop	After M00 is executed, the program stops and the system displays “Pause”, and then the program continuously runs after the cycle start key is pressed.	
M01	Program Optional Stop	M01 code is controled by the optional stop of panel.	The indicator lights of the optional stop is ON,M01 is executedthe program stops run; The indicator lights of the optional stop is OFF，M01 code is invalid.
M02	End of Program	After M02 is executed,the automatic run stops, the cursor does not return to the start of program.	
M29	entering rigid tapping state	After M29 is executed,system enter to rigid tapping mode.	
M30	End of Program run	After M30 is executed, After M29 is executed, the cursor returns to the start of program（defined by BIT4 of №.005）	
M98 P○○○○□□□□	Subprogram Call	In Auto mode, after other commands are executed in M98, CNC calls subprograms specified by P, and subprograms are executed 9999 times at most. M98 is invalid in MDI mode.	○○○○:Called subprogram number □□□□:Call times
M99 P○○○○	Return From Subprogram	The last program block must specifie M99 or M99 P in the subprogram.	○○○○: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.
M9000～M9999	Macro Program Call	call macro programs corresponding to command values（O9000～O9999）.	
M100～M199	Interchannel waits M code	M code controls wait in the course of processing of Channel 1 and Channel 2. M code used to wait.	

1.2 Tool Function

1.2.1 Tool change and tool offset

Command format:T □□ ○○;（□□：Target tool number（01-32）○○：Tool offset number（00-32））

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

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

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

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

1.2.2 Tool Life Management

Command format:

Txx99: end of current used tool group, start the tool and execute the life management in XX group.

Txx88: cancel the tool offset in XX group

```
…
T0199;          End of previous tool group, and start the tool in 01 group
…
T0188;          Cancel tool offset in 01 group(current used tool offset)
…
T0505;          Use No.05 tool and 08 tool offset without life management
…
T0299;          Start the tools in 02 group
…
T0288;          Cancel tool offset in 02 group(current used tool offset)
…
```

CHAPTER 2 G COMMANDS

2.1 G command list

Word	Group	Function	Remark
G00	01	Rapid traverse movement	Initial modal G command
G01		Linear interpolation	Modal G commands
G02		Circular interpolation(CW)	
G03		Circular interpolation(CCW)	
G05(G05.1)		Three-point circular interpolation	
G6.2		Ellipse interpolation (CW)	
G6.3		Ellipse interpolation (CCW)	
G7.2		Parabola interpolation (CW)	
G7.3		Parabola interpolation (CCW)	
G32		Thread cutting	
G32.1		Rigid thread cutting	
G33		Z tapping cycle	
G34		Variable pitch thread cutting	
G90		Axial cutting cycle	
G92		Thread cutting cycle	
G92.1		Rigid thread cutting cycle	
G83		axle drilling cycle	
G84		End rigid tapping	
G87		radial drilling cycle	
G88		Side rigid tapping	
G94		Radial cutting cycle	
G04	00	Dwell time preset	Non-modal G commands
G7.1		Cylinder interpolation	
G10		Data input	
G11		Data input cancel	
G28		Machine reference point automatic	
G30		Machine 2nd, 3rd , 4th reference point	
G31		Skip interpolation	
G36		Automatic tool compensation X	
G37		Automatic tool compensation Z	
G50		Setting workpiece coordinate system	
G52		Setting local coordinate system	
G65		Macro command	
G70		Finishing cycle	
G71		Axial roughing cycle	
G72		Radial roughing cycle	
G73		Closed c	
G74		Axial grooving cycle	
G75		Radial grooving cycle	

Word	Group	Function	Remark
G76		Multiple thread cutting cycle	
G78		Enhancement thread cutting cycle	
G50.4		Synchronous control release	
G51.4		Synchronous control start	
G50.5		Mix control release	
G51.5		Mix control start	
G50.6		Overlap control release	
G51.6		Overlap control start	
G20	06	inch input	Modal G commands
G21		metric input	
G96	02	Constant Surface Speed Control ON	Modal G commands
G97		Constant Surface Speed Control OFF	Initial modal G command
G98	03	Feedrate per Minute	Initial modal G command
G99		Feedrate per Rev	Modal G commands
G68	04	Balanced Cutting start	Modal G commands
G69		Balanced Cutting cancel	Initial modal G command
G40	07	Cancel cutter radius compensation	Initial modal G command
G41		Tool nose radius compensation left	Modal G commands
G42		Tool nose radius compensation right	
G22	09	Open the stored stroke limit detection 2	Non-modal G commands
G23		Close the stored stroke limit detection	
G66	12	Modal call	Non-modal G commands
G67		Modal call cancel	
G54	14	Workpiece coordinate system 1	Modal G commands
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	
G09	15	Exact stop between blocks	Non-modal G commands
G61		Exact stop begins between blocks	
G64		Cancel the exact stop between blocks	
G17	16	XY plane	Modal G commands
G18		ZX plane	Initial modal G command
G19		YZ plane	Modal G commands
G12.1	21	Polar coordinate interpolation	Non-modal G commands
G13.1		Polar coordinate interpolation cancel	

2.1.1 Modal, One-Shot & Initial Mode

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

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

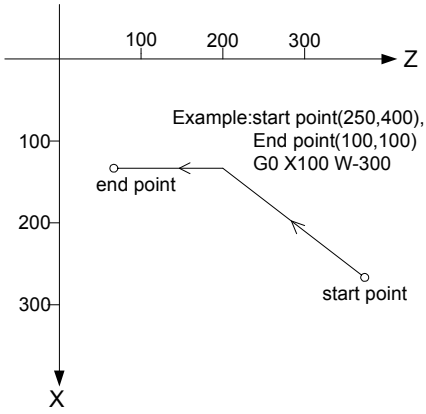
After the system is switched on, the valid modal G commands which are not executed their functions or states are called initial mode G command. Take it as the initial mode G command to be executed when it is not be input after the system is switched on.

2.2 Rapid Traverse Movement G00

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

Command specification:X(U)、Z(W) end point coordinates;

Command function: X, Z rapidly traverses at the respective traverse speed to the end points,the short axis arrives the end point,and the length axis continuously moves to the end point and the compound path may be not linear.

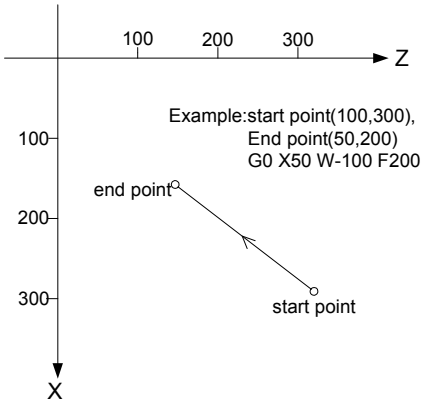


2.3 Linear Interpolation G01

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

Command specification: X(U)、Z(W) end point coordinates;F:command value is the vector compound speed of X and Z.

Command function: The movement path is a straight line from starting point to end point.



2.4 Plane Selection G17～G19

Command format：

G17……XY plane

G18……ZX plane

G19……YZ plane

Command function: use G commands to select the plane of the arc interpolation or the one of the cutter compensation.

Command explanation: G17, G18, G19 are modal, and the plane does not change in the block without the command.

Notes:

1) **Firstly set the basic axis Y when the system selects G17, G19 plane;**
2) **Cannot switch the planes in C tool compensation;**
3) **G71~G76，G90，G92，G94 can be used in G18 plane;**
4) **The plane selection code can be in the same block with G codes in the other groups;**
5) **The movement command is not relevant to the plane selection;**
6) **Diameter or radius programming: currently, because there is only one bit parameter No 1.2 to select the diameter or the radius programming and is valid to only X axis, Z and Y axis use the only radius programming in G2, G3, and X axis is selected by the parameter;**
7) **The tool nose direction of C tool compensation is 0 in G17, G19;**
8) **Angel Q in the oval and parabola command, in G18 panel, it is an angle along Y axis in G19, an**

angle along X axis in G17;

9) The relevant instruction in this manual is base on G18 panel,when select G17 or G19 panel，the address of command is replace to according to selecting panel：

G17 { G02 / G03 } X(U)__ Y(V)__ { R__ / I__ J__ } F__ G19 { G02 / G03 } Y(V)__ Z(W)__ { R__ / J__ K__ } F__

2.5 Circular Interpolation G02, G03

Command format:

{ G02 / G03 } X(U)__ Z(W)__ { R__ / I__ K__ } F__;

Command specification:

G02:CW direction;

G03:CCW direction;

X、Z:end point absolute coordinates;

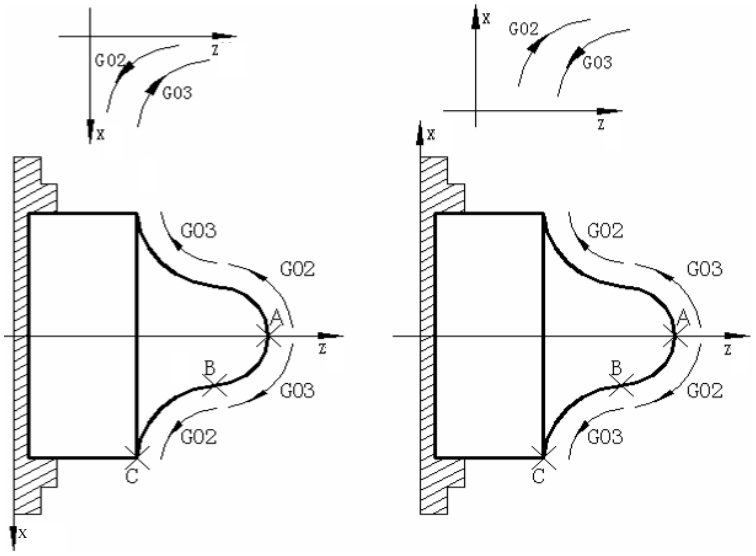
U、W:the increment value from starting point to end point;

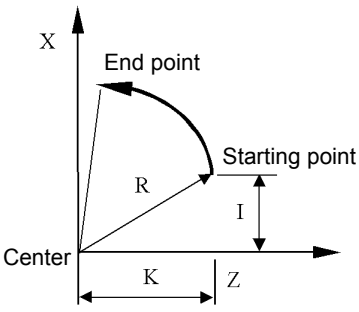
R:arc radius;

I、K:the vector from starting point to center point of arc;I, K are with sign symbol. When directions of I, K are the same as those of X, Z, they are positive, otherwise, they are negative;

F:Cutting federate.

Command path:





2.6 G02, G03 Helical Interpolation(optional function)

Command format:

G02 / G03 X(U)__ Z(W)__ { R__ / I__ K__ } Y(V)__ F__;

Command specification:

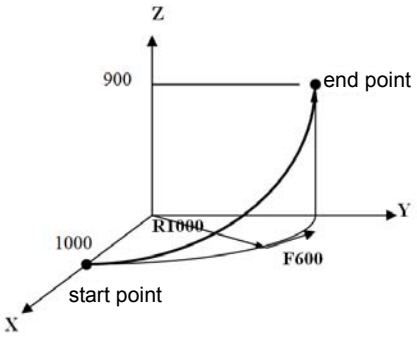
When there is displacement int the third axis perpendicular to the arc plane,The action of G02/G03 is helical interpolation.

X、Z:End point coordinates of arc;

Y:Linear end point coordinates;

R:arc radius;

I、K:center point coordinates of arc;



G17 G03 X0 Y1000 R1000Z900 F600;

2.7 Three-point Circular Interpolation G05、G05.1

Command format:

G05 / G05.1 X(U)__ Z(W)__ I__ K__ F__;

Command specification: X、Y、Z are end point absolute coordinates;

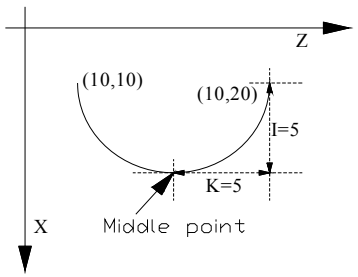
U、V、W are the increment value from starting point to end point;

I: incremental coordinate value(X) (radius value, direction) of the middle point where the circular passes corresponding to the starting point, range:±99999999×least input increment;

K: incremental coordinate value(Z, direction) of the middle point where the circular passes corresponding to the starting point, range:±99999999×least input increment;

Command function:G05(G05.1) is modal; coordinates of three points on the arc, you can use G5 to confirm the arc direction through the middle position.

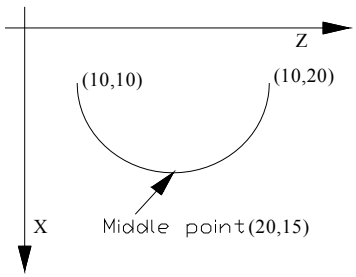
Example:



Program 1

G00 X10 Z20;

G05 X10 Z10 I5 K-5 F150;

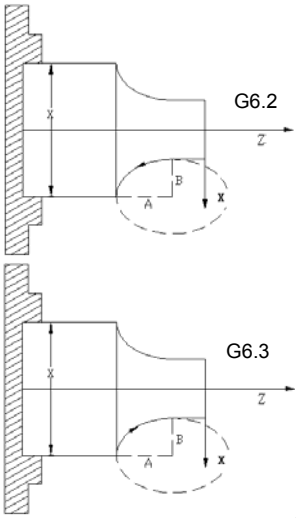


Program 2

G00 X10 Z20;

G05.1 X10 Z10 I20 K15;

2.8 Ellipse Interpolation G6.2, G6.3



Command format:

G6.2 / G6.3 } X(U)__ Z(W)__ A__ B__ Q__;

Command specification:G6.2:CW ellipse interpolation;
G6.3:CCW ellipse interpolation;

X、Z:end point absolute coordinates;

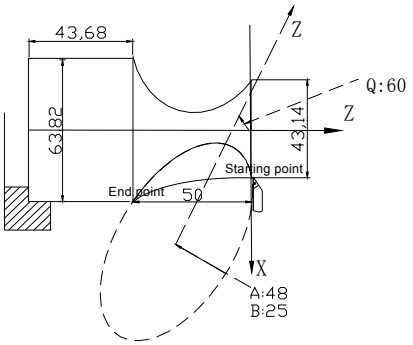
U、W:the increment value from starting point to end point;

A: Length of ellipse's long radius;

B: Length of ellipse's short radius;

Q: Q value is the gained viewing XZ plane along with the positive direction of Y axis in the Cartesian right-hand coordinate system, as well, it is an angle when the Z axis positively rotates to the the overlapping with the ellispse long axis along with the CW.

Example: machine from Φ43.14 to Φ63.82:



Program:

G6.2 X63.82 Z-50.0 A48 B25 Q60000;

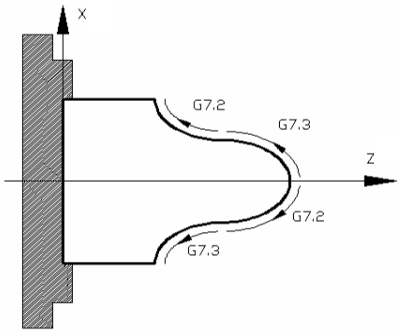
or

G6.2 U20.68 W-50.0 A48 B25 Q60000;

2.9 Parabola Interpolation G7.2, G7.3

Command format:

G7.2 / G7.3 } X(U)__ Z(W)__ P__ Q__;



Command specification:G7.2:CW direction;
G7.3:CCW direction;

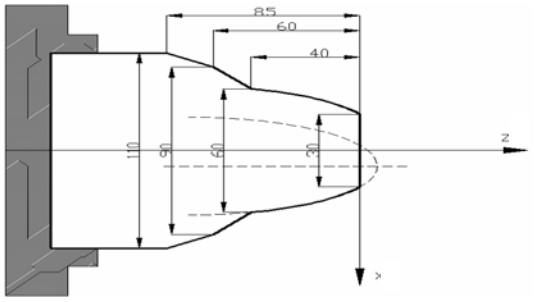
X、Z:end point absolute coordinates;

U、W:the increment value from starting point to end point;

P:P is in the parabola standard equation $Y^2=2PX$;

Q:Q value is the gained viewing XZ plane along with the positive direction of Y axis in the Cartesian right-hand coordinate system, as well, it is an angle when the Z axis positively rotates to the the parabola with the ellispse long axis along with the CW.

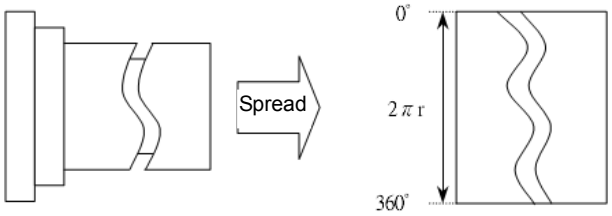
Example: when the parabola P=100(the least increment is 0.0001mm), its symmetrical axis is parallel with Z. Its machining sketch map and programming are as follows:

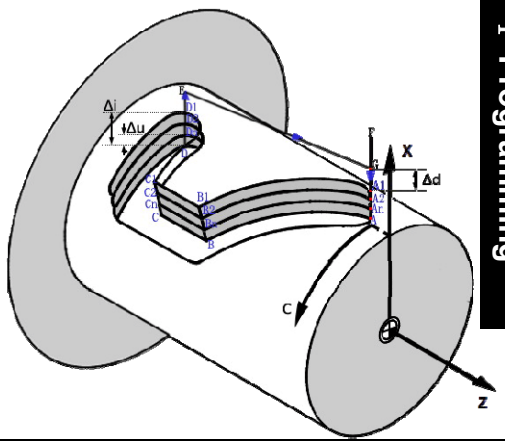



```
G00 X120 Z100 T0101 M03 S800;
G00 X0 Z0 M08;
G01 X30 F200;
G7.3 X60 Z-40 P100000 Q180000;
G01 X90 Z-60;
X110 Z-85;
X120;
G00 X120 Z100;
M30;
```

2.10 Cylindrical Interpolation G7.1

The cylindrical interpolation function allows the side of a cylinder to be developed for programming (below).





Command format:

Type I	Type II
G07.1 C(c); (1) starting the cylindrical interpolation	G07.1 C(c) J(Δi) L(d) R(Δd) K(Δu);(1) starting the cylindrical interpolation
…	…
… (2) cylindrical interpolationp block interval	… (2) cylindrical interpolation roughing block interval
…	…
G7.1 C0;(3) cancel the cylindrical Interpolation	G7.1 C0;(3) cancel the cylindrical interpolation

Command specification:

G7.1 is non-modal;

The feedrate in the cylindrical interpolation mode specifies G98(feed per minute).

C(c):the radius of the cylinder; Radius ≠0:starts the cylindrical interpolation mode;

Radius＝0:cancels the cylindrical interpolation mode.

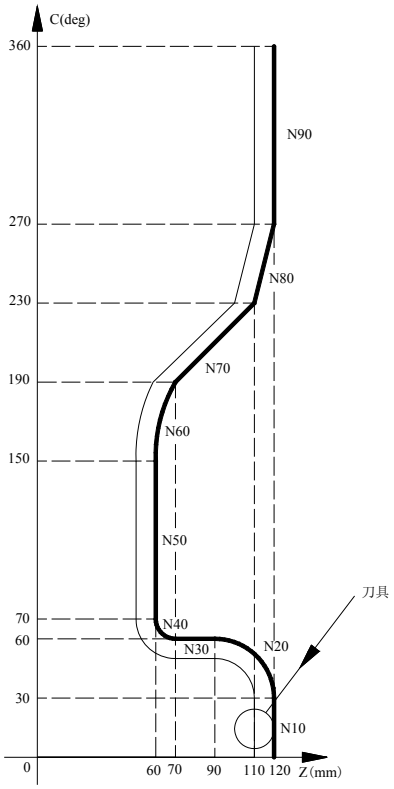
J(Δi):Total cutting value along X axis in the rough-turning, diameter,with sign;

R(Δd):Tool-retraction value along X axis in the rough-turning;

K(Δu):Finish-turning allowance along X axis;

L(d):Cutting times(without including the finish-turning path).

Example: The radius of the cylinder is 57.299mm, and the developed cylindrical surface is shown below:



Type I：

O0071（Cylinder interpolation G7.1 application example）
G18;
G98;
G00 X150 Z105 C0;
T0101;
G01 X114.598 Z105 F200;
G07.1 C57.299
G41 G01 Z120;
N10 G01 C30.0;
N20 G03 Z90 C60 R30;
N30 G01 Z70;
N40 G02 Z60 C70 R10;
N50 G01 C150;
N60 G02 Z70 C190 R75;
N70 G01 Z110 C230;
N80 G03 Z120 C270 R75;
N90 G01 C360;
G40 G01 Z105;
G07.1 C0;
M30;

Type II：

O0071（Cylinder interpolation G7.1 application example）
G18;
G98;
G00 X150 Z105 C0;
T0101
G01 X114.598 Z105 F200;
G07.1 C57.299 J-10 K0.25 R2 L4;
G41 G01 Z120;
N10 G01 C30.0;
N20 G03 Z90 C60 R30;
N30 G01 Z70;
N40 G02 Z60 C70 R10;
N50 G01 C150;
N60 G02 Z70 C190 R75;
N70 G01 Z110 C230;
N80 G03 Z120 C270 R75;
N90 G01 C360;
G40 G01 Z105;
G07.1 C0;
M30;

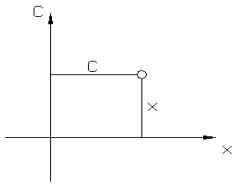
Note: In the progrmming example of the type II, the overall cutting value is 10mm, the finish machining residual is 0.25mm, tool is 2mm distant from the workpiece along with X direction, the rough-turning cycle times is 4, finish-turning cycle times is 1; tool retreats from the start position (116.598,105,0) after the rough-tuning cutting is performed each time.

2.11 Polar Coordinate Interpolation G12.1, G13.1

Command format:

Type I： rectangular coordinate programming

G12.1; start polar coordinate interpolation
……
……
G13.1; cancel polar coordinate interpolation



X:distance of linear axis, unit: mm/inch;
C:distance of rotary axis, unit: mm/inch;
The linear axis can use the diameter or radius programming and the rotary axis uses only the radius.

Type II：polar coordinate programming

G12.1; start polar coordinate interpolation

……;

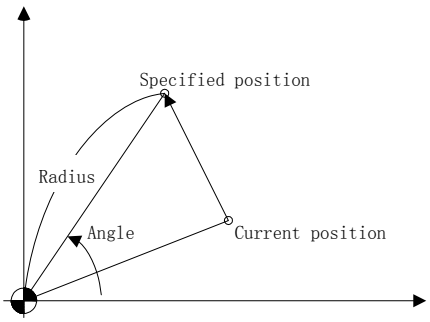
G16; start polar coordinate programming

……;

G15; cancel polar coordinate programming

……

G13.1; cancel polar coordinate interpolation



Example:

O0000(O0000);

T0101;

G0 X80 C0 W0;

G12.1;

G6.3 X0 C20 A40 B20 F1000;

G2 X-10 C15 R5;
G3 X-10 C-15 R15;
} replace to →
G16
G2 X15.8114 C108.435 R5
G3 X15.8114 C251.565 R15
G15-------cancel the above programming mode and the followings are Cartesian coordinate programming

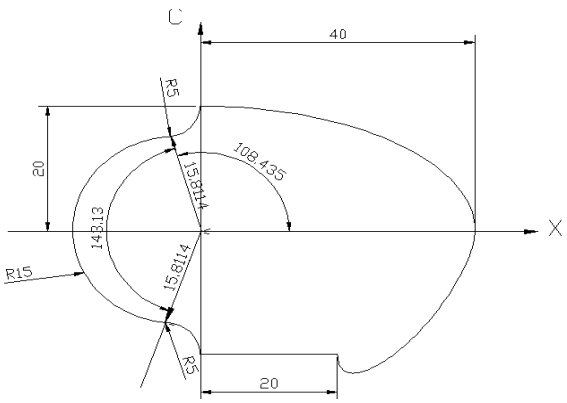
G2 X0 C-20 R5;

G1 X40 C-20;

G7.3 X80 C0 P100000 Q0;

G13.1;

M30;



2.12 Machine Zero Function

2.12.1 Machine 1st Reference Point G28

Command format:G28 X(U)__ Z(W)__;

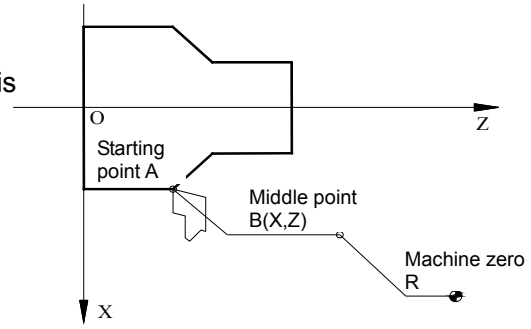
Command specification: G28 is non-modal; the tool rapid traverses to the middle point defined by X(U), Z(W) from starting point and then return to the machine zero.

X、Z:absolute coordinates of middle point;

U、W:Difference value of absolute coordinates between middle point and starting point.

Running path:

1. Rapid traverse to middle point of specified axis from current position(A point→B point) ;
2. Rapid traverse to reference point from the middle point(B point→R point) ;
3. LED is ON when the machine reference point return is completed.



2.12.2 Machine 2nd, 3rd, 4th Reference Point G30

Command format:

G30 P2 X(U) Z(W) ;
G30 P3 X(U) Z(W) ;
G30 P4 X(U) Z(W) ;

Command specification: G30 is non-modal. the tool rapidly traverses with the rapid traverse speed to the middle point specified by X(U) , Z(W). when P is omitted, returning to machine 2nd reference point.Execute the machine 2nd, 3rd, 4th reference point return after you manually execute the machine reference point return or G28 (machine reference point return).

X:X absolute coordinate of the middle point;

U:difference value of X absolute coordinate value between the middle point and starting point;

Z:Z absolute coordinate of the middle point;

W:difference point of Z absolute coordinate between the middle point and starting point.

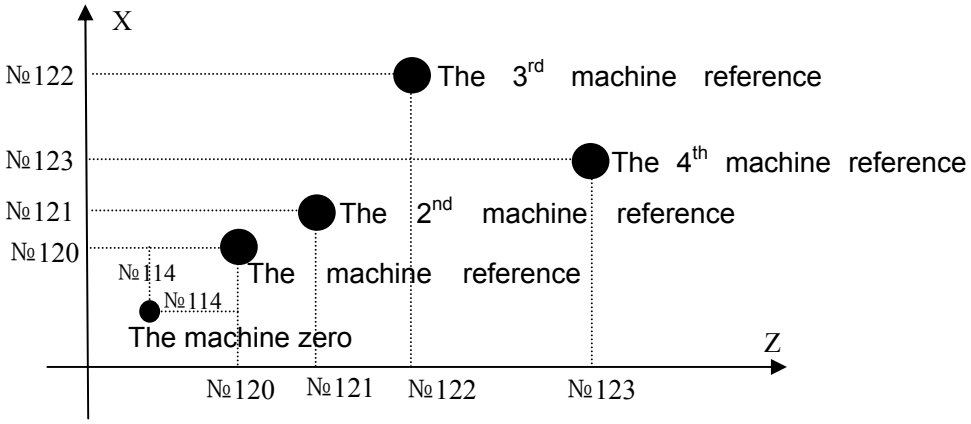
reference point setting:

The coordinates of machine 2nd reference point，set by No.121;

The coordinates of machine 3rd reference point，set by No.122;

The coordinates of machine 4th reference point，set by No.123;

machine 2nd, 3rd, 4th reference point is as follows:



2.13 Skip Interpolation G31

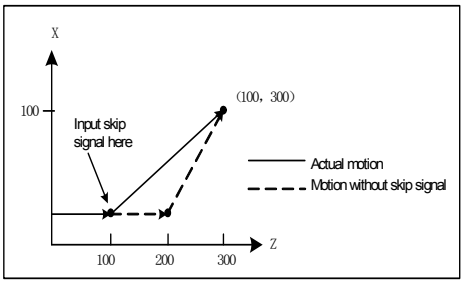
Command format:G31 X(U)_ Z(W)_ F_ P_;

Command specification:G31 is non-modal G command (00 group);

X、Z:End point;

U、W:the increment value from starting point to end point;

P:Skip signal address selection, its range:1～4 The skip signal address from the corresponding P1～P4 are set by parameters №148～№151, it is regarded as 1 by default when the P value does not input.



Function:in executing the command, when the outside skip signal (X3.5) is input, the system stops the command 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.

Program：
G31 Z200 F100;
G01 X100 Z300;

2.14 Automatic Tool Offset G36, G37

Command format:

G36 X__; Automatic Tool Offset in X direction,Only use the absolute programming;

G37 Z__; Automatic Tool Offset of in Z direction ,Only use the absolute programming;

Command specification: when the command is executed to make the tool move to the measured position, the CNC automatically measures the difference between the current actual coordinates and the command coordinates to be the tool offset value.

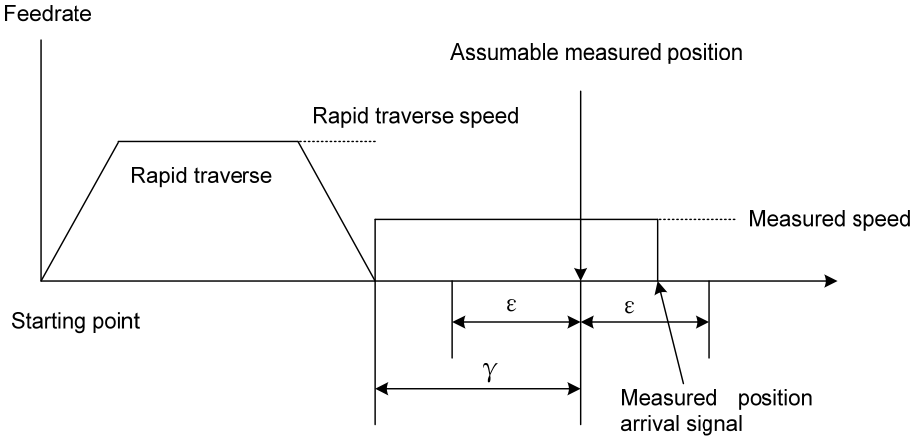
Measured position arrival signal:

XAE(X3.6) ————corresponding to G36

ZAE(X3.7) ————corresponding to G37

Operations is shown below:

When G36X_(or G37Z_) is executed, the tool firstly rapidly traverses to the position measured by the command, and decelerates and temporarily stop the position before the measured position, and then, reaches to the measured position at the speed set by No.141.When the measured position arrival signal corresponding to G command becomes “1”, and the tool is in the measured position range ±ε, CNC 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.



2.15 Workpiece Coordinate System G50

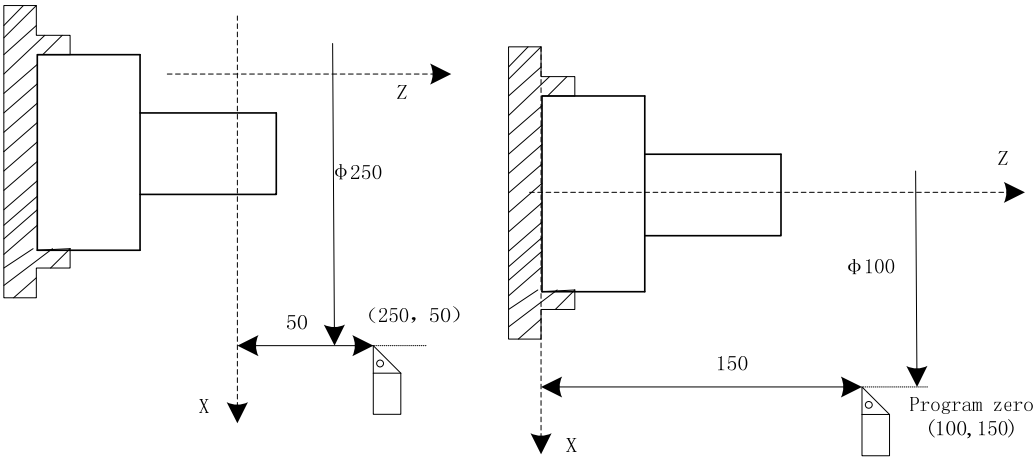
Command format:G50 X(U)__ Z(W)__;

Command specification: G50 is non-modal;

X、Z:New absolute coordinates of current position;

U、W:Different value between the new absolute coordinates of current position and the absolute coordinates before executing commands.

Function:define the absolute coordinates of current position and create the workpiece coordinates system (called floating coordinates system) by setting the absolute coordinates of current position in the system. In G50, when X(U) or Z(W) are not input, the system does not change current coordinates position as program zero.



Before setting coordinate system with G50 After setting coordinate system with G50

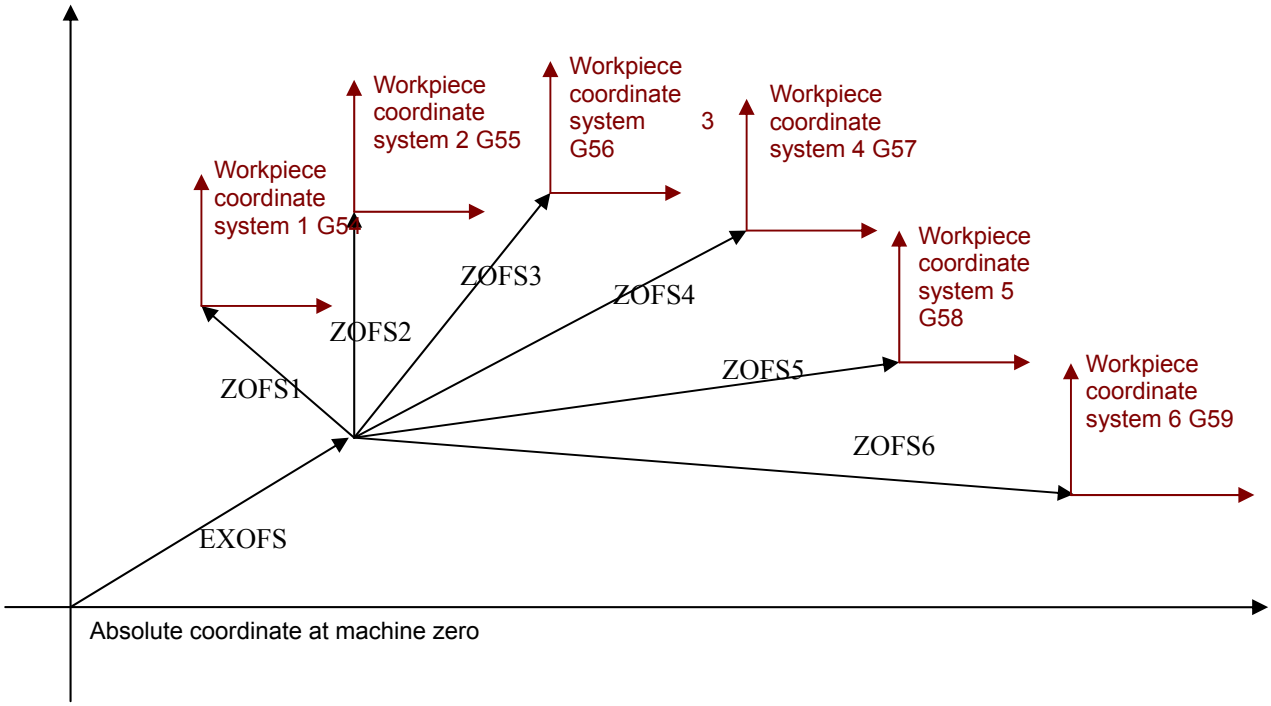
create the above-mentioned workpiece coordinate system and set (X100 Z150) to the reference point of program after executing “G50 X100 Z150”.

2.16 Workpiece Coordinate System G54～G59

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 specification: G54～G59 are modal, each coordinate system should correspond to ZOFS1~ ZOFS6 of zero offset value of one workpiece coordinate system, relation between ZOFS1~ ZOFS6 and previous workpiece is shown below:



Notes：

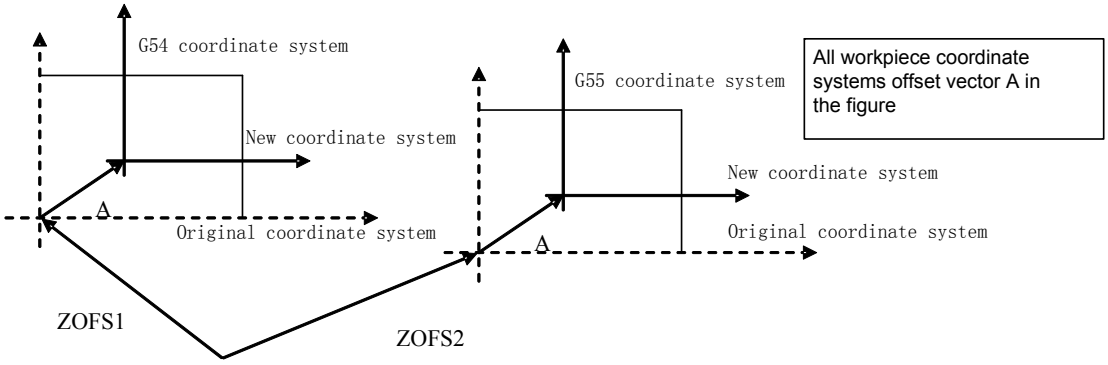
1:When №11.7(APRS) is set to 1, an absolute coordinate system is set after machine zero return, and then EXOFS and ZOFSn setting values are offset. new workpiece coordinates=current absolute coordinates-(ZOFSn + EXOFS).

2:When №12.7(APRS) is set to 0, the absolute coordinate system after machine zero return is not set, and the EXOF and ZOFSn offset cannot be executed because the workpiece coordinate system is not set again.

3:Using the G54～G59,G50 can not be used.When G50 sets the coordinate system, origin of the current workpiece system is modified. So, there is an offset value of origins between a new coordinate system set by G50 and previously current coordinate system.

4:G54～G59 workpiece coordinate system switch.

Specifying G54~G59 can switch 6 workpiece coordinate systems to make the system work in different workpiece systems. Absolute coordinate variation of current position is the origin offset value between the new workpiece coordinate system and the old. Namely:new absolute coordinates =current absolute coordinates–(ZOFS new – ZOFS old),Offset value A are acting on all workpiece coordinate systems, which are shown below:



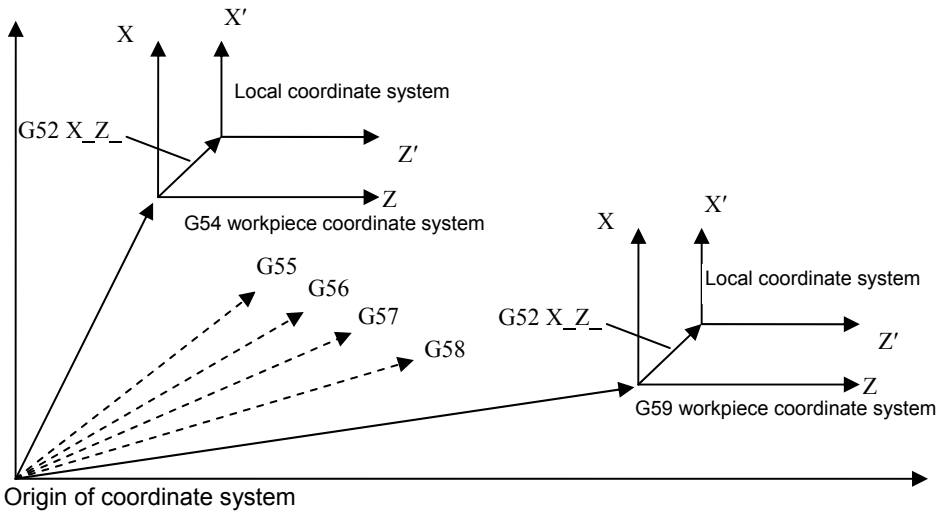
2.17 Local Coordinate System G52

Command format:G52 X(U)__ Z(W)__;

Command specification: G52 is in Group 00.

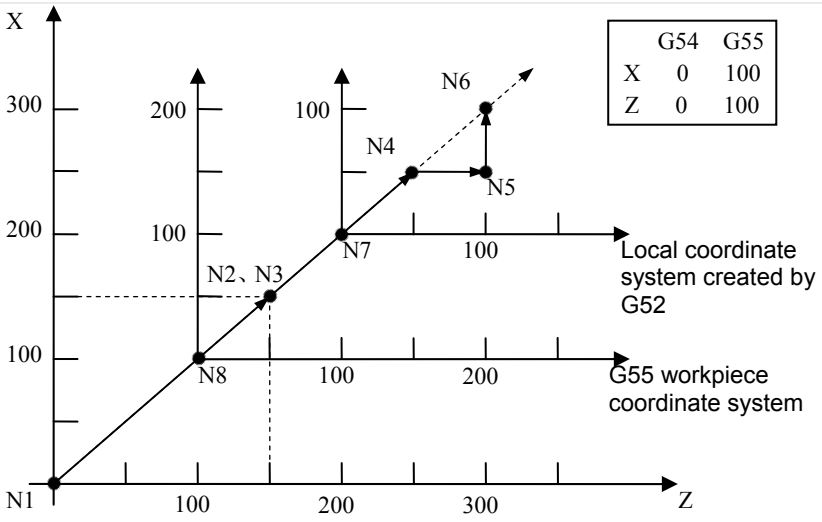
- X(U)__ Z(W)___is the position of origin of the specified local coordinate system in the current workpiece coordinate system.
- when G52 is executed, all workpiece coordinate system can (G54～G59) set a local coordinate system. Origin of each local coordinate system is the specified position X(U) Z(W)__ of each workpiece coordinate system.
- When a local coordinate system is cancelled, its zero and that of workpiece coordinate system are consistent, i.e., command G52 X0 Z0 or G52 U0 W0, When G50 sets a workpiece coordinate system, local coordinate systems of all workpiece coordinate systems which axes are specified are cancelled.

Corresponding relation of workpiece coordinate systems is shown below:



Example:

```
N1 G54 X0 Z0;
N2 G55 X50 Z50;
N3 G52 X100 Z100
（local
coordinate system
created）
N4 G00 X50 Z50;
N5 G01 Z100 F100;
N6 X100;
N7 G52 X0 Z0;（local
coordinate system
created cancelled）
N8 G00 X0 Z0;
N9 M30;
```



2.18 Axial Cutting Cycle G90

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

Command specification: G90 is modal;

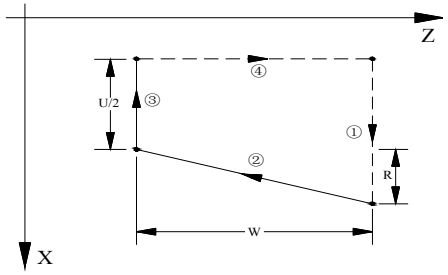
X、Z:absolute coordinates of cutting end point;

U、W:Different value of absolute coordinate between end point and starting point of cutting;

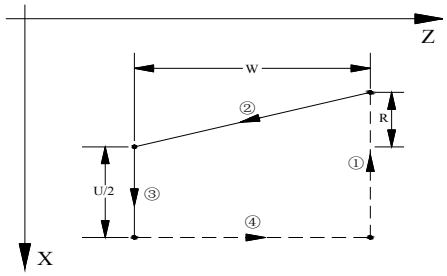
R:Different value (radius value) of X absolute coordinates between end point and start point of cutting,｜R｜≤｜U/2｜; when R＝0 or the input is default, the cylinder cutting is executed.

Cutting path: Relative position between cutting end point and starting point with U, W, R, and tool path of U, W, R with different signs are shown:

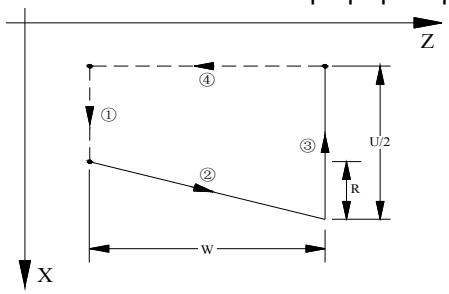
1）U>0，W<0，R>0



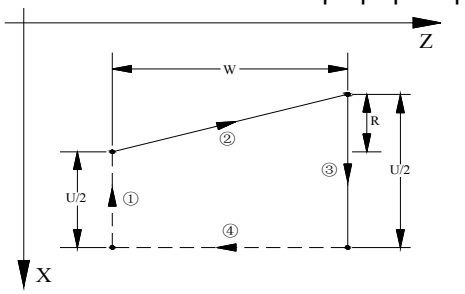
2） U<0，W<0，R<0



3）U>0，W>0，R<0 、｜R｜≤｜U/2｜



4） U<0，W>0，R>0、｜R｜≤｜U/2｜

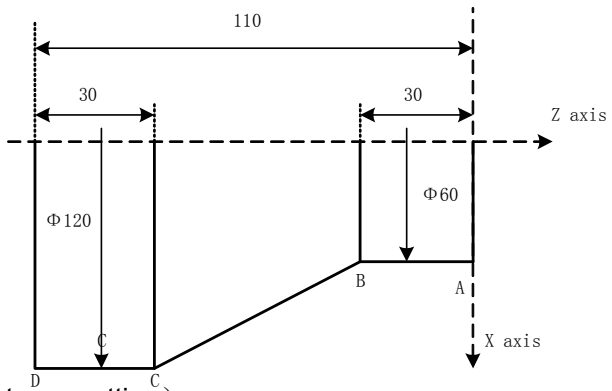


Example:Φ125×110

```
Program : O0002;
M3 S300 G0 X130 Z3;
G90 X120 Z-110 F200; (A→D,cutΦ120)
X110 Z-30;
X100;
X90;
X80;
X70;
X60;
G0 X120 Z-30;
G90 X120 Z-44 R-7.5 F150;
Z-56 R-15;
Z-68 R-22.5;
Z-80 R-30;
M30;
```

(A→B,6 times cutting cycle Φ60, increment of 10mm）

(B→C,4 times taper cutting）



2.19 Radial Cutting Cycle G94

Command format:G94 X(U)__ Z(W)__ R__ F__;

Command specification: G94 is modal;

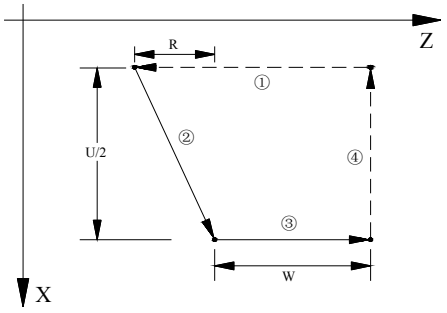
X、Z:absolute coordinate of end point of cutting;

U、W:Different value of absolute coordinate from end point to starting point of cutting;

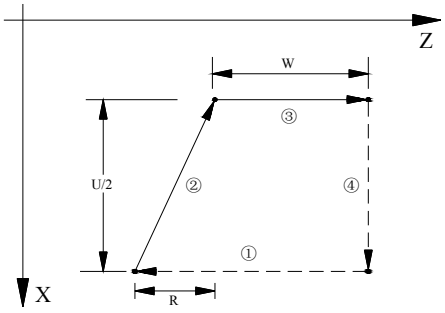
R:Different value(R value) of Z absolute coordinates from end point to starting point of cutting,|R|≤|W|.

Cutting path: Relative position between cutting end point and starting point with U, W, R.

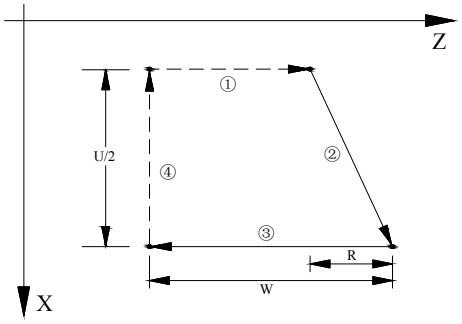
1）U>0 W<0 R<0



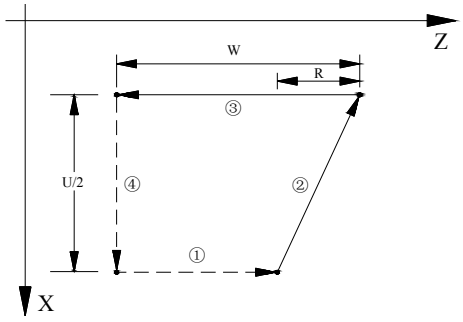
2）U<0 W<0 R<0



3）U>0 W>0 R<0 (|R|≤|W|)

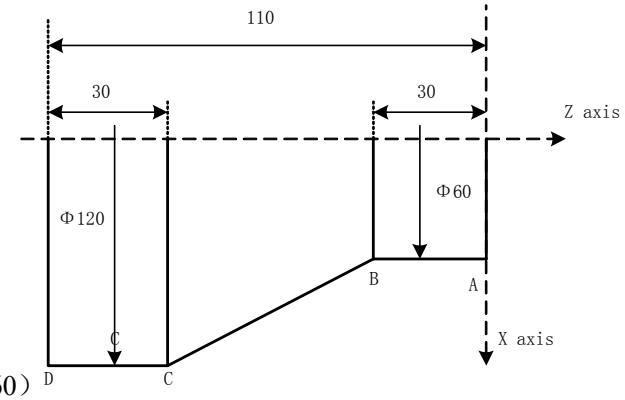


4）U<0 W>0 R<0 (|R|≤|W|)



Example:Φ125×112

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



2.20 Axial Roughing Cycle G71

G71 has two kinds of roughing cycle: type I and type II.

1.if X(U) is sole specified in N(ns) blocks,this is type I,external contour must be the monotonous increasing or the monotonous decreasing.

2.if X(U)_ ,Z(W) are specified at the same time in N(ns) blocks, this is type II,can work grooves,W0 must be specified when Z does not move, the workpiece can be up to 10 grooves.

Type I

G71 U(Δd) R(e) F__ S__ ;

G71 P(ns) Q(nf) U(Δu) W(Δw) K__J__;

N(ns) G00(G01) X(U)_;
F;
S;
N(nf)..... ;

Type II

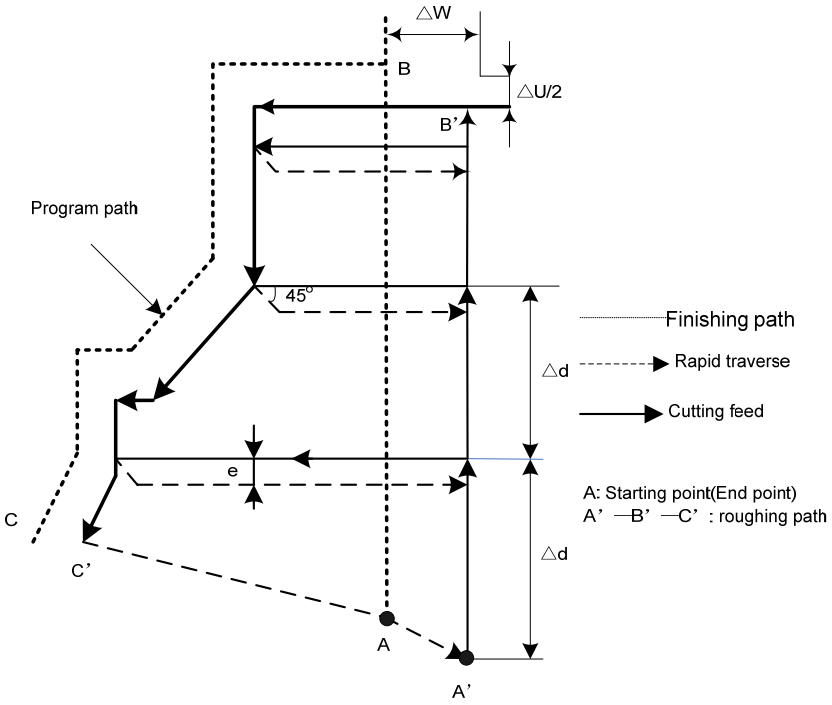
G71 U(Δd) R(e) F__ S__ ;

G71 P(ns) Q(nf) U(Δu) W(Δw) K__J__;

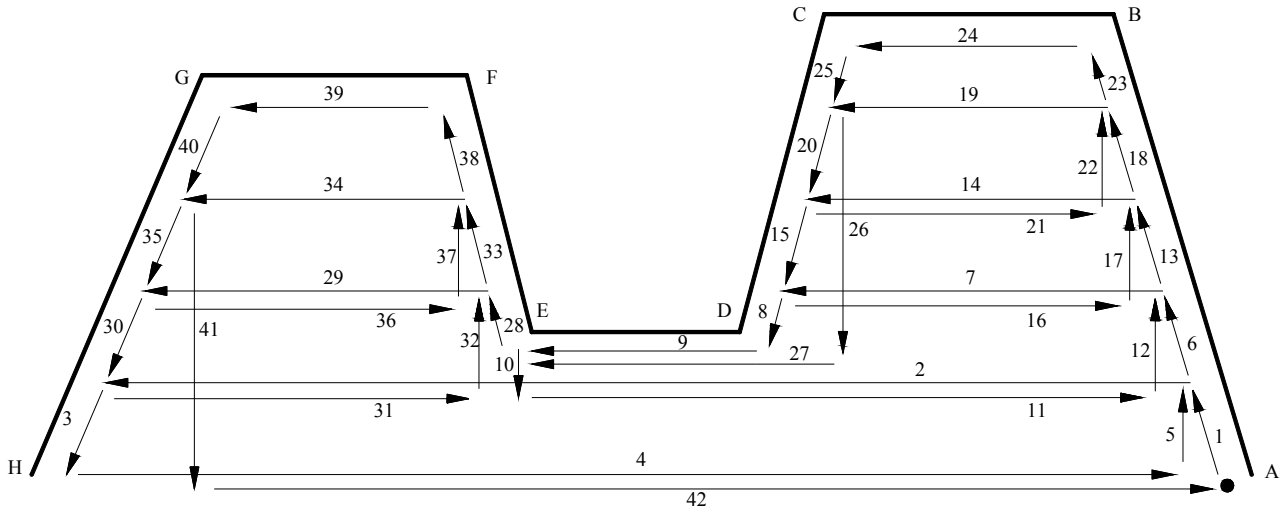
N(ns) G00(G01) X(U)_ Z(W)_
F ;
S ;
N(nf)..... ;

- U(Δd):It is each travel of X tool infeed in roughing, unit: mm/inch,radius value;
- R(e):It is travel of X tool retraction in roughing, unit: mm/inch,radius value;
- U(Δu):X finishing allowance, diameter, unit: mm/inch, with sign;
- W(Δw):Z finishing allowance, unit: mm/inch, with sign;
  For type II, only X finishing allowance can be specified; when Z finishing allowance is specified, the whole machining path offsets;
- P(ns):Block number of the first block of finishing path;
- Q(nf):Block number of the last block of finishing path;
- ns～nf block is only G00, G01;
- In ns～nf, the program block quantity cannot exceed 100;
- There are no the same block number in ns~nf;
- K:When K is not input or is not 1, the system does not check the program monotonicity; K=1, the system checks the program monotonicity;
- J:J is valid in type II,When J is not input or J is not 1, the system does not execute the run along the roughing contour; J=1: the system executes the run along the roughing contour;
- F: Feedrate; S: Spindle speed;
- M, S, F: They can be specified in the first G71 or the second ones, M, S, T, F functions of M, S, T, F blocks are invalid in G71, and they are valid in G70 finishing blocks.

Type I path:

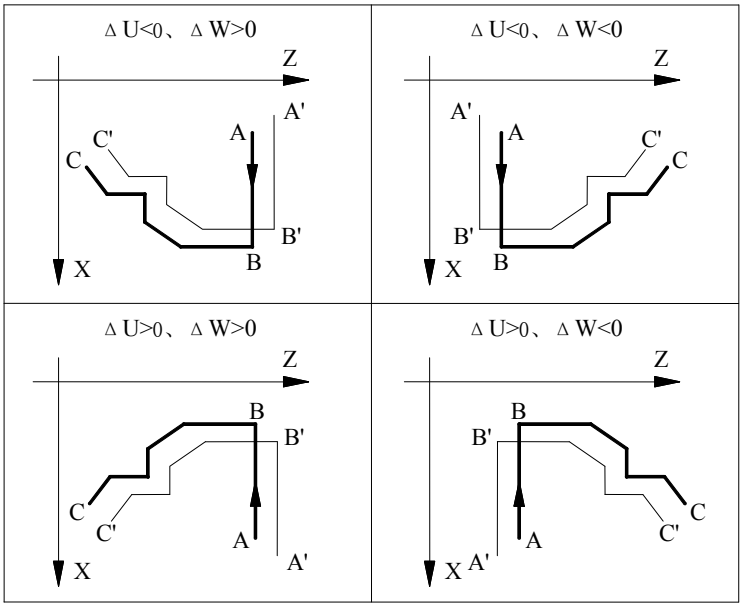


Type II path:

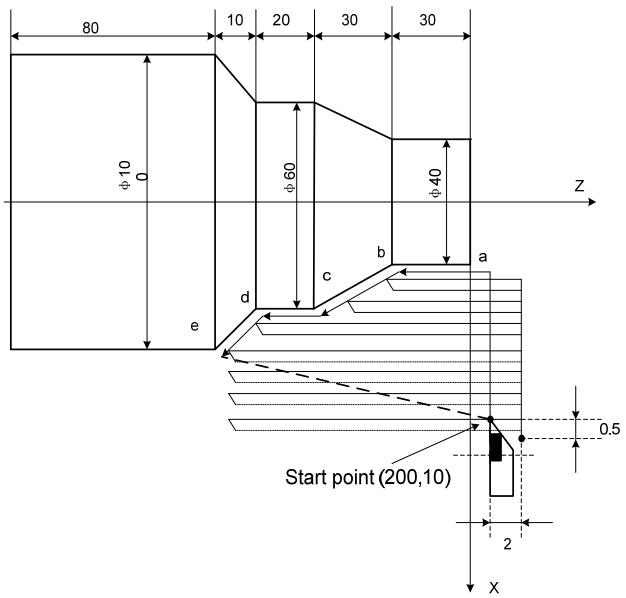


Coordinate offset direction with finishing allowance:

Δu, Δw define the coordinate offset and cut-in direction in finishing, and their sign symbol are as follows Fig. 3-64: B→C for finishing path, B'→C' for roughing path and A is the tool start-up point.



Example:(type I)



Program:O0004

G00 X200 Z10 M3 S800; (Spindle clockwise with 800 r/min)
G71 U2 R1 F200; (Cutting depth each time 4mm, tool retraction 2mm [in diameter])
G71 P80 Q120 U1 W2; (roughing a---e, machining allowance: X, 1mm;Z, 2mm)
N80 G00 X40 S1200; (Positioning)
G01 Z-30 F100 ; (a→b)
X60 W-30; (b→c)
W-20; (c→d)
N120 X100 W-10; (d→e)

a→b→c→d→e blocks for finishing path

G70 P80 Q120; (a---e blocks for finishing path)
M30; (End of block)

2.21 Radial Roughing Cycle G72

G72 has two kinds of roughing cycle: type I and type II.

1.if Z(W) is sole specified in N(ns) blocks,this is type I,external contour must be the monotonous increasing or the monotonous decreasing.

2.if X(U)_,Z(W) are specified at the same time in N(ns) blocks,this is type II,can work grooves,W0 must be specified when X does not move, the workpiece can be up to 10 grooves.

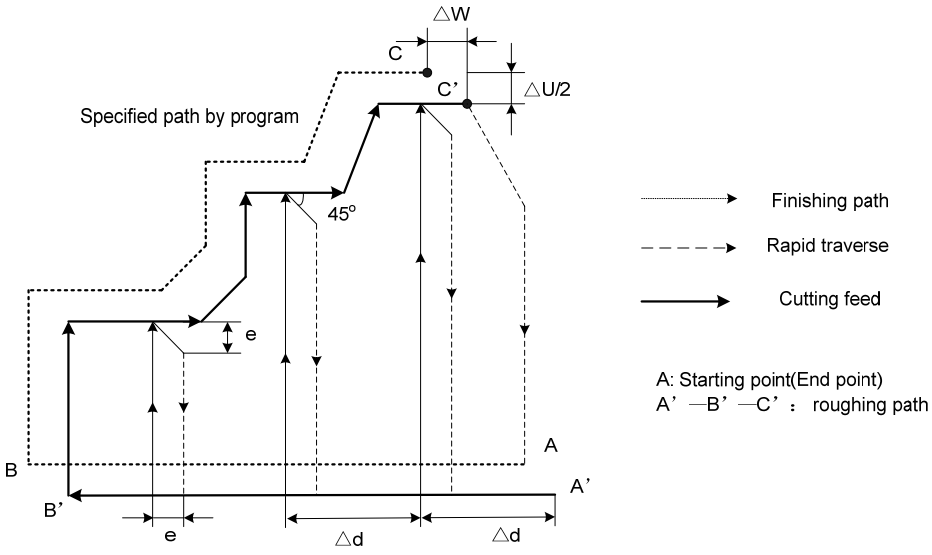
Type I

G72 W(Δd) R(e) F__ S__ ;
G72 P(ns) Q(nf) U(Δu) W(Δw) K__J__;
N(ns) G00(G01) Z(W)_;
F;
S;
N(nf)..... ;

Type II

G72 W(Δd) R(e) F__ S__ ;
G72 P(ns) Q(nf) U(Δu) W(Δw) K__J__;
N(ns) G00(G01) X(U)_ Z(W)_
F ;
S ;
N(nf)..... ;

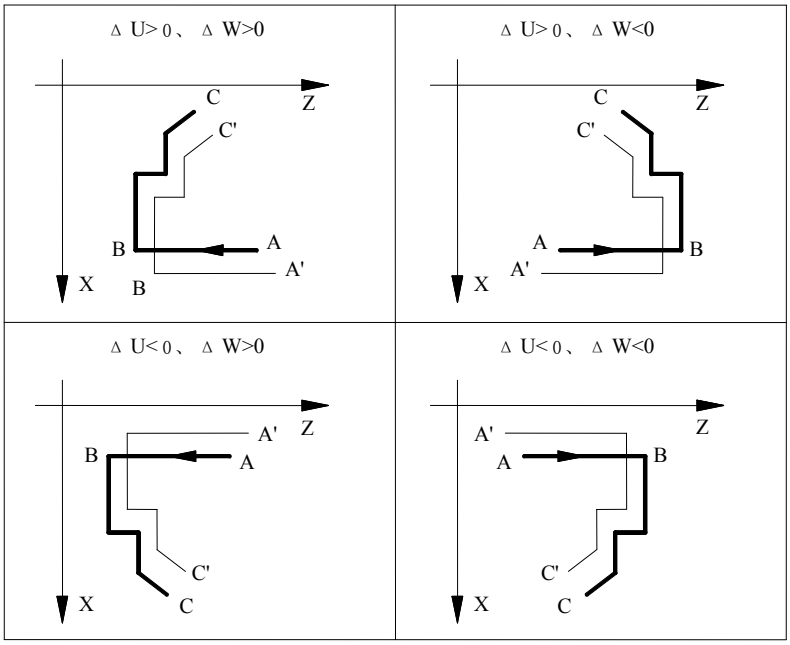
- W(Δd):it is Z cutting in roughing, without sign symbol;
- R(e):it is Z tool retraction clearance in roughing, without sign symbol;
- U(Δu)、W(Δw)、P(ns)、Q(nf)、K、J、M、S、F of relevant definitions are the same as G71's.



Type I command path

Coordinate offset direction with finishing allowance:

Δu, Δw define the coordinate offset and its direction of cut-in in finishing, and their sign symbol are as follows Fig. 3-29: B→C for finishing path, B'→C' for roughing path and A is the tool start-up point.



Program:

O0005;

G00 X176 Z10 M03 S500

G72 W2.0 R0.5 F300;

G72 P10 Q20 U0.2 W0.1;

N10 G00 Z-55 S800 ;

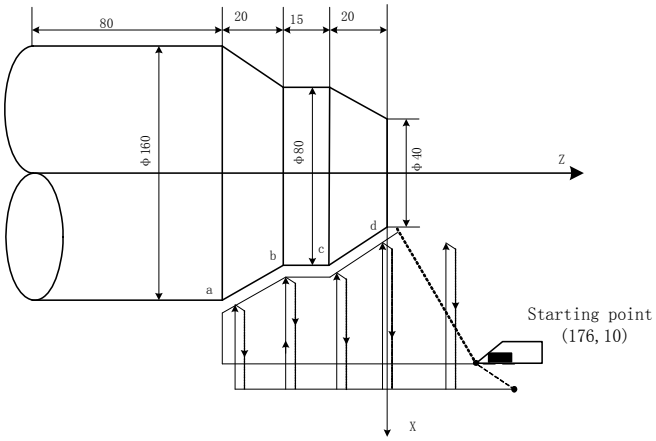
G01 X160 F120; (Infeed to a point))

X80 W20; (Machining a—b)

W15; (Machining b—c)

N20 X40 W20 ; (Machining c—d)

G70 P010 Q020 M30; (Finishing a—d)



2.22 Closed Cutting Cycle G73

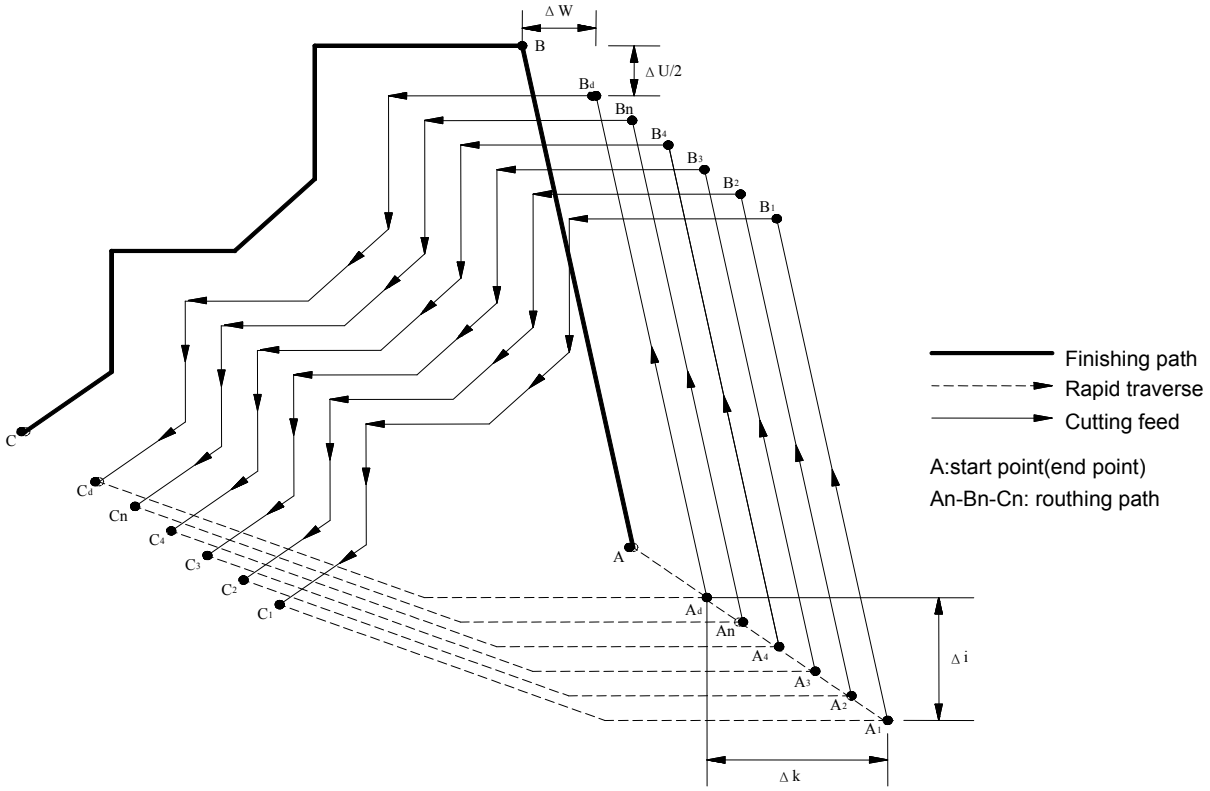
Command format:

G73 U(Δi) W (Δk) R (d) F__ S__ T__;
G73 P(ns) Q(nf) U(Δu) W(Δw);
N__(ns) . . . ;
. . . F;
. . . S;
. . . ;
N__(nf) . . . ;

Blocks for finishing path

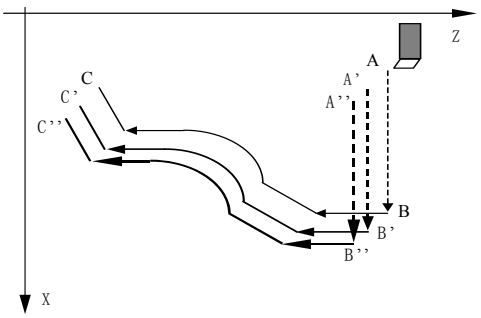
Relevant definitions:

- U(Δi):It is X tool retraction clearance in roughing, radius, with sign symbol;
- W (Δk):It is Z tool retraction clearance in roughing, with sign symbol;
- R (d):It is the cutting times;
- U(Δu)、W(Δw)、P(ns)、Q(nf)、K、J、M、S、F of relevant definitions are the same as G71's.

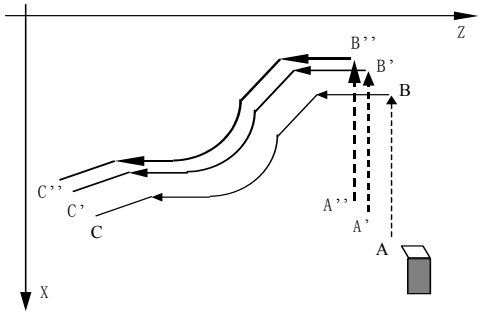


Coordinate offset direction with finishing allowance:

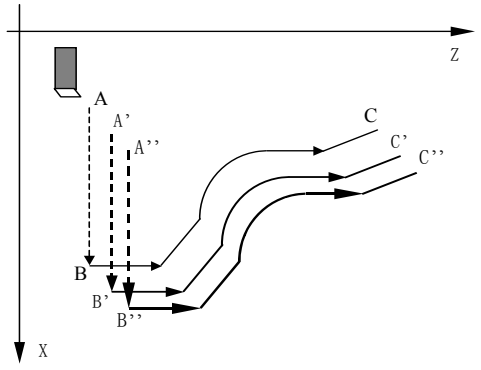
1）Δi<0 Δk>0， △u<0 Δw>0;



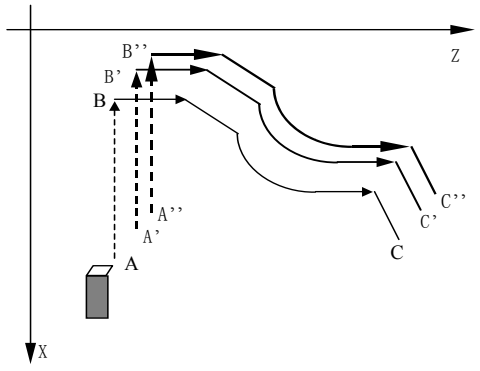
2）Δi>0 Δk>0， △u>0 Δw>0;



3）Δi<0 Δk<0， △u<0 Δw<0;

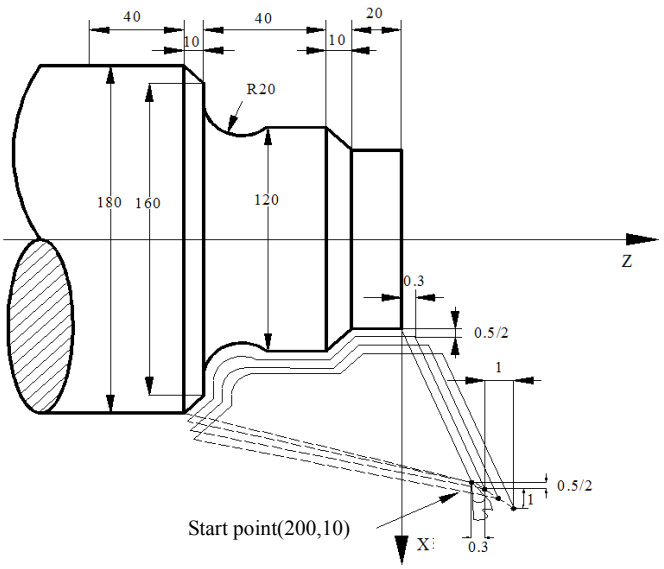


4）Δi>0 Δk<0， △u>0 Δw<0;



Program:

```
O0006;
G99 G00 X200 Z10 M03 S500;
G73 U1.0 W1.0 R3 ;
G73 P14 Q19 U0.5 W0.3 F0.3 ;
N14 G00 X80 Z0 ;
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;
```



2.23 Finishing Cycle G70

Command format:G70 P(ns) Q(nf);

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

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

After executing G71, G72 or G73 to roughing, execute G70 to finishing,there are no the same block number in ns~nf when compound cycle commands are executed repetitively in one program.

2.24 Axial Grooving Multiple Cycle G74

Command format:G74 R(e);

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

Command specification:

X:X absolute coordinate value of cutting end point $B_{f;}$

U:Different value of X absolute coordinate between cutting end point B_f and starting point;

Z: Z absolute coordinate value of cutting end point B_f (unit: mm);

W:Different value of Z absolute coordinates between cutting end point B_f and starting point;

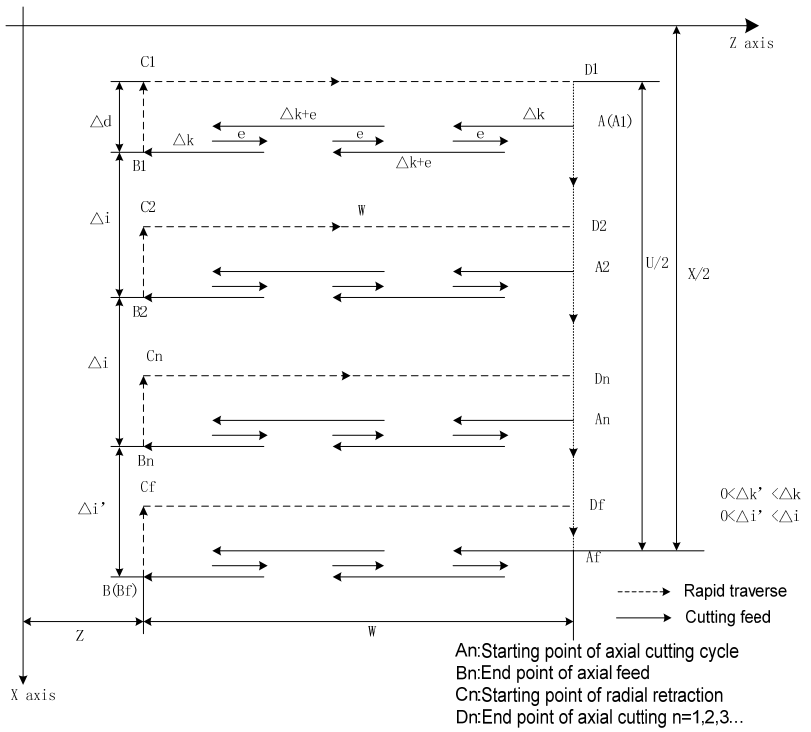
R(e):it is the tool retraction clearance after each axial(Z) tool infeed, unit:mm,radius;

P(Δi):radial(X) cutting for each axial cutting cycle, diameter value;

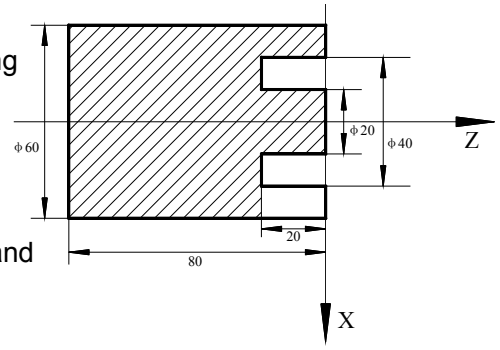
Q(Δk):radial(Z) cutting for each axial cutting cycle;

R(Δd):radial (X) tool retraction after cutting to end point of axial cutting, diameter value.

Notes:The unit of P(Δi)、Q(Δk)、R(Δd) is determined by the system control precision:IS_B(0.001 mm) or IS_C(0.0001 mm).



Example：suppose the grooving tool width is 4mm, the system least increment is 0.0001mm



G0 X36 Z5 M3 S500; (Start spindle and position to starting point of machining add tool width in X direction)

G74 R0.5 ; （Machining cycle)

G74 X20 Z-20 P30000 Q50000 F50;(Z tool infeed 5mm and tool retraction 0.5mm each time; rapid return to starting point (Z5) after cutting feed to end point (Z-20), X tool infeed 3mm and cycle the above-mentioned steps.)

M30; (End of program)

2.25 Radial Grooving Multiple Cycle G75

Command format:G75 R(e);

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

Command specification:

X:X absolute coordinate value of cutting end point B_f;

U:Different value of X absolute coordinate between cutting end point B_f and starting point;

Z:Z absolute coordinate value of cutting end point B_f;

W:Different value of Z absolute coordinate between cutting end point B_f and starting point A;

R(e):It is the tool retraction clearance after each radial(X) tool infeed, unit: mm, radius value;

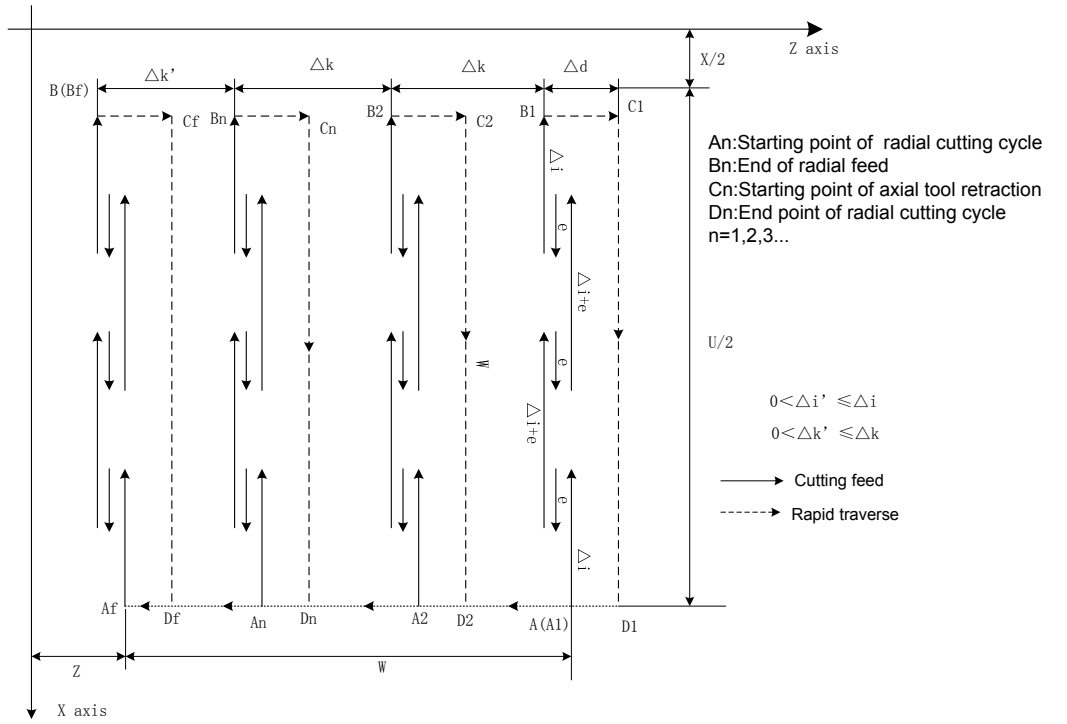
P(Δi):Radial(X) discontinuous tool infeed of each axial cutting cycle;

Q(Δk):Axial(Z) discontinuous tool infeed of each radial cutting cycle;

R(Δd):Axial (Z) tool retraction clearance after cutting to end point of radial cutting.

The unit of P(Δi)、Q(Δk)、R(Δd) is determined by the system control precision:IS_B(0.001 mm) or IS_C(0.0001 mm).

Command path:



Example：suppose the grooving tool width is 4mm, the system least increment is 0.0001mm.

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

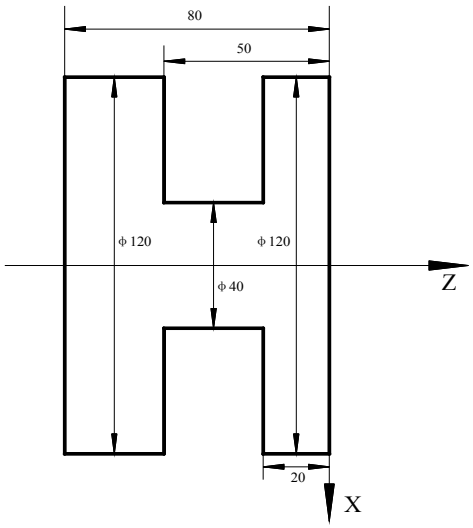
G0 X125 Z-24; (Position to starting point of machining, add tool width in Z direction)

G75 R0.5 F150; (Machining cycle)

G75 X40 Z-50 P60000 Q30000; (X tool infeed 6mm every time, tool retraction 0.5mm, rapid returning to starting 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 starting point of machining)

M30; (End of program)



2.26 Drilling Cycle G83 or G87

Command format:

axle drilling G83 X(U)__ C(H)__ Z(W)__ P__ Q__ R__ F__ K__M__ J0/1;

radial drilling G87 Z(W)__ C(H)__ X(U)__ P__ Q__ R__ F__ K__ M__ J0/1;

cancel drilling cycle:G80

Code definition：

G83, G87 is modle G code, It is necessary to select the G18 panel when performing the G83 or G87 command;

G83: Axial drilling cycle G code, axial (Z axis) drilling;

G87: Radial drilling cycle G code, axial (X axis) drilling;

X(U)、C(H):Drilling position;------G83

Z(W):at the bottom of a hole;------G83

Z(W)、C(H):Drilling position;------G87

X(U):at the bottom of a hole;------G87

P:It is pause time at the bottom, modal data, Unit: ms;

Q:In-feed value each time, radius value,When the Q value is 0, directly feed to the botton of the hole. Unit:least input increment;

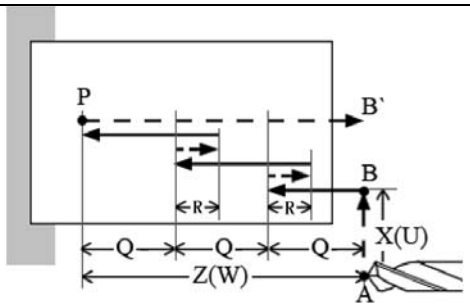
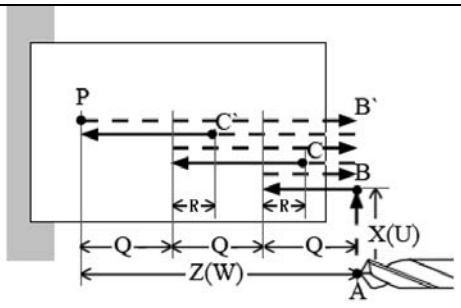
R:In-feed value after the in-feed for each time,unit:mm/inch,radius value;

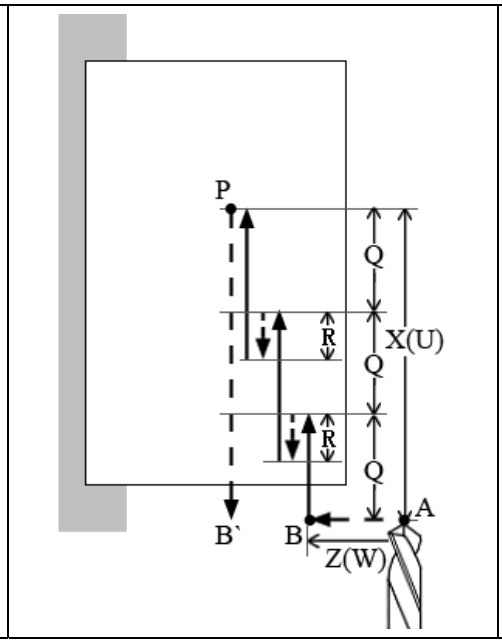
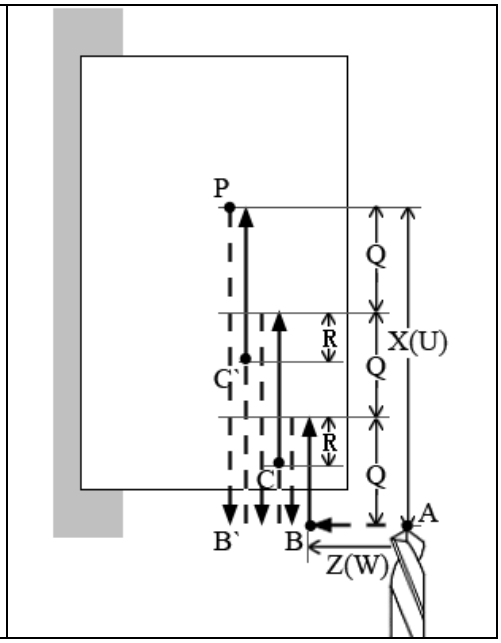
K:Drilling repeated times can be omitted;it is regarded as 1 by default when ignores;

M:The M code is used for clamping the indexing spindle;

J:0:High speed peck drilling cycle;1:Peck drilling cycle, modal data. Its initial value regardes as 0 by default;

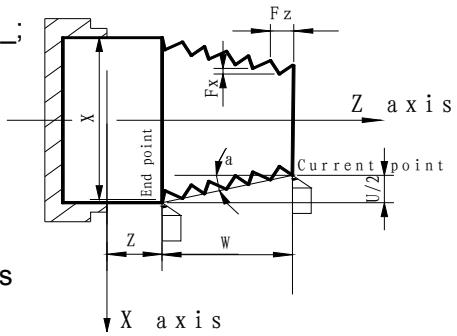
F:Cutting feddrate.

	Common drilling	Peck drilling
G83		

G87		

2.27 Thread Cutting with Constant Lead G32

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



Command specification: G32 is modal;

X(U)、Z(W):end point coordinates of thread interpolation;

F:Thread pitch ,F is modal parameter;

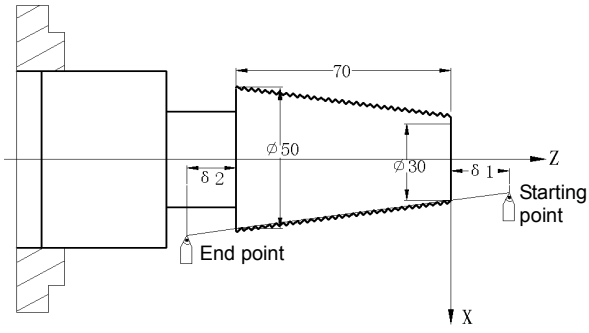
I:Teeth per inch. It is ones per inch;

J:Movement in the short axis in thread run-out,J value is the modal parameter. with negative sign;

K:Length in the long axis in thread run-out, K is modal parameter, without direction;

Q:Initial angle, unit: 0.001 degree, Its initial angle is 0° if Q is not specified.

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




```
O0009;
G00 X29.1429 Z3;      (First cut-in 1mm)
G32 X50.5714 W-75 F2.0;(First taper cutting)
G00 X55;              (Tool retraction)
W75;            (Z returns to the starting point)
X28.6529;       (Second tool infeed 0.5mm)
G32 X50.0714 W-75 F2.0;(Second taper
                       thread cutting)
G00 X55;              (Tool retraction)
W75 ;                 (Z returns to the starting point)
M30;
```

2.28 Rigid Thread Cutting G32.1

Command format:G32.1 X(U)__ Z(W)__ C(H)__F(I)__ J__ K__ S__;

Command format:When using G32.1 code,The spindle must is servo spindle,and work mode must is the position controllable method,the running path is the same as that of G32.

X(U)、Z(W):End point coordinates of thread interpolation;

C(H):Start angle of thread interpolation;

F(I):Thread lead, F(I)> 0 right-hand thread, F(I)< 0 left-hand thread;

J:Movement in the short axis in thread run-out, with negative sign,unit:mm;

K:Length in the long axis in thread run-out, without direction,unit:mm;

S:spindle speed.

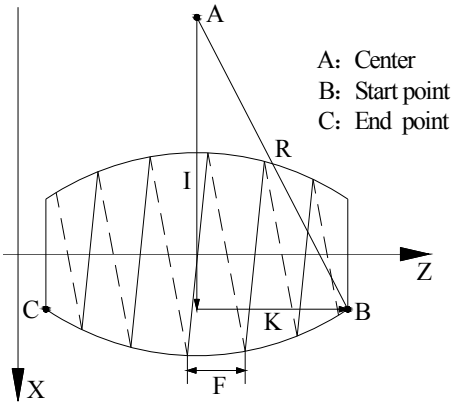
Example**:**The thread is the right-hand, its lead is 2mm, the spindle speed is 500 r/m in thread cutting, thread cutting length is 20mm.

G00 X100 Z100 T0101;	change the tool
G00 X25 Z2;	position to the thread starting point
M14;	switch the spindle from speed control mode to position control mode
G50 C0;	set the zero of the rotary axis
G32.1 Z-20 F2 S500 M08;	thread turning
G00 X30;	tool retraction
Z2 C0;	return to tool change position
X24;	execution the 2nd machining procedure
G32.1 Z-20 F2 S500;	thread turning
G00 X100;	tool retraction
M15;	switch the spindle from position control mode to speed control mode
M30;	end of program

2.29 Arc Threading Cutting Code G32.2 or G32.3

Command format:

G32.2 / G32.3 } X(U)__ Z(W)__ R__ (I__ K__) F__ Q__



Command specification: G32.2/G32.3 is modal.

G32.2:CW arc thread machining;

G32.3:CCW arc thread machining;

X(U)、Z(W):end point coordinates of thread interpolation;

R:arc radiusL;

I、K:the vector from starting point to center point of arc;

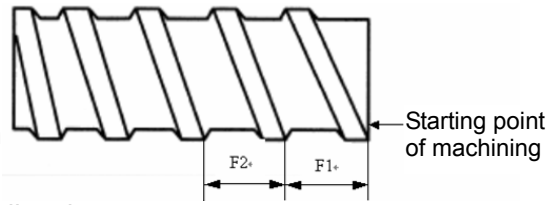
F(I):Thread pitch;

Q :Initial angle, unit: 0.001 degree;

2.30 Thread Cutting with Variable Lead G34

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

Command specification: G34 is modal;



X(U)、Z(W):End point coordinates of thread interpolation;

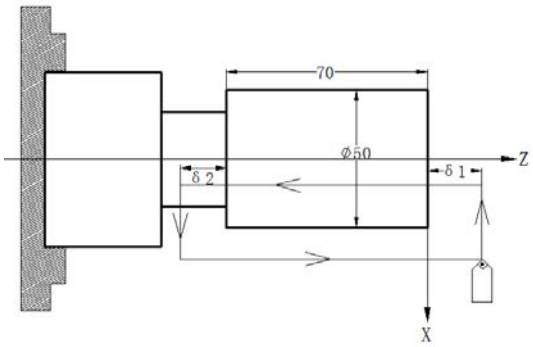
J:Movement in the short axis in thread run-out,with negative sign;

K:Length in the long axis in thread run-out,without direction;

F(I):first thread pitch;

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

Example:First pitch of starting point**:** 4mm, increment 0.2mm per rev of spindle. Use macro variables to simplify programming when G34 is used many times. δ1 = 4mm,δ2 = 4mm, total cutting depth 4mm, total cutting cycle 15 times; first tool infeed 0.8mm, gradual decreasing cutting every time 0.2mm, min. infeed 0.2mm.




```
O0010;
G00 X60 Z4 M03 S500;
G65 H01 P#102 Q0.8;          First tool infeed: assignment #102=0.8mm
G65 H01 P#103 Q0;            Cycle count: assignment #103=0
N10 G65 H02 P#104 Q#103      Cycle count starting:#104=#103+1
R1;                                  #103=#104
G65 H01 P#103 Q#104;         Total cutting cycle times:#104=15，jump to block N30
G65 H81 P30 Q#104 R15;       Tool infeed to Φ50
G00 U-10;                    Cutting infeed: #100=#102
G65 H01 P#100 Q#102;         Tool infeed
G00 U-#100;                  Variable pitch cutting
G34 W-78 F3.8 J5 K2 R0.2;    Tool retraction
G00 U10;                     Z returns to starting point
Z4;
G00 U-10;                    Decreasing of cutting feed again:#101=#100—0.2
G65 H03 P#101 Q#100 R0.2;    Assignment again #102=#101
G65 H01 P#102 Q#101;         Infeed: Jump to block N20 when #102≦0.2mm
G65 H86 P20 Q#102 R0.2;      Unconditionally jump to block N10
G65 H80 P10;                 Min. infeed: #102=0.2
N20 G65 H01 P#102 Q0.2;      Unconditionally jump to block N10
G65 H80 P10;
N30 M30;
```

2.31 Z Thread Cutting G33

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

Command specification: G33 is modal command;

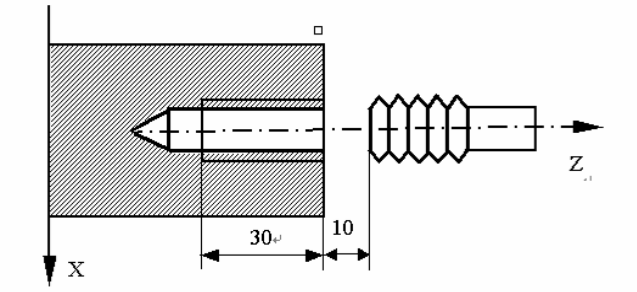
F:Thread pitch;

I:Specify thread teeth per inch;

L:Multi threads: The system defaults it is single thread when L is omitted.

Example:thread M10×1.5

```
O0011;
G00 Z90 X0 M03;        Start spindle
G33 Z50 F1.5;          Tap cycle
M03;                   Start spindle again
G00 X60 Z100;          Machine continuously
M30
```



2.32 Rigid Tapping G84, G88

Command format:G84 X(U)__ C(H)__ Z(W)__ P__ F(I)__ K__M__;（End face rigid tapping）

G88 Z(W)__ C(H)__ X(U)__ P__ F(I)__ K__ M__;（Side rigid tapping G88）

G80;（cancel rigid tapping）

Command specification:G84、G88 are modal G codes,The spindle must is servo spindle, and work mode must is the position controllable method.

X(U)、C(H):Tapping hole position---G84;

Z(W):Hole bottom position of tapping;------G84

Z(W)、C(H):Tapping hole position;------G88

X(U):Hole bottom position of tapping hole;------G88

P:Pause time (ms) when tapping to the hole bottom

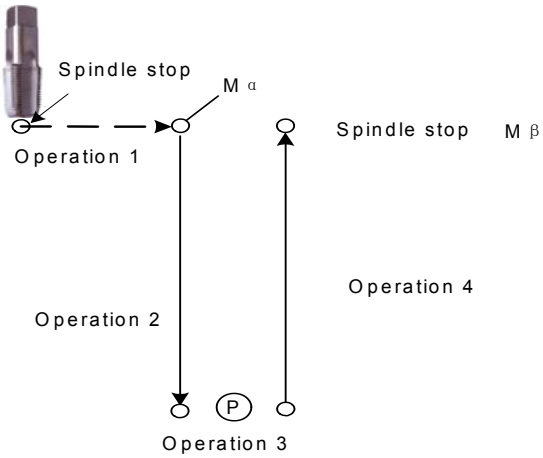
F(I):Thread lead

F(I)> 0 right-hand tapping

F(I)< 0 left-hand tapping

K:Repetitive count of tapping

M;Used to clamp the graduation spindle.



Rigid tapping method:

M29 S_ before G84/G88 is specified below

M29 S_;

G84(G88) X_C_(Z_C_) Z_(X_) P_ F_ K_ M_;

2.33 Thread Cutting Cycle G92

Command format:G92 X(U)_ Z(W)_ R_ F(I)_ J_ K_ L_;

Command specification: G92 is modal;

X:X absolute coordinate of end point of cutting;

U:different value of X absolute coordinate from end point to starting point of cutting;

Z:Z absolute coordinate of end point of cutting;

W:Different value of Z absolute coordinate from end point to starting point of cutting;

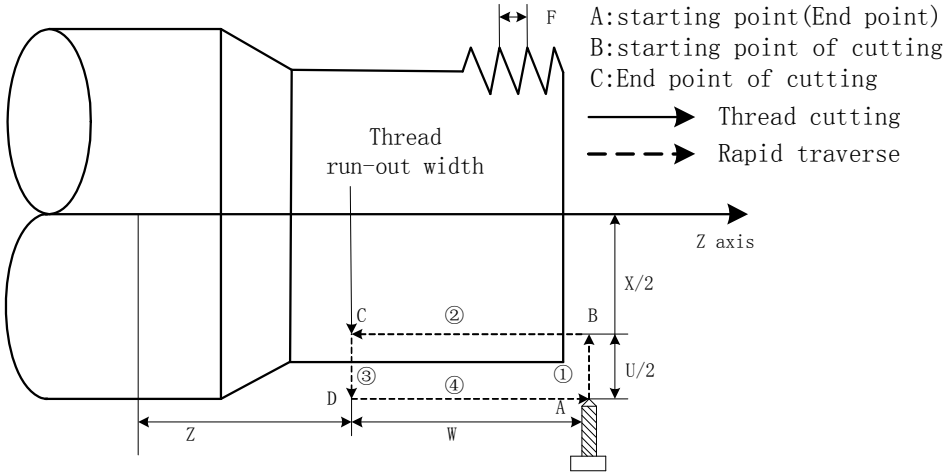
R:Different value(radius value) of X absolute coordinate from end point to starting point of cutting;

F(I):Thread lead;

J:Movement in the short axis in thread run-out, unit: mm/inch, without direction.

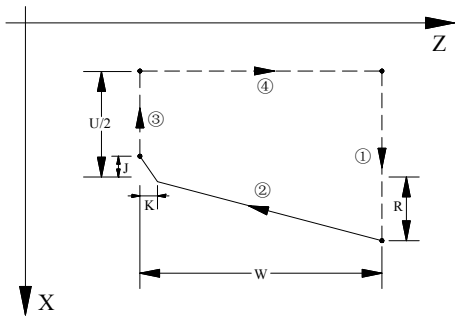
K：Movement in the long axis in thread run-out, unit: mm/inch, without direction. when J and K are omitted, Length of thread run-out is specified by №019;

L:Multi threads, The system defaults it is single thread when L is omitted.

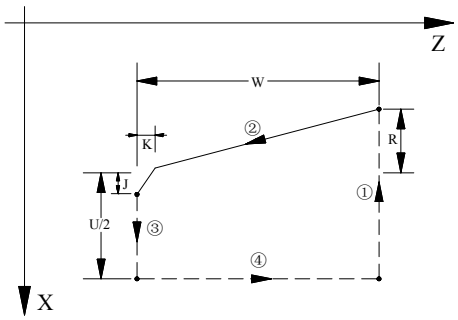


Relative position between thread cutting end point and starting point with U, W, R and tool path and thread run-out direction with different U, W, R signs below:

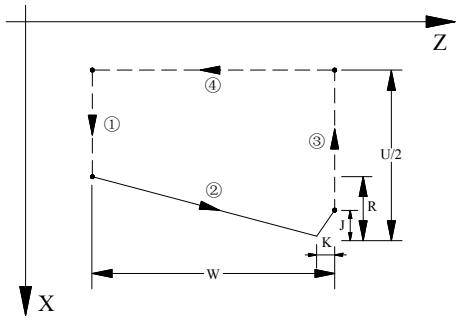
1） U>0，W<0，R>0



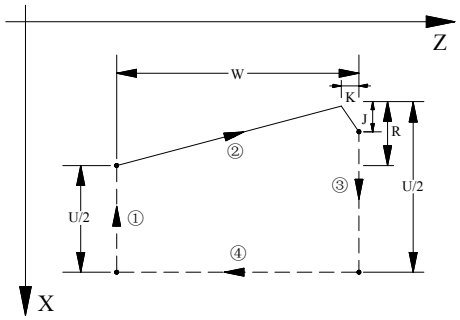
2） U<0，W<0，R<0



3） U>0，W>0，R<0 、|R|≤|U/2|

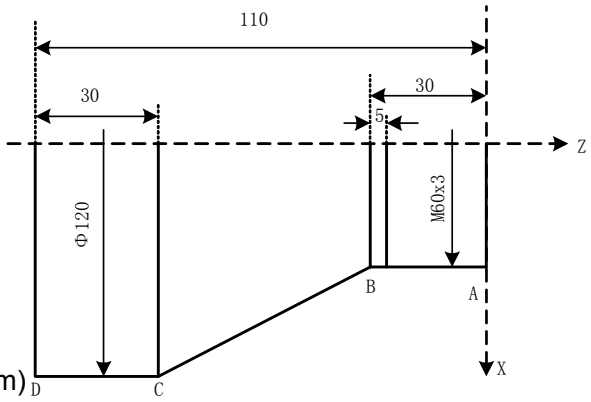


4） U<0，W>0，R>0、|R|≤|U/2|



Example：

```
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 ;                     (The second tool
        infeed 1mm)
X57;                        (The third tool infeed 0.7mm)
X56.9;                      (The fourth tool infeed 0.1mm)
M30;
```



2.34 Rigid Thread Cutting Cycle G92.1

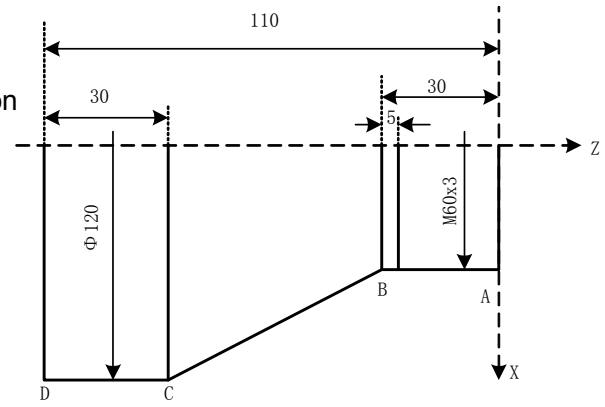
Command format:G92.1 X(U)_ Z(W)_ F(I)_ L_ S_ R_;

Command specification: G92.1 is modal; The spindle must is servo spindle, and work mode must is the position controllable method,the path is the same as that of G92.

- X、U、Z、W、R、F、L of relevant definitions are the same as that of G92.
- The sign and tool path of U、W、R are the same as that of G92.
- S:The speed of thread cutting.

Example：

```
O0012;
G0 X150 Z50 T0101;
M14;     (Shift the spindle to the position
         controllable method)
G50 C0; (The C coordinate will set 0)
G0 X65 Z5;
G92.1 X58.7 Z-28 F3 S600;
X57.7;
X57;
X56.9;
M15;
M30;
```



2.35 Multiple Thread Cutting Cycle G76

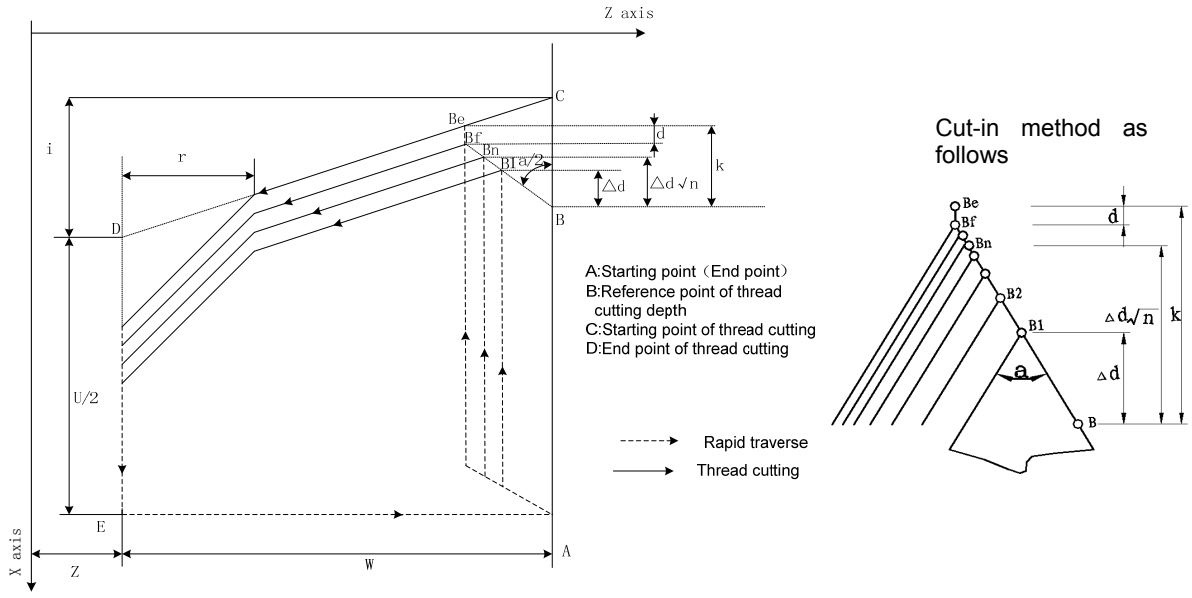
Command format:G76 P(m)(r)(a) Q(△dmin) R(d) J_ K_;

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

Command specification:

- X:X absolute coordinate (unit: mm) of thread end point;
- U:Different value (unit: mm) of X absolute coordinate between thread end point and starting point;
- Z:Z absolute coordinate (unit: mm) of thread end point;
- W:Different value (unit: mm) of Z absolute coordinate between thread end point and starting point;
- P(m):Times of thread finishing: 00～99 (unit: times);
- P(r):Width of thread run-out 00～99(unit: 0.1×L,L is the thread pitch);

- P(a):Angles at taper of neighboring two tooth, unit:deg(°);
- Q(△dmin)：Minimum cutting travel of thread roughing, unit: least input increment, radius value;
- R(d):It is the cutting amount in thread finishing, radius value without sign symbols;
- R(i):It is thread taper and is the different value of X absolute coordinate between thread starting point and end point, unit:： mm/inch, radius value;
- P(k):Depth of thread tooth, the total cutting depth of thread, unit: least input increment, radius value, without sign symbols;
- Q(△d):Depth of the 1st thread cutting, unit: Least input increment, radius value, without sign symbols;
- F:metric thread pitch, I:thread teeth per inch for inch thread;
- J:Movement amount in the short axis when thread run-out, unit: mm/inch, without direction;
- K:Movement amount in the long axis when thread run-out, unit: mm/inch, without direction;
- J, K are not compiled in G76, the thread run-out is executed according to No.19.



Example: threadM68×6,system least input increment is 0.0001mm

```
O0013;
G0 X100 Z50 M3 S300;
G04 X2;                    (dwell 2 seconds,Spindle speed wait stable)
G00 X80 Z10;          (Rapid traverse to starting point of machining)
G76 P020560 Q1500 R0.1; (Finishing 2 times, chamfering width 0.5mm, tool angle 60°, min.
                              cutting depth 0.15, finishing allowance 0.1)
G76 X60.64 Z-62 P36800 Q18000 F3;(Tooth height 3.68, the first cutting depth 1.8)
G00 X100 Z50 ;                (Return to starting point of program)
M30;                              (End of program)
```

2.36 Enhancement Thread Cutting Cycle G78

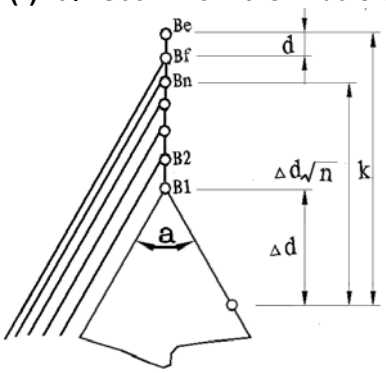
Command format:G78 P(m)(r)(a) Q(△dmin) R(d) J_ K_ D_ L_ E_;
G78 X(U)__ Z(W)__ R(i) P(k) Q(△d) F(I)__;

Command specification:

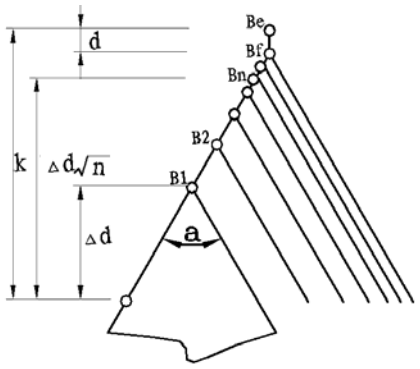
- **The relevant definitions** of X、U、Z、W、Q(△dmin)、R(d)、R(i)、P(k)、F、I、J、K are the same as that of G76.
- P(m):Cutting depth selection 0: Equidistance in-feed; 1: Diminishing in-feed;
- P(r):Cutting method 0: Blade cuts in along with the netrual line of the thread theeth;
  1: Blade cuts in along with the left of the thread theeth;
  2: Blade cuts in along with the right of the thread theeth;
  3: Blade cuts in along thread teeth with left/right in turn;
- P(a):Angles at taper of neighboring two tooth, unit:deg (°);
- D:It is the cutting amount;
- L:Head quantity of multi thread, The system defaults it is single thread when L is omitted;
- E:The rotation distance value along with unit:mm,radius value、without sign symbol.

Cut-in method:

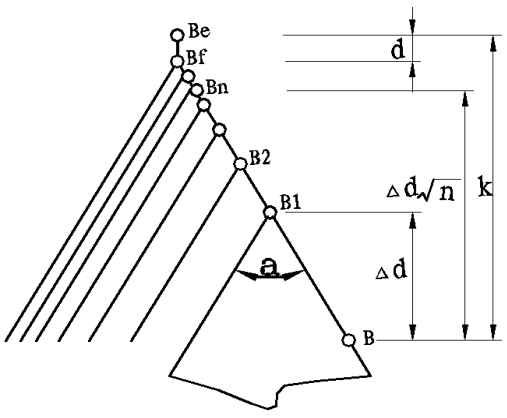
P(r)=0：Cut in from the middle of the teeth



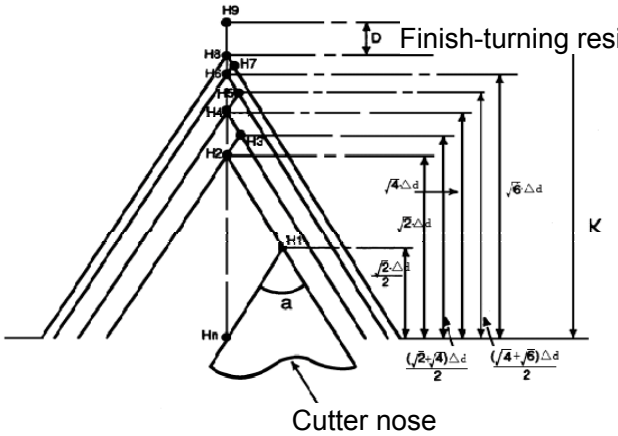
P(r)=1：Left cut-in method



P(r)=2：Right cut-in method



P(r)=3：Shifting cut-in method between left and right



2.37 Dwell G04

Command format:G04 P_ Q_ ;or
G04 X_ Q_ ;or
G04 U_ Q_ ;or
G04;

Command specification: G04 is non-modal.The system exactly stop a block when P, X, U, Q are not input.

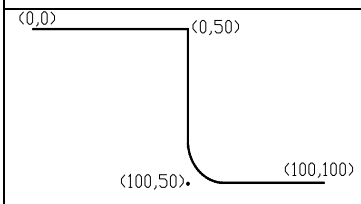
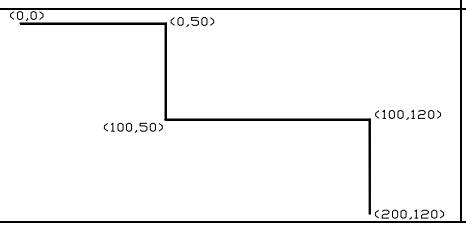
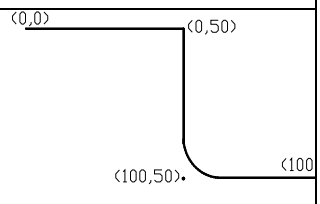
P :unit: ms;

X、U unit: s;

Q Skip signal address selection, solution range 1～4,The skip signal address from the corresponding P1～P4 are set by parameters №148～№151.

2.38 Exact Stop Function Between Blocks G61

Command format:G09;
G61;
G64;

	G09	G61	G64
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., Non-modal G commands	The blocks included the overall motions between the current block and the G64 are exactly performed, and then execute the next block, modal command.	Cancel the G61 modal, the next block will not perform the exact stop
Example	N1 G00 X0 Z0 N2 G09 G01 Z50 N3 X100 N4 Z100 N5 M30	N1 G00 X0 Z0 N2 G61 N3 G01 Z50 N4 X100 N5 Z120 N6 X200 N7 G64 N8 M30	N1 G00 X0 Z0 N2 G61 N3 G01 Z50 N4 G64 N5 X100 N6 Z100 N7 M30
			
Remax	G09, G61/G64 are disabled in the G71, G72 and G73 cycle procedure;		

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

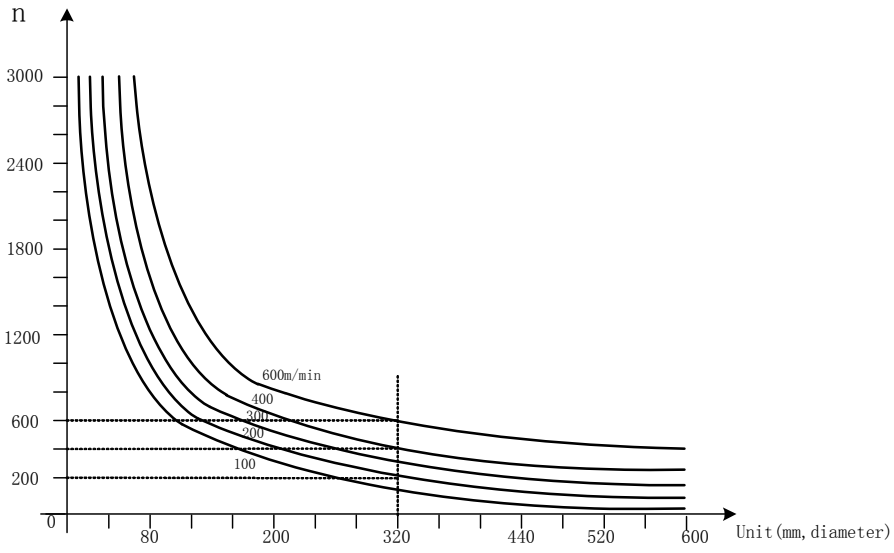
Command format:G96 S__;The constant surface speed control is valid

G97 S__;The constant surface speed control is cancelled

G50 S__;define max. spindle speed limit in the constant surface speed control (r/min)

When the spindle speed controlled by the analog voltage is valid,in G99 feedrate per rev,the constant surface control is valid. the system defaults G97 is valid when the system is switched on.

The spindle speed is changed along with the absolute value of X absolute coordinates of programming path in the constant speed control. If the absolute value of X absolute coordinates adds, the spindle speed reduces, and vice verse, which make the cutting surface speed as S command value.



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

2.40 Feedrate per Minute G98, Feedrate per Rev G99

Command format:G98 F__;

G99 F__;

G98 is feedrate per min，the modal G command, cutting feed rate is specified as mm/min, the system defaults G98 is valid when the system turns on.

G99 is feedrate per rev,the modal G command, cutting feed rate is specified as mm/r; The actual cutting feedrate is gotten by multiplying the F command value (mm/r) to the current spindle speed(r/min). If the spindle speed varies, the actual feedrate changes too. In G99 state, a spindle encoder should be fixed on the machine tool to machine the workpiece.

Reduction formula of feed between per rev and per min:$F_m = F_r \times S$

F_m：feed per min(mm/min)

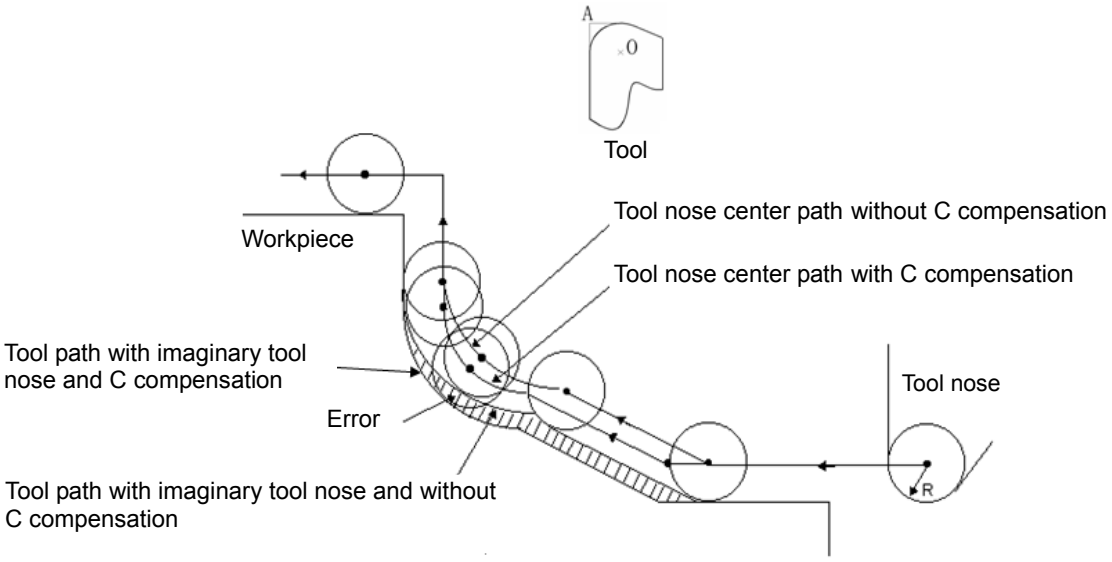
F_r：feed per rev(mm/r)

S：spindle speed (r/min)

2.41 TOOL NOSE RADIUS COMPENSATION (G41, G42)

2.41.1 Application

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. 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.



2.41.2 Imaginary Tool Nose Direction

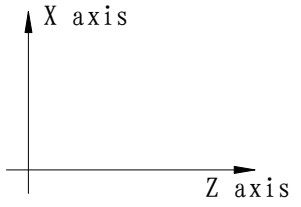
Suppose that it is generally difficult to set the tool nose radius center on the initial position,as follow left;suppose that it is easily set the tool nose on it,as follow right:

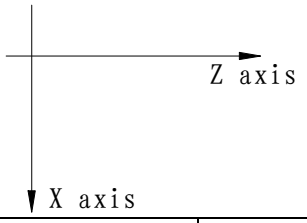
Programming with tool nose　　Programming with imaginary tool nose

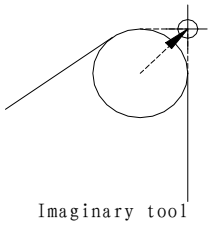
The tool is supposed to one point in programming but the actual cutting blade is not one ideal point owing to machining technology. Because the cutting blade is not one point but one circular,

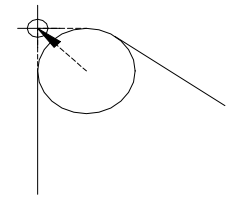
machining error is caused which can be deleted by tool nose circular radius compensation. In actual machining, suppose that there are different position relationship between tool nose point and tool nose circular center point, and so it must create correct its direction of imaginary tool nose.

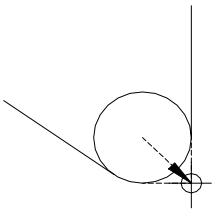
From tool nose center to imaginary tool nose, set imaginary tool nose numbers according to tool direction in cutting. Suppose there are 10 kinds of tool nose setting and 9 directions for position relationship. The tool nose directions are different in different coordinate system (rear tool post coordinate system and front tool post coordinate system) even if they are the same tool nose direction numbers as the following figures. In figures, and end point of arrowhead is the imaginary tool nose.

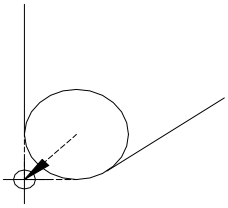


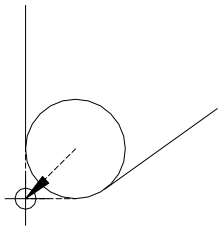


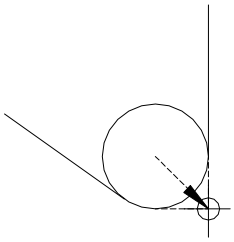

Imaginary tool nose No.1

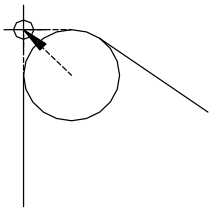

Imaginary tool nose No. 2

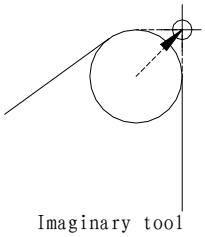

Imaginary tool nose No. 1

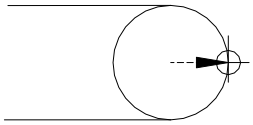

Imaginary tool nose No.2

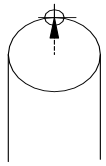

Imaginary tool nose No.3

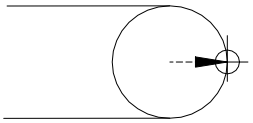

Imaginary tool nose No.4

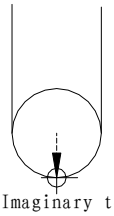

Imaginary tool nose No.3

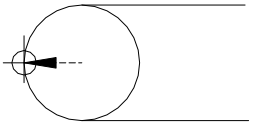

Imaginary tool nose No.4

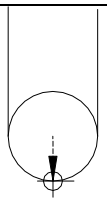

Imaginary tool nose No.5

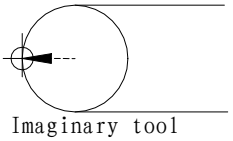

Imaginary tool nose No.6

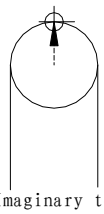

Imaginary tool nose No.5


Imaginary tool nose No.6


Imaginary tool nose No.7


Imaginary tool nose No. 8


Imaginary tool nose No.7


Imaginary tool nose No.8

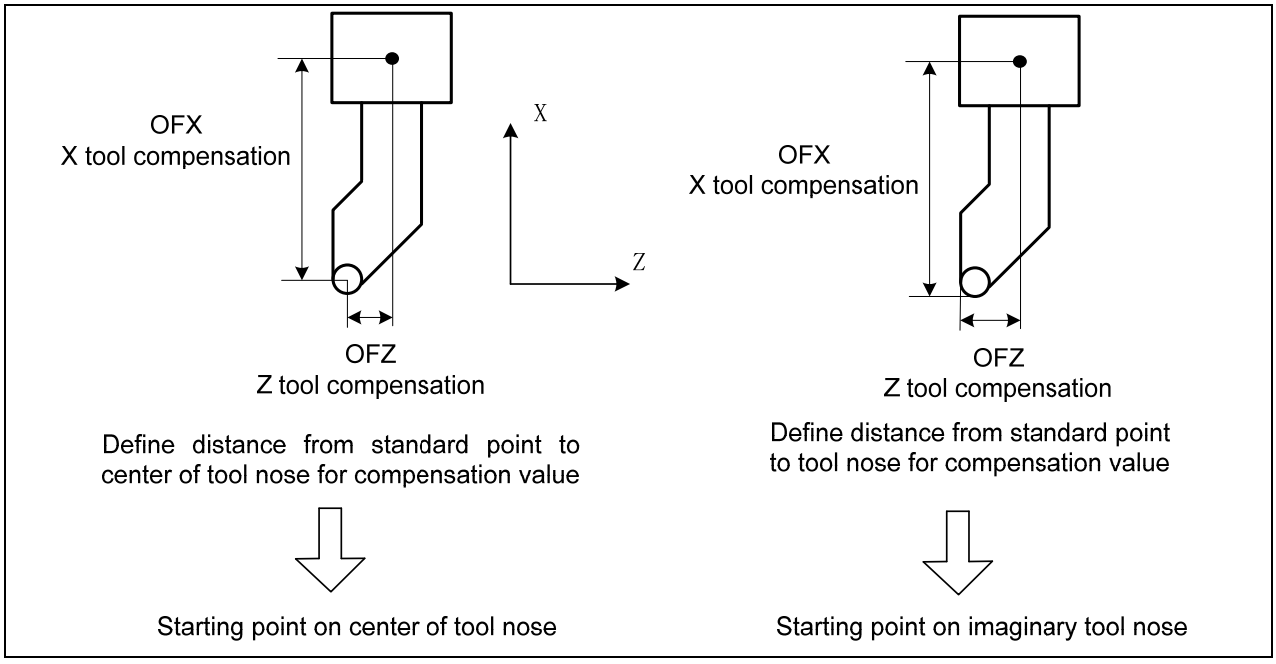
Tool nose center on starting point,Imaginary tool nose NO.0

2.41.3 Compensation Value Setting

Preset imaginary tool nose number and tool nose radius value for each tool before executing tool nose radius compensation. Set the tool nose radius compensation value in ***OFFSET*** window , R is tool nose radius compensation value and T is imaginary tool nose number.

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

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).



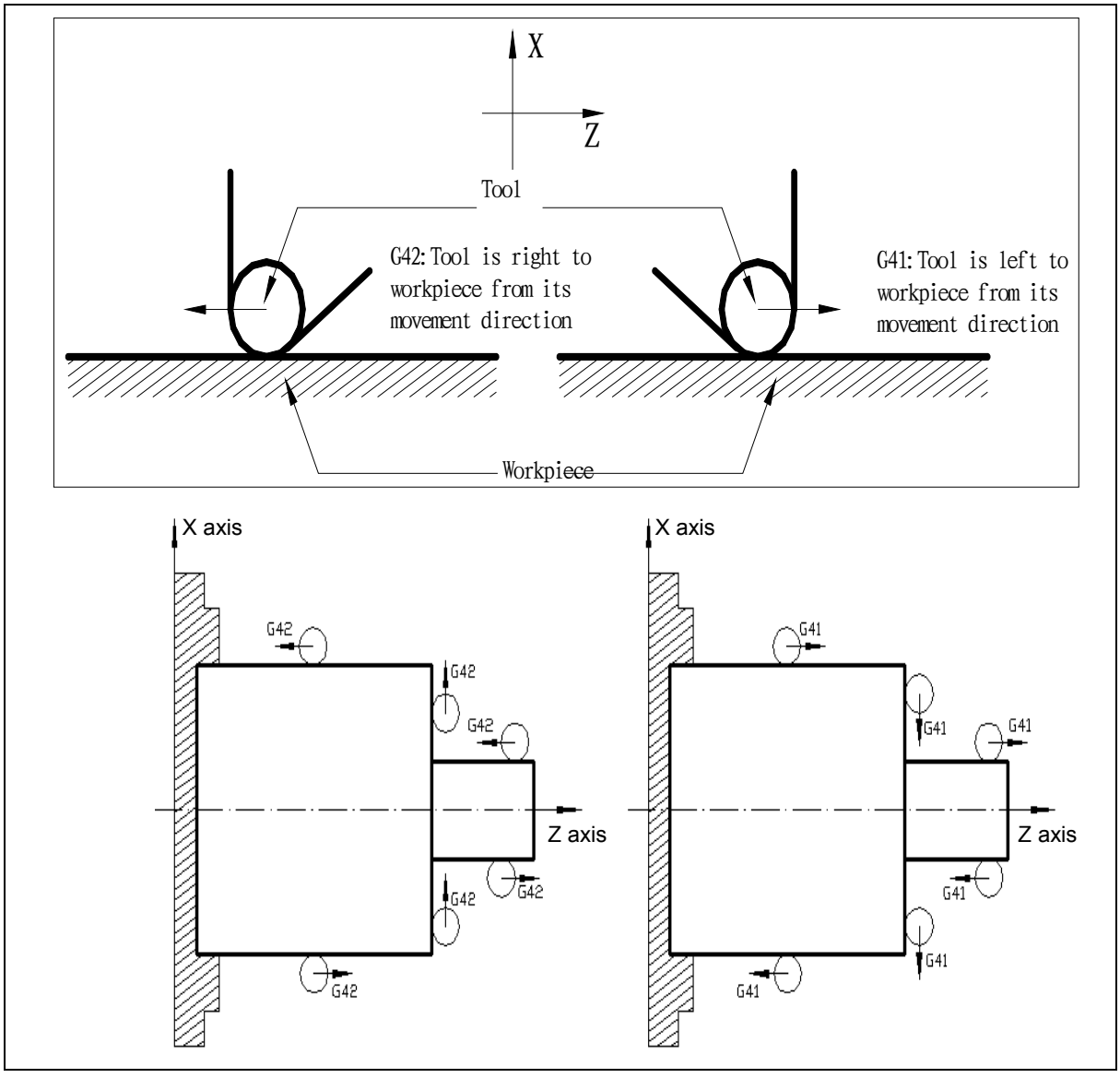
2.41.4 Command Format

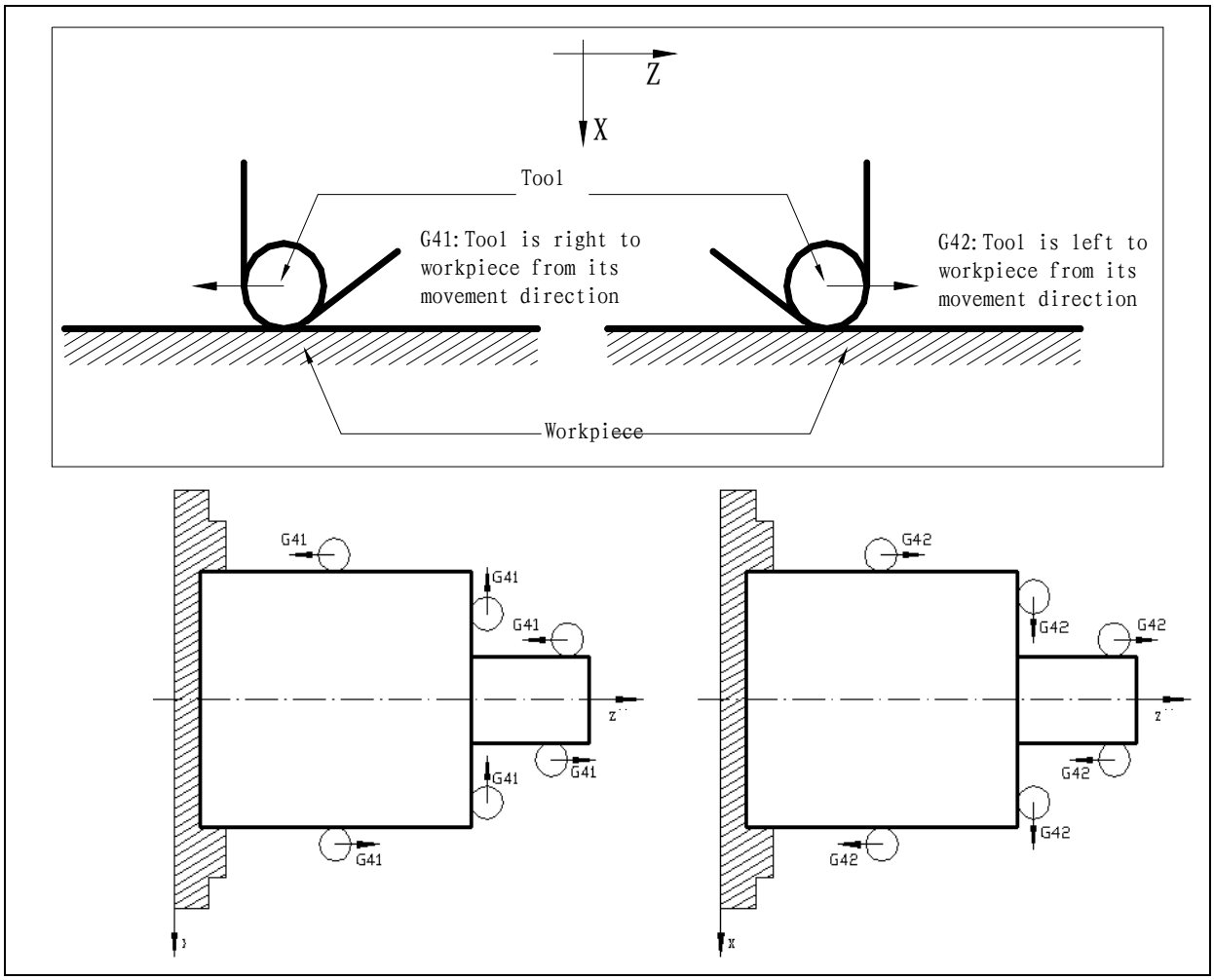
G40
G41 } X_ Z_ ;
G42

Commands	Function specifications	Remark
G40	Cancel the tool nose radius compensation	
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	

2.41.5 Compensation Direction

Specify its direction according to relative position between tool nose and workpiece when executing tool nose radius compensation.





2.41.6 Application

Machine a workpiece in the front tool post coordinate system . Tool number: T0101, tool nose radius R=2, imaginary tool nose number T=3.

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

Set the tool nose radius R and imaginary tool nose direction in "***TOOL OFFSET&WEAR"*** window as following:

No.	X	Z	R	T
001			2.000	3
002	…	…	…	…
…	…	…	…	…
007	…	…	…	…
008	…	…	…	…

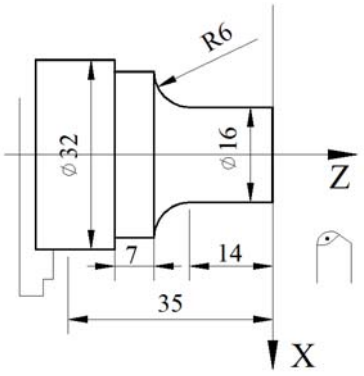
G00 X100 Z50 M3 T0101 S600;（Position, start spindle, tool change and execute tool compensation）

G42 G00 X0 Z3; （Set tool nose radius compensation）

G01 Z0 F300; （Start cutting）

X16;

```
Z-14 F200;
G02 X28 W-6 R6;
G01 W-7;
X32;
Z-35;
G40 G00 X90 Z40;      (Cancel tool nose radius compensation）
G00 X100 Z50 T0100;
M30;
```



2.42 Metric/Inch Switch G20、G21

Command format:G20;(inch input)
G21;(metric input)

Command specification:

input unit of CNC system is divided into two, i.e., metric unit: mm and inch unit: inch;
Modifying BIT0 (metric/inch): 0: metric input 1: inch input;
No.004＃0(SCW) output command unit change 0: metric output (0.001mm) 1: inch output (0.0001inch).

2.43 Editable Parameter Read/Write G10 L52

Command format: G10 L52; Set the parameter read or write method
N_/K_/D_/DT_/DC_ R(W)_; Read/write program of the system/K/D/DT/DC parameters
N_ P_ R(W)_; Read/write program of axis, spindle or channel parameter
…
G11; Cancel the parameter read/write

Code explanation: N_:Parameter number;
K_:K parameter number of PLC;
D_:D parameter number of PLC;
DT_:DT parameter number of PLC;
DC_:DC parameter number of PLC;
R(W) _:R is the read address from parameter, which only can be regarded as the macro variable number;W is the write-in value from parameter, which can be treated as the constant or marco variable;When the read/write parameter is the state one, its value from reading/writing is binary, and the others are decimal system;
P_:The axis number 1 to the top controllable axis number (from 1 to the Max. channel number) can be used for reading the axis, spindle or channel parameter.

Note:

1) **In the parameter write/read method, simultaneously, when two or more command words (including the repeated display for a same command word) of N_/K_/D_/DT_/DC_ are displayed at the same line, system alarm is then shown: 178 alarm, parameter (N_/K_/D_/DT_/DC_) (including the inexistent specified parameter) does not specify in the parametr read/write mode or the parameter is repeatedly specified;**
2) **In the parameter write/read method, when block specifies the operation command (R/W), but it does not exist any one command word of the N_/K_/D_/DT_/DC_, alternatively, there is the**

command word without data, system alarm is then shown: 178 alarm, parameter (N_/K_/D_/DT_/DC_) (including the inexistent specified parameter) does not specify in the parametr read/write mode or the parameter is repeatedly specified;

3) **In the parameter write/read method, when the block specifies the parameter number, but it does not exist in R/W operation command or R, W exist at the same time, system alam is then shown: 177 alarm; the read/write method does not specify or the read/write method is repeatedly specified in the parameter read/write method;**
4) **In the parameter write/read method, if there are command words other than the _/P_/K_/D_/DT_/DC_ R_/W_, ignore it, the command operations except from the parameter reading/writing will not be performed;**
5) **Specify P value in the block, if the axis corresponding set parameter does not exist, igore it;**
6) **When reading or writing the axis parameter or spindle parameter, if the P value does not occur, the system alarm is then displayed: 178 alarm; the axis number does not specify when reading/writing the axis parameter;**
7) **The system can not set the parameter with 2 authority levels or above if it is under the 3 or less authories. If the program exists the block to be modified the parameter with 2 or more authority levels, the system alarm is then displayed: 174 alarm, the current authority prohibits to be modified the parameter;**
8) **In the parameter write/read method, R_ can only be accompanied with macro variable, otherwise, the alarm occurs: 180 alarm, as well, R only can be specified the marco variable in this method;**
9) **When writing the bit parameter, if the W_ is followed with the operation number other than the binary constant and macro variable, the system alarm is then displayed: 176 alarm. The W value or the macro variable value followed with the W are not the enabled binary value;**
10) **When writing the bit parameter, when the value followed with the W exceeds the corresponding parameter range, the system alarm is then displayed: 175 alarm; The W value or the macro variable value followed with the W are not the enabled binary value;**
11) **The dual-channel system forbids other channels to perform the machining procedure in the parameter read/writing state; otherwise, the system alarm is then displayed: 186 alarm; the other channels can not performed the machining procedure during the parameter is read or written;**
12) **The parameter read/write function+n can not be used together with the tool life span administration function.**

Programming example:

1) State parameter read, write example

Program	**Explanation**
＃101 ＝ 10000000;	Macro variable No. 101 is the setting value of the state parameter No.2.
G10 L52;	Set the parameter read or write method
N1 R#100;	Read the state parameter No.1 to the macro variable No.100.
N1 W00000010;	Set the state parameter No.1 as 00000010.
N2 W#101;	Set the state parameter No.2 as the value of the macro variable No.101.
G11;	Cacnel the read or write method of the parameter.

2) Data parameter read/write example

Program	Explanation
＃101＝50;	Marco variable ＃101 is the setting value of the data parameter No.31.
G10 L52;	Set the parameter read or write method
N30 R#100;	Read data parameter No.30 to the macro variable ＃100.
N30 W200;	Set the data parameter No.30 as 200.
N31 W#101;	Set the data parameter No. to the value of the macro variable ＃101.
G11;	Cacnel the read or write method of the parameter.

3) Axle type parameter read or write example

Program	Explanation
＃103＝00000101;	Macro variable ＃103 is the setting value of the data parameter No.188 along with the 1st axis (X axis).
＃104＝100;	Macro variable ＃104 is the setting value of the data parameter No.23 along with the 5th axis (C axis).
＃105＝6000;	Macro variable ＃105 is the setting value of the data parameter No.38 along with the 2nd axis (S2).
G10 L52;	Set The read or write method of parameter.
N187 P1 R#100;	Read the 1st axis parameter value of the axle-type state parameter No, 187 to the macro variable ＃100.
N187 P5 W01100001;	Set the axle type state parameter No.187 along with the 5th axis is 01100001.
N188 P1 W#103;	Set the 1st axis of the axle type state parameter No.188 to the value of the macro variable ＃103
N22 P1 R#101;	Read the 1st axis parameter value of the axle type data parameter No.22 to the macro variable ＃101.
N22 P2 W8000;	Set the 2nd axis parameter of the axle type data parameter No.22 as 8000.
N23 P5 W#104;	Set the 5th axis of the axle type data parameter No.23 to the value of the macro variable ＃104
N37 P1 R#102;	Read the 1st axis parameter value of the axle type data parameter No.37 to the macro variable ＃102.
N37 P3 W8000;	Set the 3rd axis parameter of the spindle data parameter No.37 as 8000.
N38 P2 W#105;	Set the 5th axis of the spindle data parameter No.38 to the value of the macro variable ＃105
G11;	Cacnel the read or write method of the parameter.

Note: The read/write programming method of the K parameter is consistent with the one of the system state parameter; The read/write programming method of the D/DT/DC parameters are coincident with the one of the system data parameter.

2.44 Balanced Cutting G68, G69

Command format: G68: balanced cutting mode start
G69: balanced cutting mode cancel

Function explanation:

The balanced cutting mode starts when the program in channel 1, 2 specifies G68; it cancels when G69 is specified. When G68/G69 is specified in only one of two channels, the program waits.

In the balanced cutting mode, when cutting feedrate move command is specified in two channels, and the move amount is not 0, the system executes the balanced cutting. The balanced cutting is not executed in rapid traverse command.

In balanced cutting mode, programs simultaneously start to execute for each block having specified cutting feedrate traverse command.

The balanced cutting is not executed when any channel is in Dry Run, Machine Lock mode. But, executing G68/G69 makes the program wait.

Notes:

1.Specify G68 pr G69 in a single block.

2.The program does not wait when executing G68 after the balanced cutting has started or executing G69 after the balanced cutting has cancelled.

3.The balanced cutting function makes cutting feedrate in the two channels simultaneously execute, which does not ensure the following synchronous. For synchronously move the two tool posts, must simultaneously set feedrate commands, movement amount, feedrate, and feedrate override in the two channels.

4.The two channels execute pause during the course of balanced cutting, the balanced cutting cannot be executed when restart, but the balanced cutting wait is executed in the next block specifying cutting move command.

5.Cancel the balanced cutting function when emergency stop, reset.

2.45 Synchronous/Mix/Overlap Control

- **Synchronous control：**

Command format： G51.4 P_ Q_ L_;
G50.4 Q_;

Command function: G51.4 means synchronous control relationship of controllable axis specified by P, Q starts.
G50.4 means synchronous control relationship of controllable axis specified by Q with the slave axis releases.

Command explanation:G51.4 and G50.4 are non-modal G command in Group 00;

P:identification number of synchronous main axis(range:0～999);

Q:identification number of slave main axis(range:0～999);

L:parking state(range:0～2);

0:do not park and the parking is released.

1:the main control axis parks and parking of the salve control axis releases.

2:the salve control axis parks and parking of the main control axis releases.

Notes: L can be omitted and it is defaulted to 0 when it is omitted.

- **Mix control：**

Command format： G51.5 P_ Q_;

G50.5 P_ Q_;

Command function： G51.5 means mix control relationship of controllable axis specified by P, Q starts.

G50.5 means mix control relationship of controllable axis specified by P, Q Releases.

Command explanation: G51.5 and G50.5 are non-modal G command in Group 00;

P:identification number of mix axis 1 (range:0～999)

Q:identification number of mix axis 2(range:0～999)

- **Overlap control：**

Command format： G51.6 P_ Q_; overlap control starts

G50.6 Q_; overlap control releases

Command function: G51.6 means overlap control relationship of controllable axis specified by P, Q starts.

G50.6 means synchronous control relationship of controllable axis specified by Q with the slave axis releases.

Command explanation: G51.6 and G50.6 are non-modal G command in Group 00.

P:identification number of overlap main axis(range:0～999);

Q:identification number of overlap main axis(range:0～999).

- **Example：**

Identification number of X, Z in channel 1 is separately set to 11, 12; Identification number of X, Z in channel 2 is separately set to 21, 22.

Programs in channel 1	Programs in channel 2	Note
……	……	
M100	M100	Interchannel waits
G51.4 P12 Q22 L0		Z axis in channel 2 is synchronous with that of channel 1, but does not park.
M101	M101	
……	……	
M102	M102	
G50.4 Q22		Release Z axis in channel 2 being the slave control's synchronous relationship.
M103	M103	

- **Explanation:**

1、 P, Q identification number in synchronous/mix/overlap control needs to set in No.209.

2、 For synchronous/mix/overlap control, controllable axis selection signal or program commands can start/release synchronous/mix/overlap control. Namely, after controllable axis selection signal starts synchronous/mix/overlap control, the program command can release it; or after the program command starts synchronous/mix/overlap control, fall edge of controllable axis selection signal can release it.

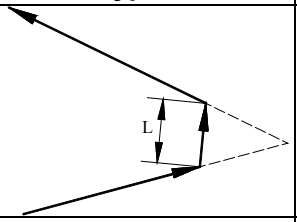
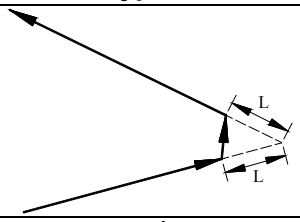
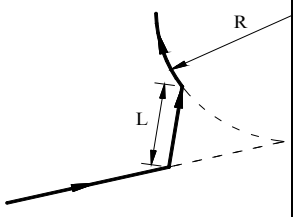
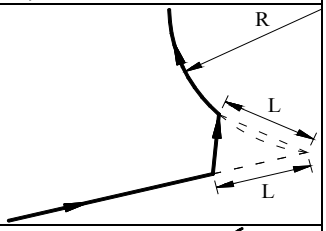
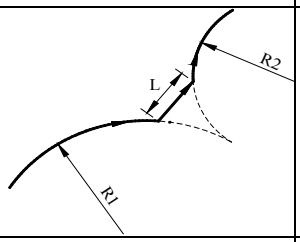
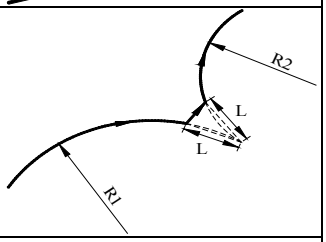
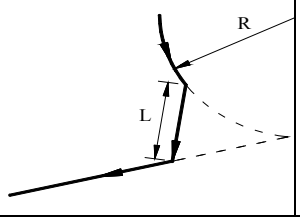
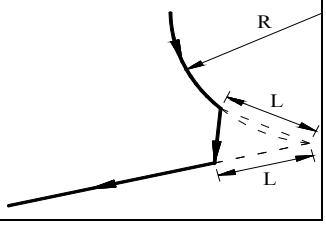
3、 Add the wait M code before starting/releasing synchronous/mix/overlap to pause program preread, which ensures data of controllable axis is normal after starting/releasing synchronous/mix/overlap control.

CHAPTER 3 AUTOMATIC CHAMFERING FUNCTION

3.1 Linear chamfering

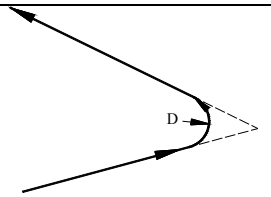
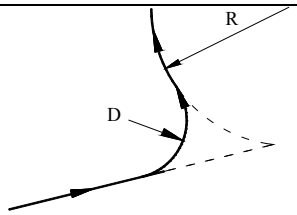
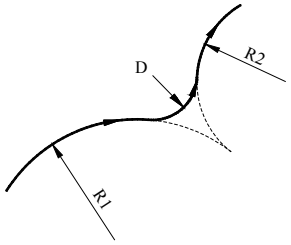
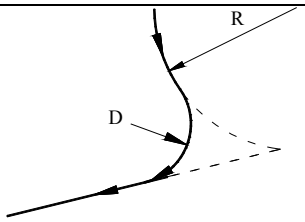
Linear chamfering: insert one straight line in the linear contours, arc contours, linear contour and arc contour. The command address of linear chamfering is L, the chamfering linear length is specified by L and its range is 0～1000 mm. L code is ignored when the value specified by L exceeds the range. The linear chamfering must be used in G01, G02 or G03 command.

Linear chamfering is divided into two types which main difference is different L definition in program. One L is a chamfering length and another is length between chamfering point and intersection point. The two types can be selected by No.181.0. In the following explanation, I type expresses L is the chamfering length and II type expresses L is the length between chamfering point and intersection point.

Connect type	command format	path	
		Type Ⅰ	Type Ⅱ
Linear to linear	G01 X(U)_ Z(W)_ L_ ; G01 X(U)_ Z(W)_ ;		
Linear to circular	G01 X(U)_ Z(W)_ L_; G02/G03 X(U)_ Z(W)_ R_; Or G01 X(U)_ Z(W)_ L_; G02/G03 X(U)_ Z(W)_ I_ K_;		
Circular to circular	G02/G03 X(U)_ Z(W)_ R_ L_; G02/G03 X(U)_ Z(W)_ R_; Or G02/G03 X(U)_ Z(W)_ I_ K_ L_; G02/G03 X(U)_ Z(W)_ I_ K_;		
Circular to linear	G02/G03 X(U)_ Z(W)_ R_ L_; G01 X(U)_ Z(W)_; Or G02/G03 X(U)_ Z(W)_ I_ K_ L_; G01 X(U)_ Z(W)_;		

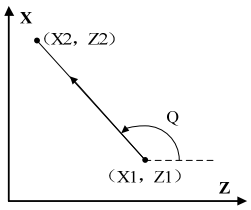
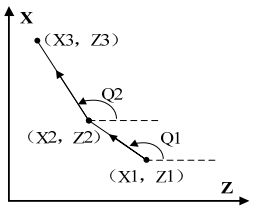
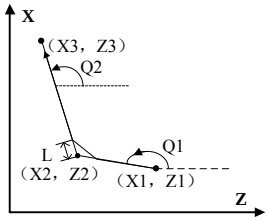
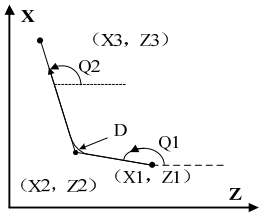
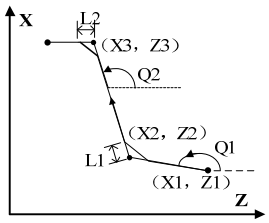
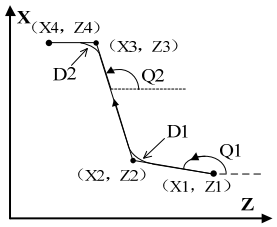
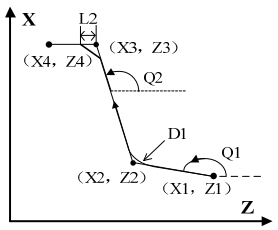
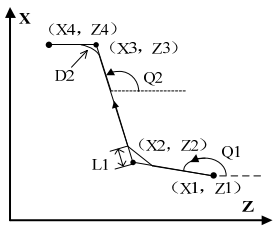
3.2 Circular chamfering

Circular chamfering: insert one circular between linear contours, circular contours, linear contour and circular contour, the circular and the contour line are transited by the tangent. The command of circular chamfering is D, and radius of chamfering arc is specified by D and its range is 0～1000mm. D is ignored when the value specified by D exceeds the range. The circular chamfering must be used in G01, G02 or G03.

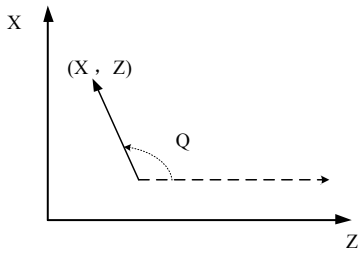
Connect type	command format	path
Linear to linear	G01 X(U)_ Z(W)_ D_; G01 X(U)_ Z(W)_;	
Linear to circular	G01 X(U)_ Z(W)_ D_; G02/G03 X(U)_ Z(W)_ R_; or G01 X(U)_ Z(W)_ D_; G02/G03 X(U)_ Z(W)_ I_ K_;	
Circular to circular	G02/G03 X(U)_ Z(W)_ R_ D_; G02/G03 X(U)_ Z(W)_ R_; or G02/G03 X(U)_ Z(W)_ R_ D_; G02/G03 X(U)_ Z(W)_ I_ K_; or G02/G03 X(U)_ Z(W)_ I_ K_ D_; G02/G03 X(U)_ Z(W)_ I_ K_; or G02/G03 X(U)_ Z(W)_ I_ K_ D_; G02/G03 X(U)_ Z(W)_ R_;	
Circular to linear	G02/G03 X(U)_ Z(W)_ R_ D_; G01 X(U)_ Z(W)_; or G02/G03 X(U)_ Z(W)_ I_ K_ D_; G01 X(U)_ Z(W)_;	

3.3 Function of Directly Inputting Graphic Dimension

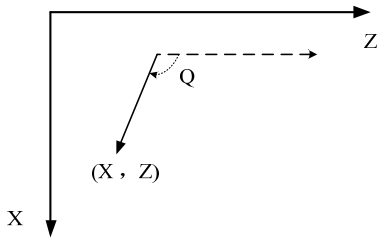
The function of directly inputting graphic dimension can make the user directly use the linear angle, chamfering value in the machining drawing to program.

Command format	Command path	Command format	Command path
G01 X2_ (Z2_) Q_		G01 Q1_ G01 X3_ Z3_ Q2_	
G01 Q1_ L_ G01 X3_ Z3_ Q2_		G01 Q1_ D_; G01 X3_ Z3_ Q2_;	
G01 Q1_ L1_ G01 X3_ Z3_ Q2_ L2_ G01 X4_ Z4_		G01 Q1_ D1_ G01 X3_ Z3_ Q2_ D2_ G01 X4_ Z4_	
G01 Q1_ D1_ G01 X3_ Z3_ Q2_ L2_ G01 X4_ Z4_		G01 Q1_ L1_ G01 X3_ Z3_ Q2_ D2_ G01 X4_ Z4_	

Note:Q_ is an angle between straight line and Z axis,its range:-99999999~99999999（unit：0.001degree），its angle direction is shown below:



Rear tool post coordinate system



Front tool post coordinate system

CHAPTER 4 MACRO COMMANDS

GSK980TTi provides the macro command which is similar to the high language, and can realize the variable assignment, and subtract operation, logic decision and conditional jump by user macro command, contributed to compiling part program for special workpiece, reduce the fussy counting and simplify the user program.

4.1 Macro Variables

- **Presentation of macro variables**

  Present with "#"＋ macro variables number.;

  Format: # i(i=100,102,103,……) ;

  Alternatively, the variable is specified by "#"+[Expression]

  Format:# [i] (i = #100 + #102 + #103）

  Example:#105，#109，# [#125 + #100 * #111]。

- **Variable Type**

  The variable is divided into four types according to the variable number:

Number NO.	Variable type	Function
#0	Null variable	The variable is null and is not valued.
#1~#33	Local variable	The local variable is used to store data in the macro program, such as result. When the system is turned off, the local variable is initialized to be null. When the macro program is called, the argument values 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	System variable

- **Macro variables reference**

1. Macro variables can replace command values

   Format: ＜Address＞+"# i" or ＜Address＞+"－# I". It shows the system takes variable value or negative value of variable value as address value.

   Example: F#103…when #103=15, its function is the same that of F15;

   Z-#110…when #110=250, its function is the same that of Z-250;

2. Macro variables can replace macro variables values.

   Format: "＃"+"9"+macro variables number

   Example: if #100 = 205, #105 = 500,

   The command function of X#9100 is the same as X500;

   The command function of X-#9100 is the same as X-500

Null variable

1) When an undefined variable (null variable) is referred, the address is ignored.

#1=<null>	#1=0
G00 X100 Z#1 is equal to G00 X100	G00 X100 Z#1 is equal to G00 X100 Z0

2). Operation

Except for using <null variable > to value, the <null variable> used to operation in other conditions is the same as that of “0”.

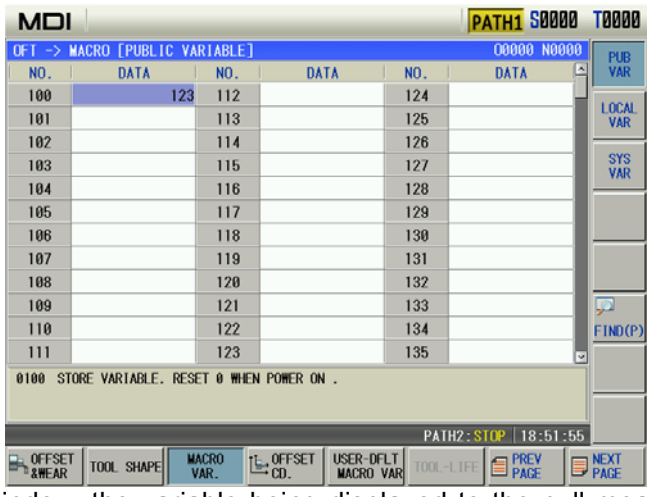
#1=<null>	#1=0
#2=#1 ↓(execution result) #2=<null>	#2=#1 ↓(execution result) #2=0
#2=#1 * 5 ↓(execution result) #2=0	#2=#1 * 5 ↓(execution result) #2=0
#2=#1+#1 ↓(execution result) #2=0	#2=#1+#1 ↓(execution result) #2=0

3). Condition expression

<null>s in EQ(=) & NE(≠) are different to “0”.

#1=<null>	#1=0
#1 EQ #0 ↓ Valid	#1 EQ #0 ↓ Invalid
#1 NE #0 ↓ Invalid	#1 NE #0 ↓ Invalid
#1 GE #0 ↓ Valid	#1 GE #0 ↓ Invalid
#1 GT #0 ↓ Invalid	#1 GT #0 ↓ Invalid

Variable display



(1) In macro window, the variable being displayed to the null means it is null, i.e. it is not defined.

(2) The share variable (#100~#199, #500~#999) values are displayed in the macro variable

window, and is also displayed the window, the data is input directly to value the share variable.

- **System variable**

（1）Interface signal: CNC only executes G and F signals. Whether there are I/O to correspond to it is defined by PLC.

Variable No.	Function
#1000~#1015	Correspond G54.0~G54.7, G55.0~G55.7 signal states
#1032	Correspond G54, G55 signal states
#1100~#1115	Correspond F54.0~G54.7, F55.0~F55.7 signal states
#1132	Correspond F54, F55 signal states
#1133	Correspond F56, F57, F58, F59 signal states

(2) Tool compensation system variable:

Compensation No.	X compensation value		Z compensation value		Tool tip radius compensation value		Imagery tool tip position T	Y compensation value	
	Wear	Geometric shape	Wear	Geometric shape	Wear	Geometric shape		Wear	Geometric shape
1	#2001	#2701	#2101	#2801	#2201	#2901	#2301	#2401	#2451
…	…	…	…	…	…	…	…	…	…
32	#2032	#2732	#2132	#2832	#2232	#2932	#2332	#2432	#2482

(3) Machined workpiece number:

Variable No.	Function
#3901	Machined workpiece number(completion)
#3902	The Max. machining numbers

(4) System modal information variable

Variable No.	Function	
#4001	G00, G01, G02, G03, G05, G32, G33, G34, G80, G84, G88, G90, G92, G94, G124(G06.2), G126(G06.3),G132(G32.1),G144(G07.2),G146(G07.3)	No. 1 group
#4002	G96, G97	No. 2 group
#4005	G98, G99	No. 3 group
#4006	G20, G21	No. 6 group
#4007	G40, G41, G42	No. 7 group
#4012	G66, G67	No. 12 group
#4013	G54,G55,G56,G57,G58,G59	No. 14 group
#4016	G17,G18,G19	No. 16 group
#4109	F command	
#4113	M command	
#4114	Serial No.	
#4115	Program No.	
#4119	S command	
#4120	T command	

(5) System variable of coordinate position information:

Variable No.	Position signal	Coordinate system	Tool compensation value	Read in running
#5001~#5005	End point of block	Workpiece coordinate system	Not including	Possible
#5021~#5025	Current position	Machine coordinate system	Including	impossible
#5041~#5045	Current position	Workpiece coordinate system		

Note:The position listed in the above table separately corresponds orderly to X, Y, Z, 4th, 5th axis. For example: #5001 meanings to be X position information, #5002 meanings to be Y position information, #5003 meanings to be Z position information and #5004 meanings to 4th position information and #5005 meanings to 5th position information.

(6) The system variable of MPG interruption value: Readable variable

Variable No.	Function
#5121	The 1st axis MPG interruption value
#5122	The 2nd axis MPG interruption value
#5123	The 3rd axis MPG interruption value
#5124	The 4th axis MPG interruption value
#5125	The 5th axis MPG interruption value

（7）Compensation values of workpiece coordinate systems

Variable No.	Function
#5201~#5205	External workpiece zero offset value of 1st ~5th axis
#5221~#5225	G54 workpiece zero offset value of 1st ~5th axis
#5241~#5245	G55 workpiece zero offset value of 1st ~5th axis
#5261~#5265	G56 workpiece zero offset value of 1st ~5th axis
#5281~#5285	G57 workpiece zero offset value of 1st ~5th axis
#5301~#5305	G58 workpiece zero offset value of 1st ~5th axis
#5321~#5325	G59 workpiece zero offset value of 1st ~5th axis

Note:Message described in the above table separately corresponds to X, Y, Z, the 4th, 5th axis. For example, #5201 means the offset message of X axis, #5202 means the one of Y axis，#5203 means the one of Z axis,#5204 means the one of the 4th axis, #5205 means the one of the 5th axis.

4.2 Macro Commands G65

4.2.1 G65 format

Command format:G65 Hm P# i Q# j R# k;

Command specification:

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

#i: macro variables name for storing values.

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

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

i = #j O # k

Operation sign specified by Hm

Macro command list :

Command format	Functions	Definitions
G65 H01 P#i Q#j	Assignment	# i = # j assign value of j to i
G65 H02 P#i Q#j R#k;	Decimal add operation	# i = # j + # k
G65 H03 P#i Q#j R#k;	Decimal subtract operation	# i = # j - # k
G65 H04 P#i Q#j R#k;	Decimal multiplication operation	# i = # j×# k
G65 H05 P#i Q#j R#k;	Decimal division operation	# i = # j÷# k
G65 H11 P#i Q#j R#k;	Binary addition	# i = # j OR # k
G65 H12 P#i Q#j R#k;	Binary multiplication(operation)	# i = # j AND # k
G65 H13 P#i Q#j R#k;	Binary exclusive or	# i = # j XOR # k
G65 H21 P#i Q#j;	Decimal square root	# i = $\sqrt{\#j}$
G65 H22 P#i Q#j;	Decimal absolute value	# i = |# j |
G65 H23 P#i Q#j R#k;	Decimal remainder	Remainder of # i = (#j÷# k)
G65 H24 P#i Q#j;	Decimal into binary	# i = BIN(# j)
G65 H25 P#i Q#j;	Binary into decimal	# i = DEC(# j)
G65 H26 P#i Q#j R#k;	Decimal multiplication/division operation	# i = # i×# j÷# k
G65 H27 P#i Q#j R#k;	Compound square root	# i = $\sqrt{\#j^2+\#k^2}$
G65 H31 P#i Q#j R#k;	Sine	# i = # j×sin(# k)
G65 H32 P#i Q#j R#k;	Cosine	# i = # j×cos(# k)
G65 H33 P#i Q#j R#k;	Tangent	# i = # j×tan(# k)
G65 H34 P#i Q#j R#k;	Arc tangent	# i = ATAN(# j / # k)
G65 H80 Pn;	Unconditional jump	Jump to block n
G65 H81 Pn Q#j R#k;	Conditional jump 1	Jump to block n if # j = # k,otherwise the system executes in order
G65 H82 Pn Q#j R#k;	Conditional jump 2	Jump to block n if # j ≠ # k, otherwise the system executes in order
G65 H83 Pn Q#j R#k;	Conditional jump 3	Jump to block n if # j > # k, otherwise the system executes in order
G65 H84 Pn Q#j R#k;	Conditional jump 4	Jump to block n if # j < # k, otherwise the system executes in

Command format	Functions	Definitions
		order
G65 H85 Pn Q#j R#k;	Conditional jump 5	Jump to block n if # j ≥ # k, otherwise the system executes in order
G65 H86 Pn Q#j R#k;	Conditional jump 6	Jump to block n if # j ≤# k, otherwise the system executes in order
G65 H99 Pn;	P/S alarm	(500+n) alarms

4.2.2 Program Example with Macro Command

Differences between user macro program call (G65, G66) and subprogram call (M98) are as follows:

1. G65, G66 can specify the argument data and send them to macro program and M98 has no such function.
2. G65, G66 can change the level of local variable and M98 has no such function.
3. G65, G66 only follows N and only P or H follows them.

4.2.2.1 One-Shot Call G65

Command format: G65 P_ L_ <argument>_;

Macro program specified by P is called, the argument (data) is sent to the user macro program body.

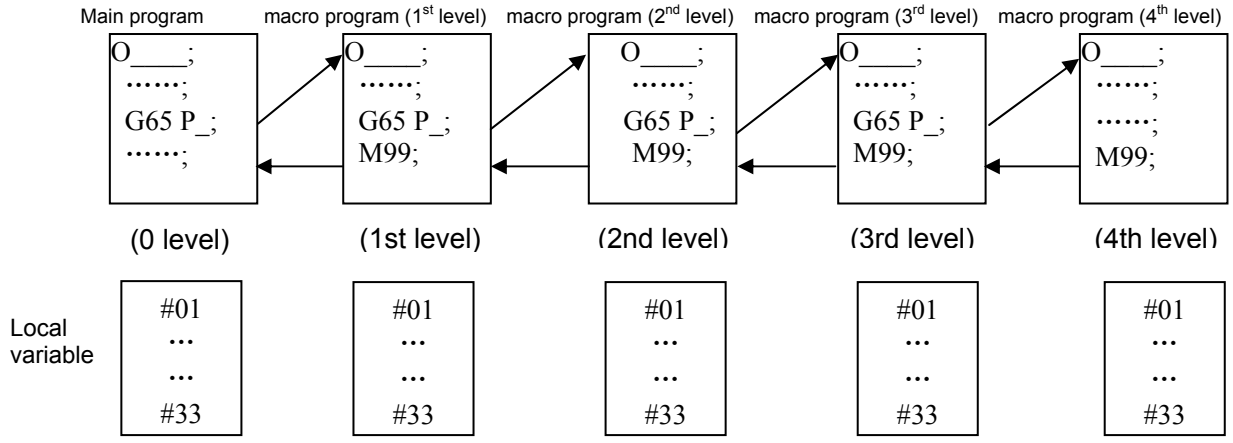
Command explanation:

P —— called macro program number;

L —— called times (it is 1 when it is omitted, it can be the repetitive times from 1 to 9999);

<Argument> ___ data sent to macro program is valued with the corresponding local variable.

Nest call: G65 call has four-level nest.



Specifying argument: the argument can be specified by two forms.

Method 1: Use the letter besides G, L, O, N, P, and each is only specified one time, and the last which is specified many times is valid.

Argument address and corresponding variable No. table in method 1

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

Method II: Use A, B, C and Ii, Ji, Ki (I is 1~10), the used letter and executed times (I, J, K) automatically decides the corresponding variable number of argument. The argument in the method specifies A, B, C one time for each and I, J, K up to 10 times (10 times replacing the more).

Argument address and corresponding variable No. table in method II

Address	Variable No.		Address	Variable No.		Address	Variable No.
A	#1		K_3	#12		J_7	#23
B	#2		I_4	#13		K_7	#24
C	#3		J_4	#14		I_8	#25
I_1	#4		K_4	#15		J_8	#26
J_1	#5		I_5	#16		K_8	#27
K_1	#6		J_5	#17		I_9	#28
I_2	#7		K_5	#18		J_9	#29
J_2	#8		I_6	#19		K_9	#30
K_2	#9		J_6	#20		I_{10}	#31
I_3	#10		K_6	#21		J_{10}	#32
J_3	#11		I_7	#22		K_{10}	#33

Note 1: The subscripts of I, J, K are used to confirming the specified sequence of argument, and are not written in the actual programming.

Note 2: The system can identify the variable number according to the present sequence and times of I, J, K in the method.

Note 3: CNC internal can automatically identify the argument specifying mode I and II. When the two modes are specified, the later specified is valid.

Example：

G65 P9010 A1 B2 C3 I14 J15 I6 J7 K9 K11 K12 J30;

Calling program O9010，the argument A ec. Are passed to the local cariable at the same time.the corresponding relationship ia as follows：

#1=1，#2=2，#3=3，#4=14，#5=15，#7=6，#8=7，#6=9，#9=11，#12=12，#11=30;

4.2.2.2 Modal Call G66

Command format: G66 P_ L_ <Argument>_;

Command explanation: P ___ called macro program number

L ___ called times (It is 1 when it is omitted, it can be the repetitive times from 1 to 9999)

<Argument> ___ data sent to macro program is valued with the corresponding local variable.

Nest call: G66 call has four-level nest.

Note:

1) **Macro program is called firstly after G66 block is executed.**
2) **The macro program is called again after G00, G01, G02, G03, G05 are executed (after G66 is**

executed and before the modal call is cancelled).

3) Call the value which is updated from the argument to the local variable.

4) **G65 call will automatically cancel G66 modal call.**

4.2.2.3 Modal Call Cancel G67

Command format: G67;

Command explanation: cancel G66 modal macro program call

Application:

O2005（O2005）;

G00 X100 Z50;

G66 P0100 L2 A2 B20 C20 I30 J20 K20;call P0100 two times when the system executes the block

G01 X80 Z50; call P0100 two times (update the local variable according to the argument) after the system has executed the block

G67; cancel G66 modal call

G01 X20 Z50; the system does not call P0100 after it executes the block

M30;

4.3 Statement Macro Command

4.3.1 Arithmetic & Logic Operation

Function	Expression format	Remark
Definition or assignment	#i = #j	
addition subtraction multiplication division	#i = #j + #k #i = #j - #k #i = #j * #k #i = #j / #k	
Or And Exclusive Or	#i = #j OR #k #i = #j AND #K #i = #j XOR #K	Logic operation is executed by the binary system
Square root Absolute value Rounding-off FUP FIX Natural logarithm Exponential function	#i = SQRT[#j] #i = ABS[#j] #i = ROUND[#j] #i = FUP [#j] #i = FIX [#j] #i = LN[#j] #i = EXP[#j]	
Sine Arc sine Cosine Arc cosine Tangent Arc tangent	#i = SIN[#j] #i = ASIN[#j] #i = COS[#j] #i = ACOS[#j] #i = TAN[#j] #i = ATAN[#i]/ [#j]	Angle unit is specified by degree. For example: 90°30’ is expressed by 90.5°
BCD to BIN BIN to BCD	#i = BIN[#j] #i = BCD[#j]	Used for switching with PMC

Relative explanation:

1. Angle unit

   Angle units of SIN, COS, ASIN, ACOS, TAN and ATAN are degree (°). For example: 90°30´means to be 90.5° (degree).

2. Arc sine # i=ASIN[#j]

   i. result output range:

   No.180#7 NAT is set to 1: 90°~ 270°;

   No.180#7 NAT is set to 0: -90°~ 90°;

   ii. when #j exceeds the range from -1 to 1, the system alarms P/S.

   iii. the constant replaces the variables #j.

3. Arccosine # i =ACOS[#j]

   i. Result output range 180°~ 0°.

   ii. When #j exceeds the range from -1 to 1, the system alarms P/S.

   iii. The constant replaces the variables #j.

4. Arc tangent #i=ATAN[#j]/[#k]

   Specify the lengths of two sides and separate them with a slash “/”.

   i. Result output range:

   When No.180#7 NAT is set to 1: 90°~ 270°;

   [For example] #1=ATAN[-1]/[-1]: #1=225°;

   When No.180#7 NAT is set to 0 -90°~ 90°;

   [For example]#1=ATAN[-1]/[-1]: #1=45.0°;

   ii. The constant replaces the variables #j.

5. Natural logarithm #i=LN[#j]

   The constant replaces the variables #j

6. Exponential function #i=EXP[#j]

   The constant replaces the variables #j

7. ROUND function

   When arithmetical operation or logic operation IF or WHILE includes ROUND, ROUND rounds in the first decimal place.

   For example: #1=ROUND[#2]: #2=1.2345, the variables 1 is 1.0.

8. FUP FIX

After CNC executes the operation, the result integer absolute value is bigger the previous absolute value, which is called FUP; the result integer absolute value is less than the one, which is call FIX. Pay more attention to the negative execution.

Example:

Hypothetically, #1=1.2, #2= -1.2

When #3=FUP[#1] is executed, 2.0 is assigned to #3.

When #3=FIX[#1] is executed, 1.0 is assigned to #3.

When #3=FUP[#2] is executed, -2.0 is assigned to #3.

When #3=FIX[#2] is executed, -1.0 is assigned to #3.

4.3.2 Transfer & Cycle

In the program, the system uses GOTO and IF statement to change the control flow. There are three types of transfer and cycle operation.

1) Unconditional transfer (GOTO statement)

Transfer to the block which serial number is n. The system alarms when others exceeds the range from 1 to 99999, and it specifies the serial number with the statement.

Format:GOTO n; n: serial number(1~99999)

Example:GOTO 1;

GOTO #101;

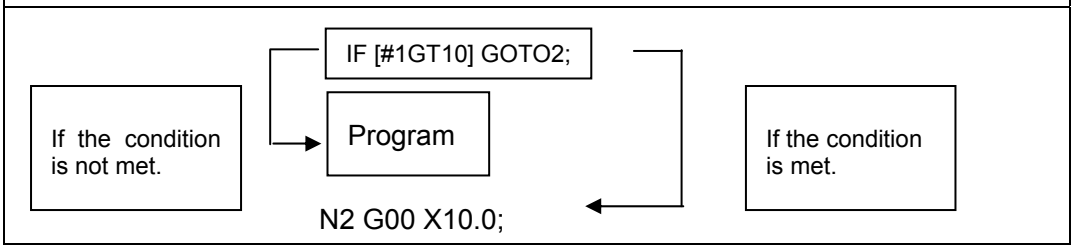
2) Conditional control (IF statement)

GOTO format: IF[conditional statement]GOTO n;

When the specified conditional statement is valid, the system transfers to the block which serial number is n; When the specified conditional statement is valid, the system executes the next block.

Example:

When the variable #1 is more than 10, the system transfers to the block which serial number is N2.



3) Conditional expression:

the conditional expression must include the conditional operator, two sides of conditional operator can be variable, constant or expression, and it must be closed with the brackets '[' ']'.

the system uses the conditional operators listed in the following table.

Conditional operator	Meaning
EQ or = =	Equal to (=)
NE or <>	Not equal to (≠)
GT or >	More than (>)
GE or >=	More than or equal to (≥)
LT or <	Less than (<)
LE or <=	Less than or equal to (≤)

Example: IF[3<>2]GOTO 2; its meaning: when 3 is not equal 2, the system skips to N2 block;

IF[#101>=7.22]THEN #101=SIN30; its meaning: when #101 is more than or equal to 7.22, the system executes the assignment after THEN. i.e. the sine value of 30 degree is assigned to the variable #101.

4) THEN format:

IF[conditional expression]THEN<macro program statement >;

When the condition expression is valid, the system executes only one statement following THEN.

Example: IF[#1 EQ #2] THEN #3=0;

When #1 value is equal to the #2, 0 is assigned to the variable #3; when they are

not

equal, the system orderly executes the followings instead of the assignment statement after THEN.

5) Typical program:

the following program counts the sum of the integer 1~10.

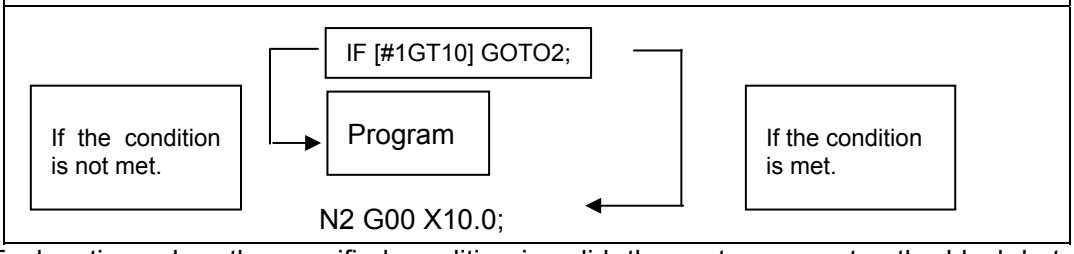
```
O9500
#1=0;… …the sum is initialized to be 0
#2=1;… …the summand number is initialized to be 1
N1 IF[#2 GT 10]GOTO2;… … the system skips to N2 when the summand is more than 10
#1= #1+#2;  … …count the sum of two numbers
#2= #2+1;  … … the summand adds 1
GOTO1;  … … unconditionally skip to the block N1
N2 M30;  … … end of program
```

6) Cycle (WHILE statement)

Specify one conditional expression after WHILE. When the specified conditional is valid, the system executes the blocks between DO and END; otherwise, the system skips to the block after END.

Example:

When the variable #1 is more than 10, the system transfers to the block which serial number is N2.



Explanation: when the specified condition is valid, the system executes the block between DO and END; otherwise, executes the block after END. The two tabs after DO and END are consistent, and the tab value can be 1, 2 or 3, otherwise, the system alarms.

Nest: the tab (1~3) in DO, END can be used many times. But the system alarms when there is the intercross repetitive cycle in the program.

4.3.3 G Code Macro Program Call

Command format:G□□□□ L_ < Agrument >_

Code value（0~9999）

Command specification: Code value: It is set by data parameter №.86~№.95;

L: The called times (It defaults to 1 if it omits, which can be specified the repeated times from 1 to 9999);

<Argument>: The data converted to the macro program, its value is assigned to the corresponding local variable, it is same that the G65 calls macro program.

Data parameter No.86~No.95 set the used G code in the macro command call, use the set G code, that is, the macro program call using G code can be defined. If the data parameters No.86~No.95 are set to 0, this function will be then disabled. The callable macro programs are O9010~O9019, the corresponding relationships between parameter number and program number are as follows:

Para. No.	Program No.	Para. No.	Program No.
86	O9010	91	O9015
87	O9011	92	O9016
88	O9012	93	O9017
89	O9013	94	O9018
90	O9014	95	O9019

For example:№.86 = 100、№.87 = 103 then:

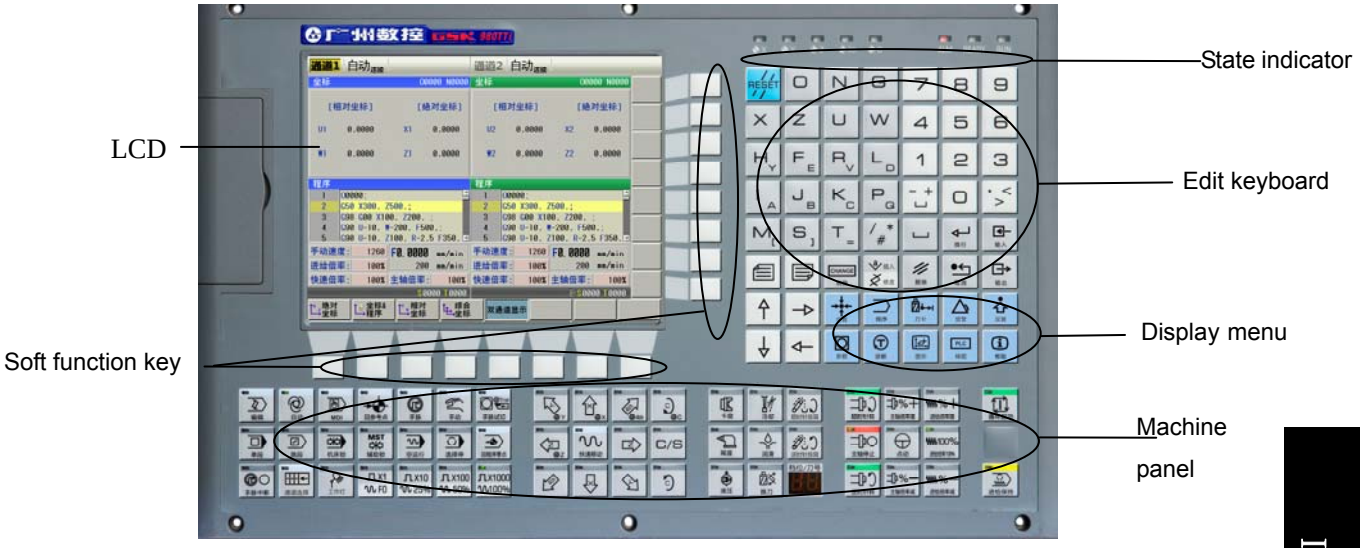
```
G00 X0 Z0;
G100;      // Call the No. O9010 macro program
G103;      // Call the No. O9011 macro program
…;
M30;
```

II Operation

CHAPTER 1 OPERATION MODE AND DISPLAY INTERFACE

1.1 Panel Division

GSK980TTi, GSK980TTi-V CNC system uses an integrated panel, which is divided as follows:



GSK980TTi panel division

1.2 Summary of Operation Mode

There are 8 modes in GSK980TTi, which are Edit, Auto, MDI, Machine zero, Step/MPG, Manual, Program Zero, and MPG trial-cut modes.

- **Edit mode**
  In this mode, the operation of part program setup, deletion and alteration can be performed.
- **Auto mode**
  In this mode, the program is executed automatically.
- **MDI mode**
  In this mode, the operation of parameter input, command blocks input and execution can be performed.
- **Machine zero mode**
  In this mode, the operation of X, Z machine zero return can be performed separately.
- **MPG / Step mode**
  In the Step/MPG feed mode, the moving is performed by an increment selected by CNC system.
- **Manual mode**
  In this mode, the operation of Manual feed, Manual Rapid, feedrate override adjustment, Rapid override adjustment and spindle ON/OFF, cooling ON/OFF, Lubricating ON/OFF, spindle jog, manual tool change can be performed.
- **Program zero return mode**
  In this mode, the operation of X, Z program zero return can be performed separately.

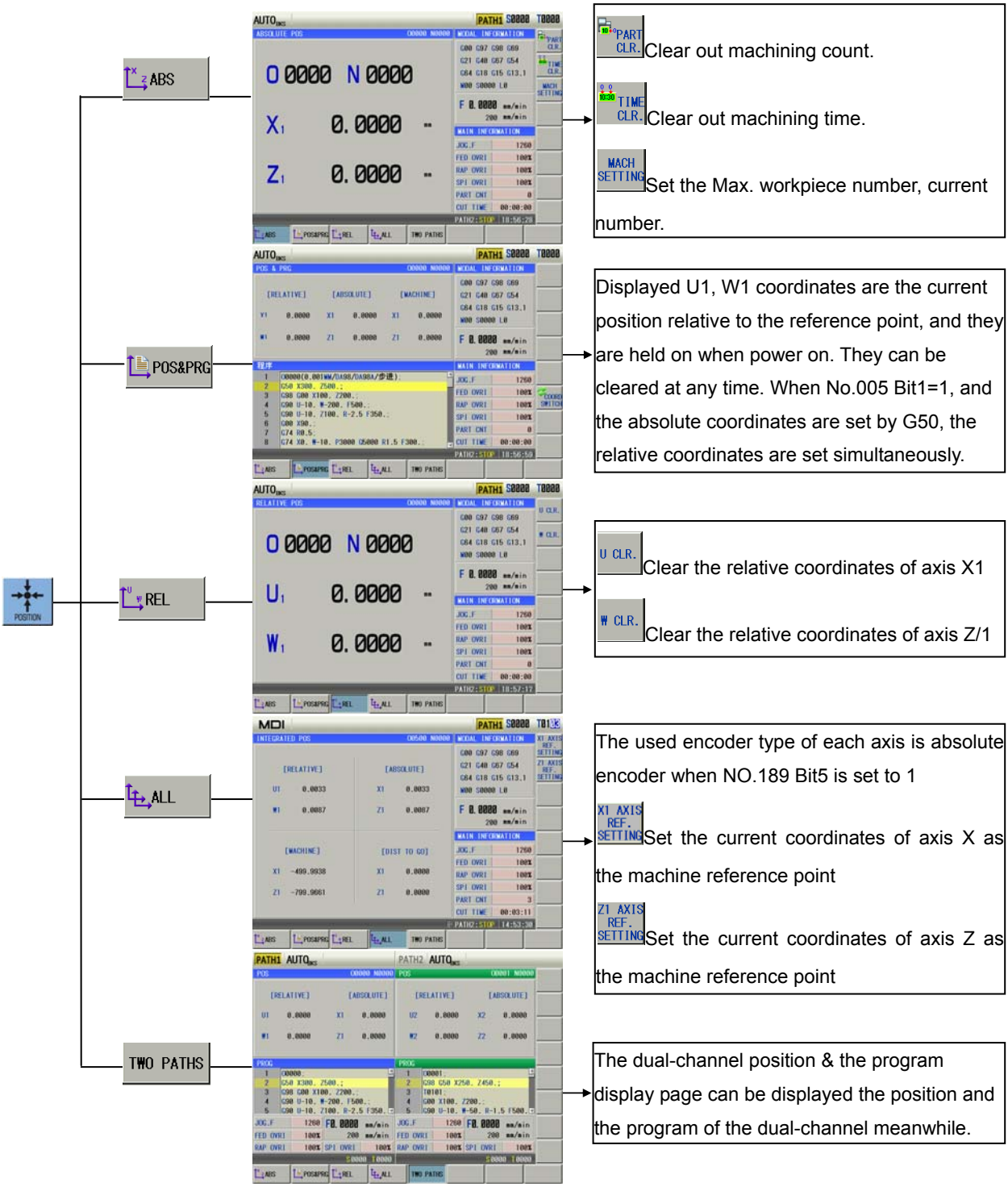
- **MDP trail-cut mode**

  In this mode, rotating the MPG can control program's execution speed, which can check whether the machining program is correct.

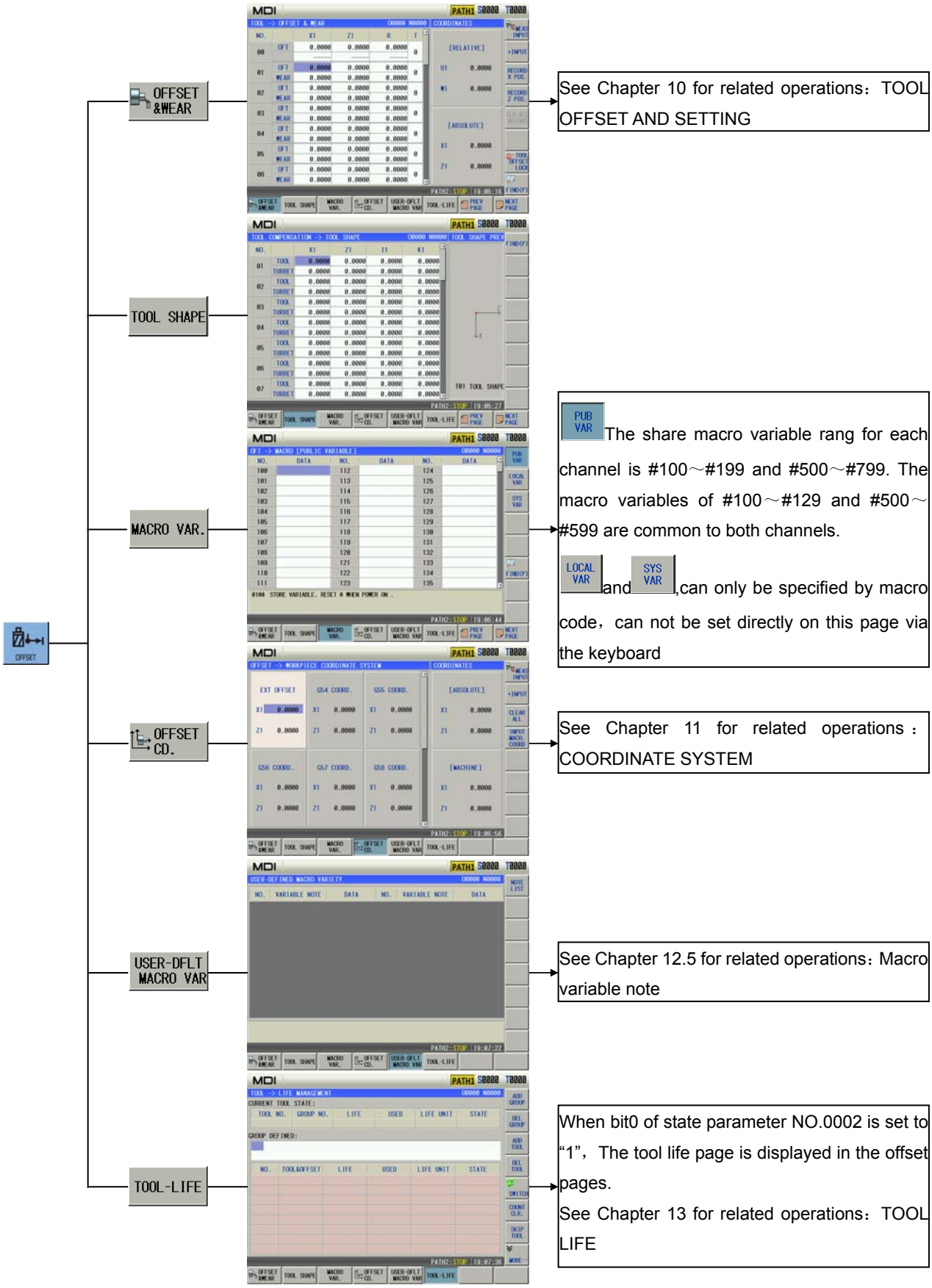
CHAPTER 2 DISPLAY INTERFACE

GSK980TTi has 9 function keys including POS, PRG, SET, etc. on its edit keyboard. Each function key corresponds to one interface which has many pages and operation soft keys.

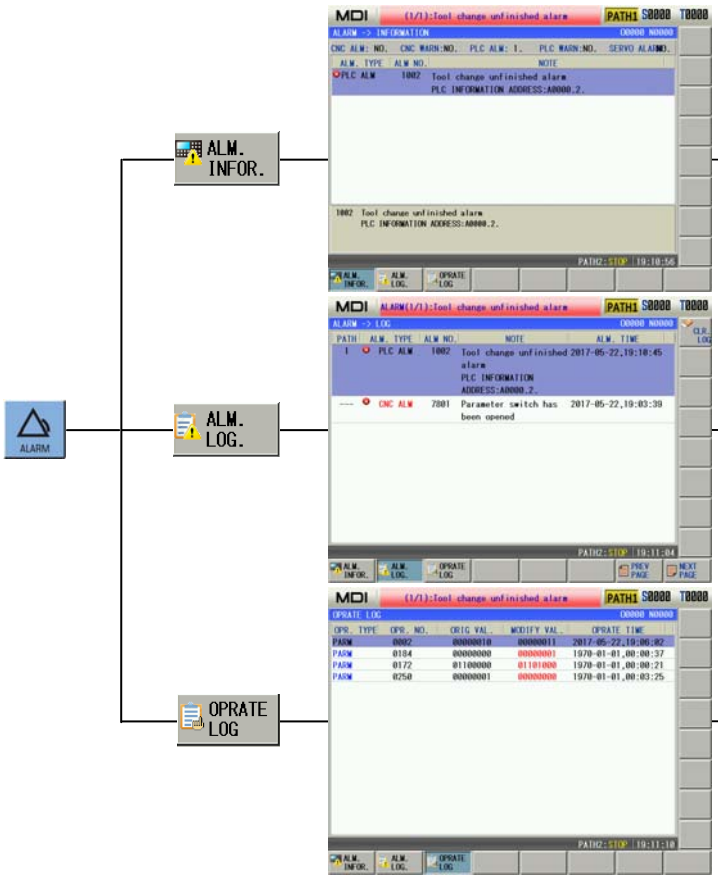
2.1 POS interface



2.2 TOOL OFFSET Interface



2.3 ALARM interface



Note: Alarm clearing: It can clear alarms by pressing  .

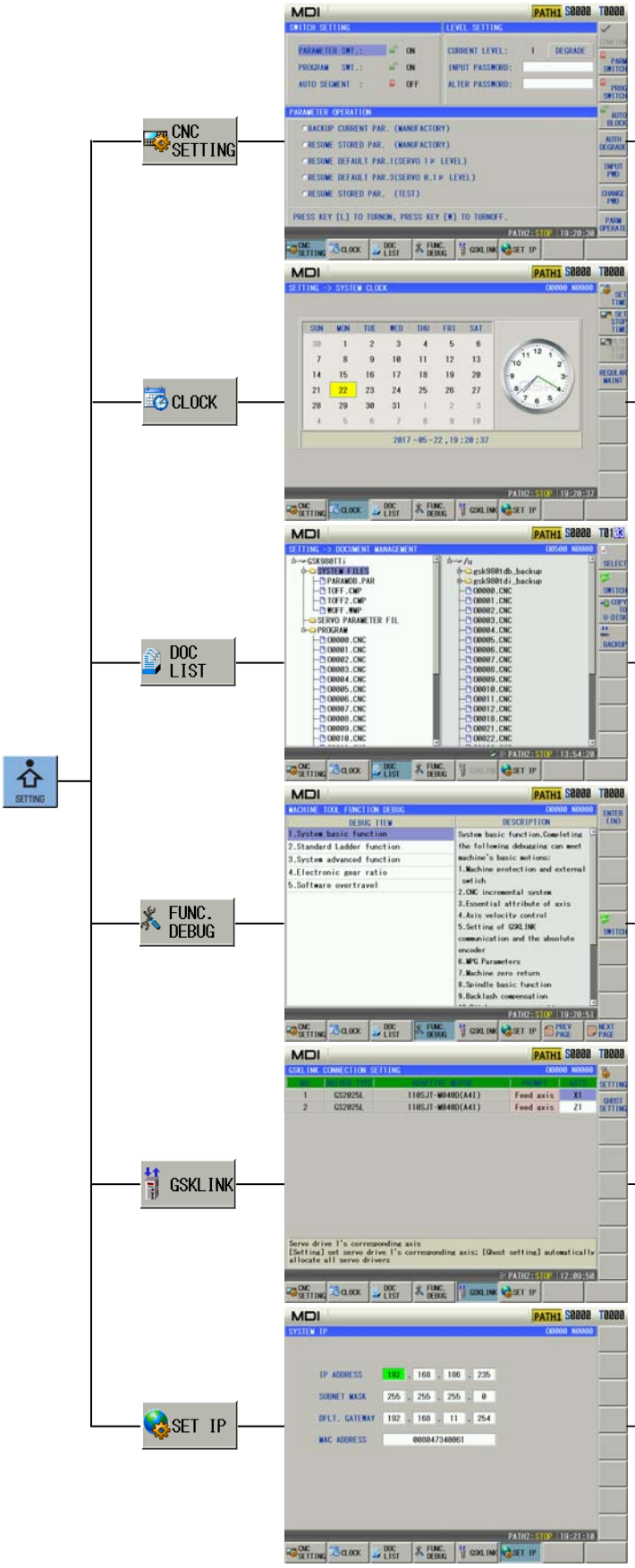
When the operation level is above or equal to the 2nd grade，can clear the history of all alarms and prompts，after clearing the page is

Can view the parameters and tool offset to modify the record.

2.4 GRAPH Interface

●Start the drawing；
●Clear the current graphic path
●Press ZOOM+ each time, the graphic path is scaled up $\sqrt{2}$ fold
●Press ZOOM- each time, the graphic path is scaled down $\sqrt{2}$ fold
●Do regulate the graphic center on when the path exceeds the display area.
●Switch X, Z axis
●Adjust mobile intervals
●Display the graphic attribution window, including the current coordinate system, translation position, scaling and mobile interval.

2.5 SETTING interface



See Chapter 12.1 and 12.2 for related operations：DATA SETTING

- Press [SET TIME] soft key to enter the alteration mode.
- Move the cursor to the required modification (Year/Month/Date/Minute/Second).
- Input directly numbers, and press[DATA INPUT] key, then [SET TIME] soft key to escape from the alteration mode.

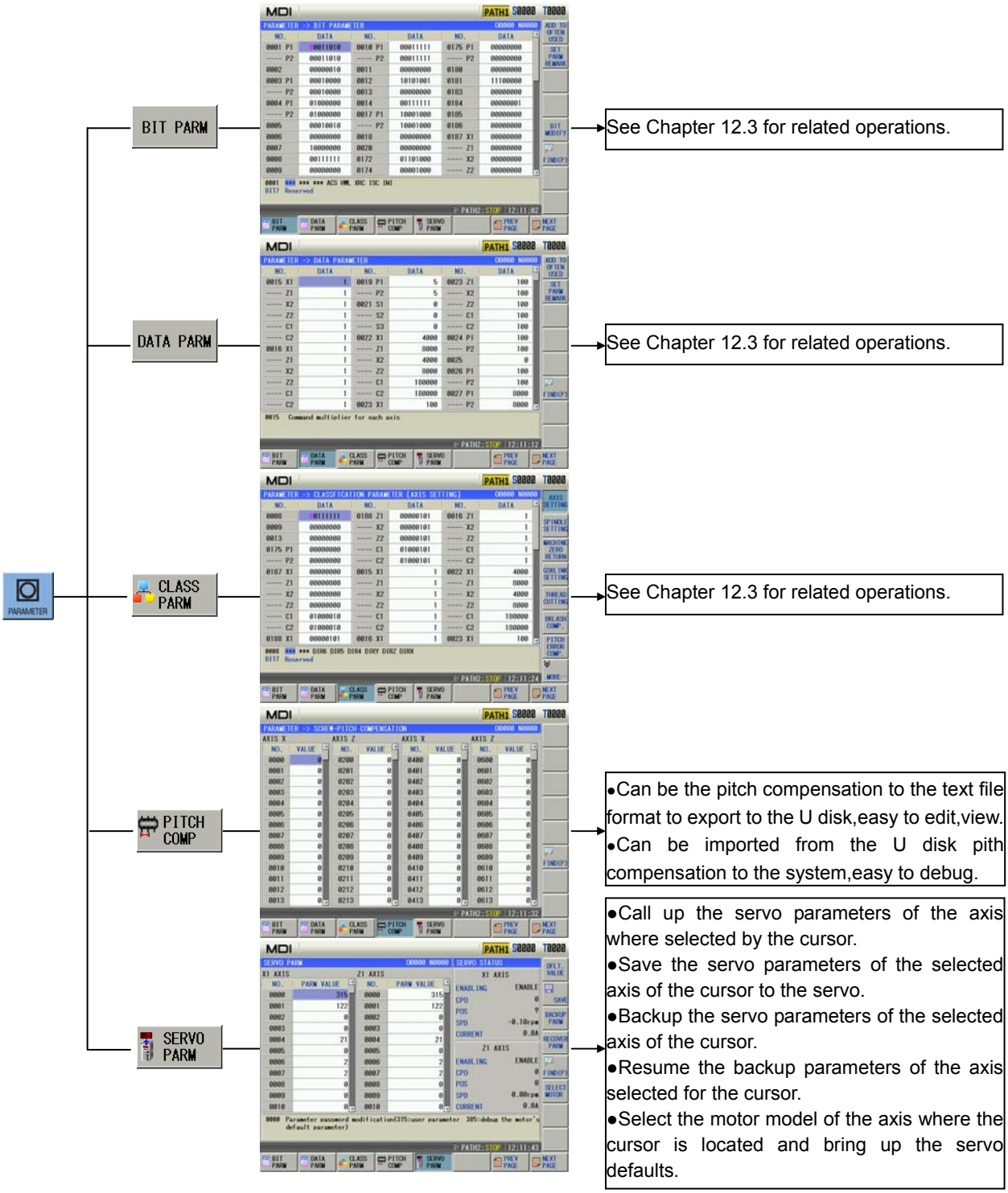
- Press the soft key [SWITCH] (or[CHANGE] key) to select the CNC catalog or the U disk catalog, move the cursor to the required option.
- Move the cursor to the target file (system parameters,tool compensation,pitch compensation),PLC programs, part program or file list，Press [SELECT] soft key (or [DATA INPUT] key)to select.
- Press [COPY TO CNC](or [COPY TO U-DIST]) soft key to copy file to target directory.

- According to function classification,debugging directly face the function options,by selecting the automatic completion of parameter settings.
- After entering the debug option,modify the options.
- In the debugging page,can be signal diagnosis.

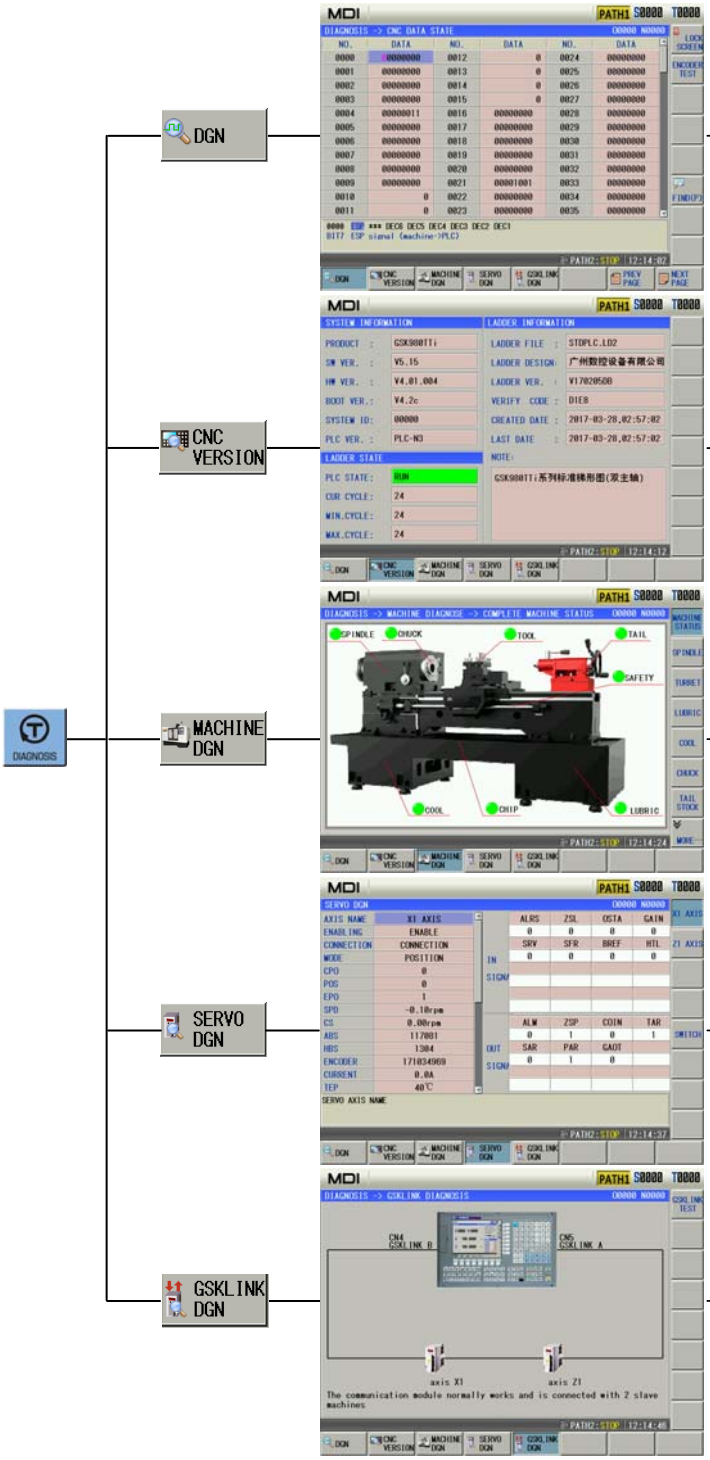
- When the connection sequence of the servo drive is: X axis, Z axis, Y axis, the 4th axis, the 5th axis, spindle 1, spindle 2 or spindle 3 (regardless of the unavailable axis), the overall servo drives can be set by [GHOST SETTING] soft key.
- If the connection sequence of the servo axis does not correct，The corresponding setting parameter of the servo is PA156：
X axis:1 Z axis:2 Y axis:3 4th axis:4 5th axis:5 Spindle 1:11 Spindle 2:12 Spindle 3:13

Move the cursor in the address bar to switch,directly enter the value you want to change.

2.6 PARAMETER Interfaces



2.7 DIAGNOSIS Interface



●Lock the current page for keyboard keypad troubleshooting.
●Spindle encoder line test,easy to machine debugging and maintenance.

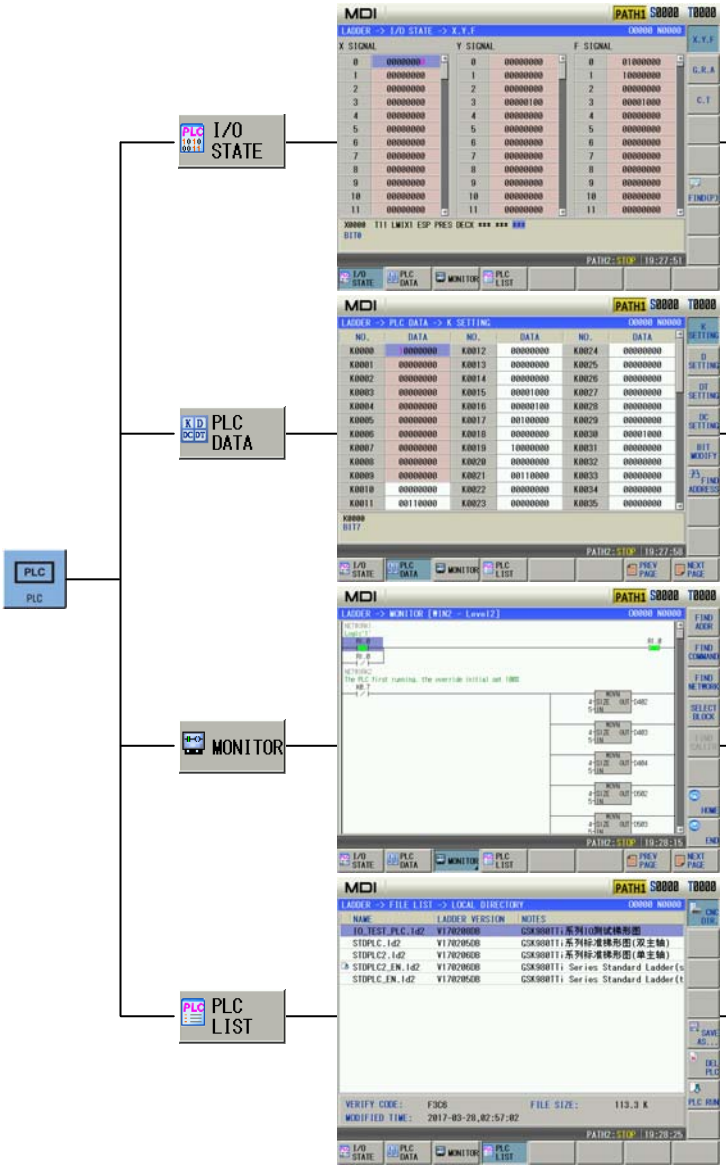
Search the product information, ladder information and ladder state.

Machine diagnosis can be directly inspected the working state of machine tool's each component

Servo diagnosis can be viewed servo operation state of each axis.

GSKLINK diagnosis page allows you to view the connection status of the bus and view the device that generated the ring break.

2.8 LADDER Interface



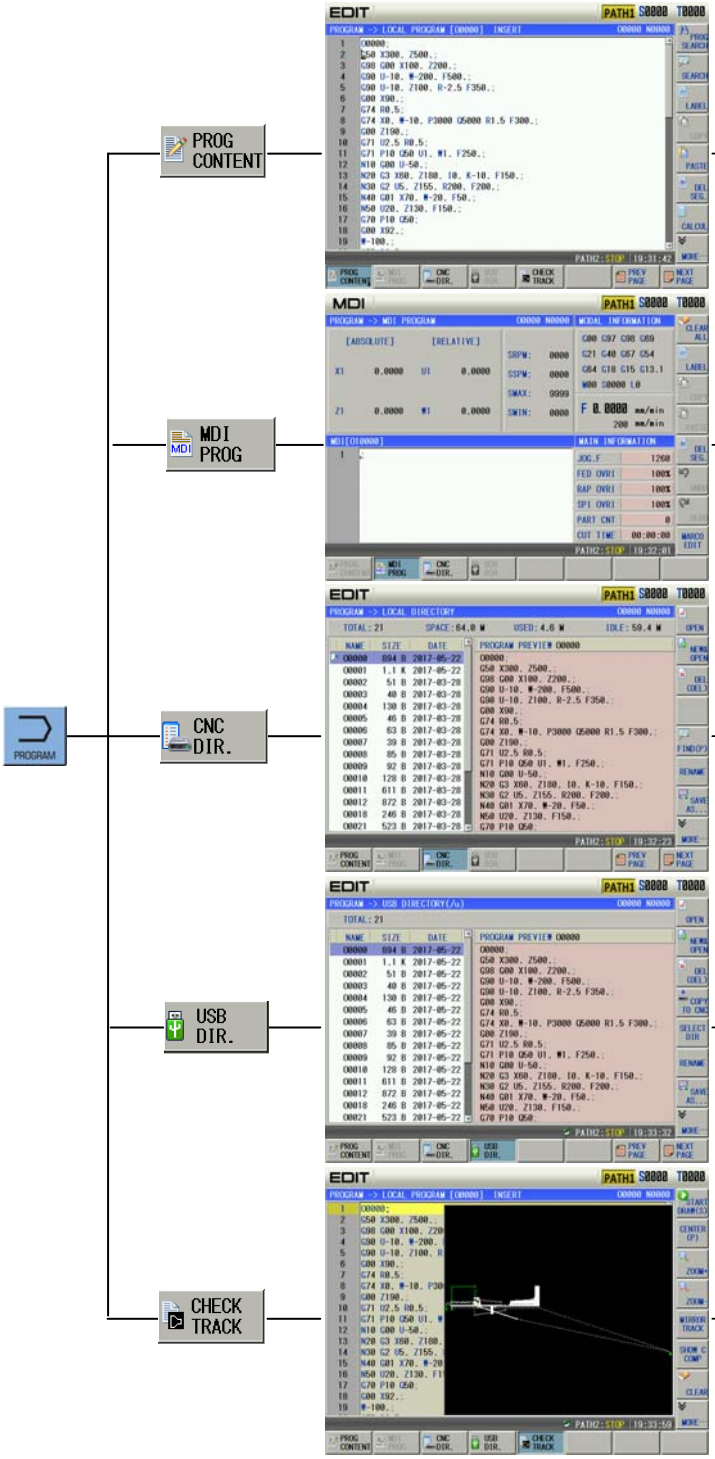
PLC state page can search states of X, Y, F, G, R, A, C,T;

PLC data page can search/set K, D, DT, DC value.

●Search parameters, commands and network
●Select subroutine.
●locate the cursor to the home or end of the current block.

Press [CNC DIR.](or [USB DIR.]) softe key to enter directory page， Press the cursor to select the PLC file to be copied，In the U-disk directory,Press [SELECT PATH] soft key to select the directory where the PLC file is located.

2.9 PRG interface



See Chapter 9 for related operations： EDIT OPERATION

See Chapter 7 for related operations： MDI OPERATION

See Chapter 9 for related operations： EDIT OPERATION

In the U-disk directory page or local directory page,you can copy the machining program to the local directory or U-disk directory.

●Display the programming path.
●When the path exceeds its range, the diagram can be automatically displayed at the middle of the screen.
●The diagram path can be zoomed in $\sqrt{2}$ times by the ZOOM+.
●The diagram path can be zoomed out $1/\sqrt{2}$ times by ZOOM-.
●The programming path and symmetric line of the programming path (cutting command) will display by MIRROR TACK.
●The programming path and the cutter compensation C path will display.
●Clear the current graphic path.

CHAPTER 3 POWER ON/OFF AND PROTECTION

3.1 System Power-on

Before GSK980TTi power on, the following items should be confirmed:

1. The machine is in a normal state.
2. The power voltage conforms to the requirement of the machine.
3. The connection is correct and secure.

3.2 System Power-off

Before power is off, ensure that:

1. The feed axes of the CNC is at stop;
2. Miscellaneous functions (spindle, cooling etc.) are OFF;
3. Cut off CNC power prior to machine power cutting off.

Note:Please refer to the machine manufacturer's manual about turn-off the machine's power supply.

3.3 Emergency Operation

This section mainly describes the resolutions that the system is capable of under the emergency situation. Please see the relative explanation on these resolutions under the emergency by machine manufacturer.

3.3.1 Reset

Press  key to reset GSK980TTi system when there are abnormal output and axis operations:

1 All axes motion stops;

2 M, S function output is inactive (the parameter sets whether the system automatically cuts off signals of spindle rotation, lubricating, cooling by pressing  key, defined by PLC ladder);

3 Automatic run ends, modal function and state remain.

3.3.2 Emergency stop

During machine running, if the emergency button is pressed under the dangerous or emergent situation (external SP signal active), the CNC system enters into emergency status and the machine movement is stopped immediately. All the outputs such as the spindle running, cooling are cut off. If the emergency button is released, the emergency alarm is cancelled and the CNC resets.

Note 1:The ESP alarm generates during the operation of the machine tool, Re-perform the machine zero return to get the correct position coordinate after the emergency alarm is cancelled (machine zero return is forbidden if there is no machine zero on the machine.).

Note 2:Only Bit3 of the bit parameter No.172 is set to 0, is the external emergency stop active.

Note 3:Press the emergency stop button before powering on and off to reduce the electrical shock of the devce.

3.3.3 Feed hold

Press 

key during the machine running to make the running pause. However, in threading cutting, tapping cycle state, the function cannot stop the running immediately.

3.3.4 Power-off

Under the dangerous or emergency situations during the machine running, the machine power should be cut off immediately to avoid the accidents. But it should be noted that there may be a large error between the CNC coordinates displayed and the actual position. So the tool-setting operation should be performed again.

CHAPTER 4 MANUAL OPERATION

Press  key, it enters Manual mode. In the mode, the system can perform the manual feed, spindle control, override adjustment, tool change etc.

4.1 Coordinate Axis Move

In Manual mode, 2 coordinate axes can perform manual feeding and rapid traverse.

4.1.1 Manual feed

Press  、  、  or  and X or Z feeds negatively or positively, and its feeding stops if the key is released; In Manual mode, press  key to make the indicator enter the manual rapid traverse mode.

4.1.2 Manual rapid traverse

Press  the key indicator is ON，press  、  、  or  and X feeds negatively or positively, and its feeding stops if the key is released;。The rapid override time-tuning is enabled.

In Manual mode, press  key to make the indicator OFF, and the rapid traverse is disabled and manual feed is executed.

Note 1:If no reference point return is performed after power on, as the rapid traverse switch is turned on (rapid indicator ON), the manual feedrate or rapid rate for the traverse is defined by the Bit0 of the bit parameter No.012 of this GSK980TTi system.

Note 2: In Edit/MPG mode,  key is inactive.

4.1.3 Speed tune

In Manual mode, press  、  、  to alter the manual feed override which is divided into 16 steps. The relation of the override and the feedrate is as follows table when data parameter No.031 is set to 1260:

Feedrate override(%)	Feedrate(mm/min)	Feedrate override(%)	Feedrate(mm/min)
0	0	80	1008
10	126	90	1134
20	252	100	1260
30	378	110	1386
40	504	120	1512
50	630	130	1638
60	756	140	1764
70	882	150	1890

In the manual rapid traverse, it can press 

key to alter the rapid override, and there are 4 steps of F0, 25%, 50%,100% for the override.(F0 is set by data parameter No.032)

The rapid override is active under the following conditions:

(1) G00 rapid traverse (2) Rapid traverse in canned cycle

(3) Rapid traverse in G28 (4) Manual rapid traverse

4.2 Spindle jog

: At the moment, the spindle is in JOG state.

Functional description:

Press 

to enter JOG mode, and the spindle JOG function ON/OFF is executed only when the spindle is in the state of stop.

In spindle JOG mode, by pressing 

key, the spindle rotates counterclockwise for jogging; by pressing 

key, the spindle rotates clockwise for jogging. The jog time and speed are set by data parameter No.108 and No.109 respectively.

When the spindle JOG rotates, 

is pressed to stop the spindle JOG rotation, the spindle brake signal is not output when the JOG rotation stops.

K10.4 is set to 1, the spindle JOG is valid in any mode. In Auto or MDI mode, the spindle is in the JOG rotation state, the program closes the spindle JOG rotation and the JOG function.

Parameter setting:

PLC parameter K104 1/0: the spindle JOG is valid in any mode/Manual, MPG, Zero return mode.

Data parameter No.108: spindle JOG time.

Data parameter No.109: rotary speed in spindle JOG.

Note:In MPG/ Zero return mode can execute spindle jog operation.

4.3 Cooling control

: In Any mode, press this key, the cooling is switched on/off.

Parameter setting:PLC parameter K10.1 1/0: the spindle lubricating and cooling output remains/closes in reset.

Note:In Any mode can execute cooling control.

4.4 Lubricating control

Function description:

1. Non-automatic lubricating (DT017 =0)

When PLC DT013=0, it is the lubricating turn output. Pressing the 

can execute the output. And the lubricating is cancelled by pressing it again. M32 is to execute lubricating output, and M33 is to cancel lubricating output.

When PLC DT013 ＞1, it is timing lubricating output. Pressing the 

can execute the output. And it is cancelled after a setting time by data parameter DT013; M32 is to execute lubricating output and the output is cancelled after a time set by DT013. If the setting time is not yet up, M33 is executed to cancel the lubricating output.

2. Automatic lubricating (DT017>0)

For automatic lubricating, the system executes the lubricating in the time set by DT017 and the stops the output. After lubricating interval time set by DT016, the output is done again, it does repetitively like this. During the automatic lubricating, M32, M33 codes as well as the 

key are all active.

Parameter setting:

PLC parameter: K10.1 1/0: the spindle lubricating/cooling output remains/closes in reset;

PLC parameter:K16.2 1/0: whether the lubricating outputs in power-on when the automatic lubricating is valid;

PLC data DT005: M execution duration(ms);

PLC data DT013: lubricating start time (0-60000ms)(0:lubricating time is not limited);

PLC data: DT016 automatic lubricating interval time (ms);

PLC data: DT017: automatic lubricating output time (ms).

Note:In Any mode can execute lubricating control.

4.4 Chuck control

: press it to switch the chuck releasing/clamping in any mode.

Function description:

When the chuck control is invalid, the system alarms in executing the chuck control M command;

When the system checks the chuck clamping (K12.1=1), the spindle cannot be started when the chuck is not clamped, and the chuck cannot be released after the spindle is started;

When using the input signal controls the chuck operation, the spindle should stop and delay the time set by DT21, the control is valid.

Parameter setting:

PLC parameter: K12.0 1/0: chuck control valid/invalid;

PLC parameter: K12.1 1/0: do not check/check the chuck clamping before the spindle start;

PLC parameter: K12.2 1/0: chuck outer/inner control mode;

PLC parameter: K12.3 1/0: check/not chuck in-position signal;

PLC data: DT018: chuck pulse output width(ms);

PLC data: DT021: spindle stops, chuck operation enabling delays(ms).

Note:In Any mode can execute chuck control.

4.5 Tailstock control

: In any mode, press it to switch the tailstock forward/backward.

Function description:

When the tailstock control is invalid, the tailstock control M command alarms;

When the tailstock and the spindle control is interlocked, before the spindle is started, the system check whether the tailstock forward is valid. After the spindle is started, the tailstock backward cannot be executed.

Parameter setting:

PLC parameter:K13.0 1/0：tailstock control valid/invalid

PLC parameter:K13.1 1/0：the spindle rotation and the tailstock forward/backward are/not interlocked

PLC parameter K13.2: 1/0: External tailstock control is enabled/disabled when the program is being operated.

Note:In Any mode can execute tailstock control.Only effective in path 1.

4.6 Hydraulic control

: press it to switch the hydraulic motor ON/OFF in any mode.

Function description:

When the hydraulic control function is valid, 

is pressed to control HPST output to start the hydraulic motor.

When the system is in non-run, the spindle stops and the speed is zero, 

is pressed to close HPST output.

Parameter setting:

PLC parameter: K14.7 1/0: hydraulic control function valid/invalid.

Note:In Any mode can execute hydraulic control.

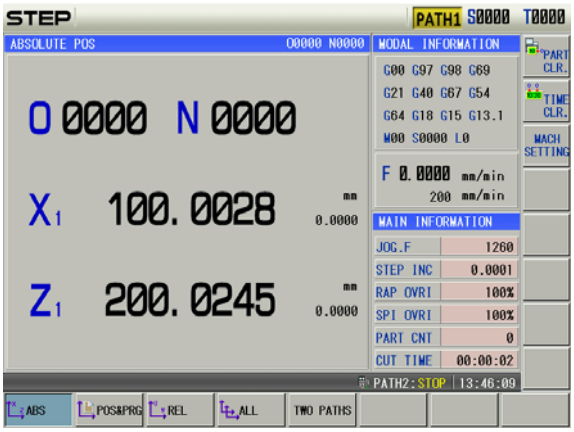
CHAPTER 5 MPG/STEP OPERATION

5.1 Step Feed

Set the system parameter No.001 Bit3 to 0, and press  key to enter the STEP working mode.

5.1.1 Increment selection

Press  to select the move increment, the increment will be shown in the page. When the BIT7(SINC) of K016 is 1,  step width value is inactive; when the BIT7 is 0,  are all active. For example, to press  key, the window is shown as follows:



5.1.2 Moving direction selection

Pressing  or  key once can move X negatively or positively by a step increment; pressing  or  key once can move Z negatively or positively by a step increment; pressing  or  key once can move Y negatively or positively by a step increment.

5.2 MPG (handwheel) Feed

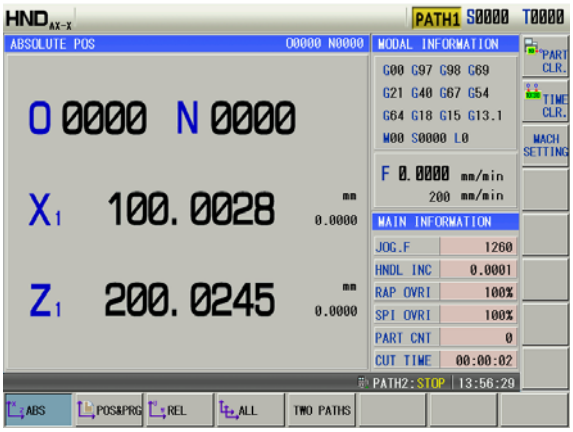
Set the BIT3 of the system parameter No.001 to 1, and press  key to enter the MPG mode.

5.2.1 Increment selection

Be the same as of step mode.

5.2.2 Moving axis and direction selection

In MPG mode, press  、  、  、  、  key, the corresponding axis will be selected. For example, press  key and the window is shown below:



The MPG feed direction is defined by its rotation direction. Generally, the MPG CW is for positive feed, and CCW for negative feed. In case of that MPG CW is for negative feed, CCW for positive feed, it may exchange the A, B signals of the MPG terminals. Bit0～Bit4 of NO. 013 selects the feed direction of MPG rotation.

5.2.3 MPG feedrate

The upper limit of the MPG feedrate is the rapid traverse rate of each axis, which can be set by data parameter No. 022.

When the state parameter No.013.6 = 0, and the MPG feedrate exceeds the rapid traverse rate, the exceeded pulse is ignored; the MPG scale is inconsistent with movement amount。

When the state parameter No.013.6 = 1, and the MPG feedrate exceeds the rapid traverse rate, the exceeded pulse does not ignore; the MPG scale is consistent with movement amount, but the axis may consecutively move when MPG stops.

5.2.4 Other operations

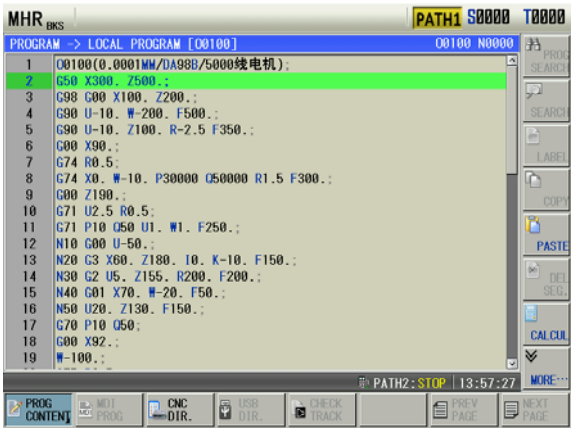
IN MPG／step mode can execute spindle operation、lubricating control、cooling control、chuck control、tailstock control、hydraulic control、manual tool change、spindle override adjusted etc.operations.

5.3 MPG Trial-cut

The user can use the MPG trial-cut function after after editing part programs,check the run path of the program. In the MPG trial-cut mode,you can control the execution speed of the program by turning the MPG,the program can execute swquentially or roll back the executed program by CW/CCW turning the MPG,this can easily check the program error conveniently.

5.3.1 Switching MPG trial-cut mode

Press  to enter the MPG trial-cut mode after a machinig program is selected。At the moment, press  and the display is shown below:



At the moment, CW rotating MPG makes the program start running swquentially; CW rotating MPG can roll back the executed program.When the execution speed of program is proportional to MPG speed, execution speed of program fastens as soon as the MPG rapidly rotates; the execution speed slows down as soon as the MPG slowly rotates. Movement amount of one pulse can be adjusted by rapid override.

When the system is in the MPG trial-cut mode, it returns to Auto mode after  is pressed. All operations in MPG trial-cut mode are the same those of Auto mode.

5.3.2 Notes in MPG trial-cut mode

Pay more attention to the followings when the system is in MPG trial-cut mode:

1) Press  in Auto mode and the CNC does not switch to MPG trial-cut mode; when the CNC is in MPG trial-cut mode, it escapes the MPG trial-cut mode from the next block by pressing the mode switch key;
2) When the system executes the MPG trial-cut control, the single block signal and the feed hold signal are valid. When the single block or feed hold stops, the program execution state is recovered to the MPG trial-cut control after the cycle start key is pressed;
3) Execution speed of blocks for movement and pause can be controlled by rotating MPG. Their speed of blocks with M, S, T, F, i.e. without movement or pause cannot be controlled(except for executing the tool offset in traverse mode) by rotating MPG which only controls whether they are executed;
4) The spindle speed is not related to the MPG pulse. The spindle rotates with the commanded speed even if the system is in the MPG trial-cut mode. For feed per rotation, reading the current spindle speed is switched to the execution after feed per minute.

5.3.3 Temporarily invalid in MPG trial-cut mode

1) Executing screw cutting commands

In the course of executing screw cutting blocks (G32, G32.1, G33, G34, G76, G84, G88, G92，G92.1), the MPG trial-cut function is temporarily invalid with the speed under the override 100%, and is valid in the next block again. The MPG control is invalid when the screw cutting is executed actually, and it is valid in others.

2) Executing from the middle point to machine zero

In the course of executing the block (G28, G30) from the middle point to machine, the MPG trial cutting function is enabled from the start to intermediate point, and the MPG trial cutting is temporarily disabled from intermediate point to mechanical zero. The block is executed with the speed under the override 100%, and the next block is valid again.

3) Measuring related G commands execution with the override 100%

When the system measures the related G commands, the MPG trial-cut is invalid temporarily, is executed with the speed under the override 100%, at the moment, execution of the next block is valid again. The related G commands include:

G31, G36, G37

5.4 MPG interrupted function

In the Auto (Auto, MDI) and Edit mode, In the Auto (Auto, MDI) and Edit mode, the movement value from MPG can be overlapped to the Auto movement by rotating the MPG. The axis performed the MPG interruption can be selected by the MPG interruption axis selection signal. The movement distance of the MPG interruption is determined by the rotation value of the MPG and hand-wheel federate.

Note:It is necessary to set the Bit 1 of parameter No.0012 to "1" when using the MPG interruption function, it is necessary to support by dedicated PLC program.

5.4.1 MPG Interrupted Operation

In the Auto, MDI and Edit mode, enter to the MPG interruption state to perform the MPG based upon the MPG axle number on the selection panel after controlling the [MPG interruption switch] (External switch) once; retreat from the MPG interruption by pressing the [MPG interruption switch] again.

The equivalent is identical between the MPG interruption and MPG feed.The velocity during the MPG interruption is the sum both in the Auto operation and MPG interruption movement, and its overlapping speed is controlled within the upper limit speed of the cutting feed. For example, cutting upper limit is 8000mm/min, if the current feedrate is 2000mm/min, the MPG interruption speed range is -10000~6000mm/min ("-" means that the MPG interruption movement direction is reverse to the feed one).

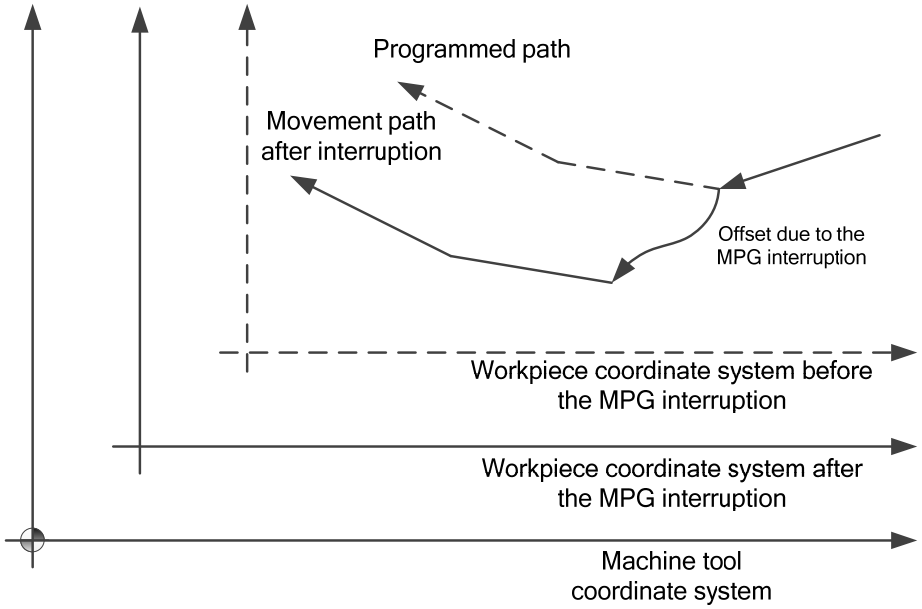
Related parameter

№012.1	MPG interruption function 0: Disabled 1: Enabled
K0019.5	MPG interruption operation 0: Disabled 1: Enabled

5.4.2 Affection of Coordinate System from MPG Interruption

Axis moves by MPG interruption, and the actual position of the movement axis changes, the machine coordinate is renewed instead of the absolute coordinate. Therefore, machine moves after the MPG interrupts, and the machine coordinate system holds, but the workpiece coordinate system will offset.

- Absolute coordinate → Absolute coordinate value will not change due to the MPG interruption.
- Relative coordinate → Due to the MPG interruption, Interrupted amount change.
- Mechanical coordinate → Due to the MPG interruption, interrupted amount change.



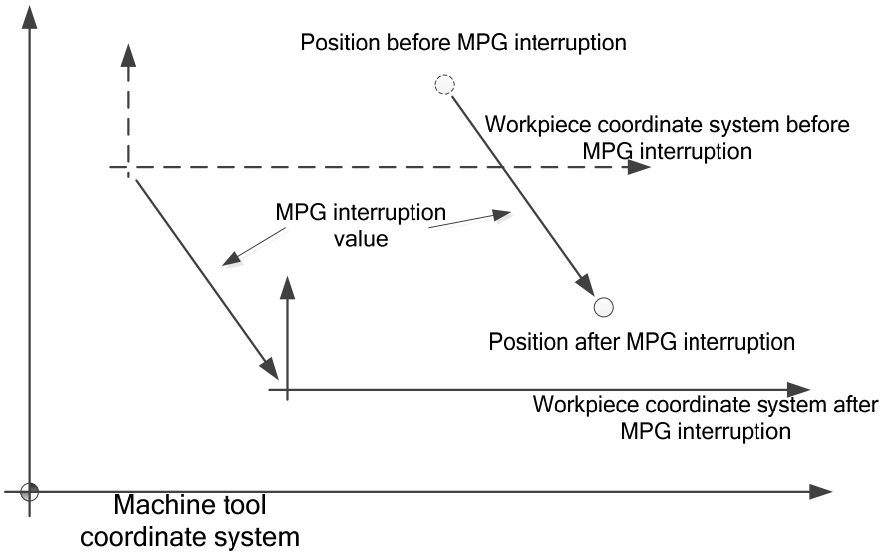
When automatically returning the reference point (G28), although the end (reference point) does not affect by MPG interruption, the intermediate point may generate the offset of interruption value by the MPG interruption at the workpiece coordinate system.

5.4.3 MPG Interruption Value Cancel

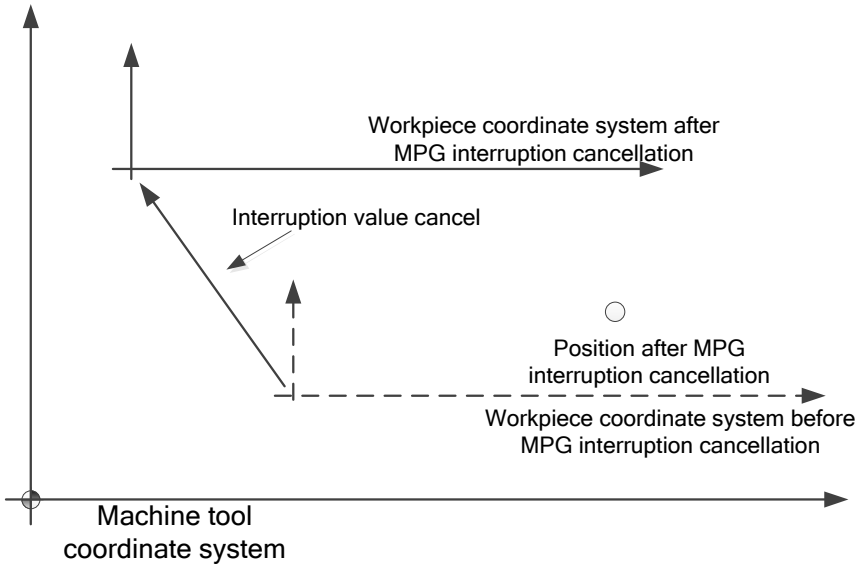
The cancellation of the MPG interruption value can be returned the workpiece coordinate system after offset to the one before offset.

When the cancellation operation of the interruption value is performed, the MPG interruption value reflects to the absolute coordinate value.

The workpiece coordinate system offset generated from MPG interruption is shown below:



The workpiece coordinate system recovers to the state before interruption after the MPG interruption is cancelled, however, the actual position is invariable, refer to the following figure:



The interruption value can be cancelled base upon the following situations:

- Reset, EMS (Set whether it is cancelled by parameter).
- Perform G28 or Manual return reference point operation. (When the absolute coordinate system is set after returning to the reference position, the interruption value will be directly eliminated regardless of the new-created absolute coordinate system. When the absolute coordinate system does not set after returning to the reference position, the workpiece system will recover to the previous state of the interruption, simultaneously, the interruption value is then eliminated).
- When the non-block reference point set is performed. (The treatment is consistent with the manual reference point return).
- When performing the G50 workpiece coordinate system setting. (In this case, the workpiece coordinate is one by G50, simultaneously, the interruption value is eliminated)

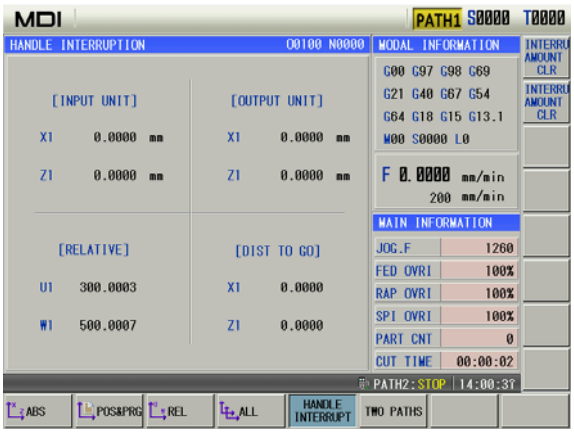
1) System restarts. The MPG interruption value will be also cleared after the system is power off, but the workpiece coordinate system holds, and therefore, the system is turned off after

the MPG interrupt, the workpiece coordinate offset generated in the MPG interruption will always enabled, and it can not recover to the previous state of interruption.).

5.4.4 MPG Interruption Display Page

The Bit 1 of state parameter No.0012 should be set to “1” before using the MPG interruption function, and then lead in the dedicated PLC program;

Press the  soft function button to enter its corresponding page after controlling the HANDLE INTERRUPT button, refer to the following figure:



The following 4 data can be simultaneously displayed at the MPG interruption display page:

1) Input unit:

The MPG interruption movement value in the input unit means the specified movement value of MPG interruption based upon its least input command unit.

2) Output unit:

The MPG interruption movement value in the output unit means the specified movement value of MPG interruption based upon its least output movement unit.

3) Relative coordinate:

The position at the relative coordinate system and the position of the current relative coordinate system may generate the change of the interruption value due to the interruption of the MPG.

4) Residual movement value:

The residual movement value at the current block is irrelevant to the MPG interruption.

The MPG interruption movement value is cleared when the manual reference point end return is performed along each axis.

Note 1: In the MPG interruption page, the letters of the coordinate axis name should be marked by in the MPG interruption axis.

Note 2:Manually clear the MPG interruption value by the soft button of the MPG interruption page; the workpiece coordinate system will not renew; that is, the coordinate system position before the MPG interruption will not recover, the workpiece coordinate offset generated in the MPG interruption will always enabled, and then can not be recover again.

5.4.5 Precaution of MPG Interruption

1) MPG interruption is disabled when the MPG feed (Including the Manual, MPG, Single-step, Machine tool zero return or program zero return) is performed;
2) The axis is controlled by PLC axis; MPG interruption is disabled.
3) MPG interruption is disabled when MPG trial-cutting is performed.
4) The axis does not move when the machine is locked, and the MPG interruption is disabled.
5) MPG interruption is disabled when returning the reference position.
6) MPG interruption is disabled when the system alarm occurs.

Note:The MPG interruption disabling are on the states 1) ~ 3), as the above-mentioned, the system will not enter to the MPG interruption state. If it on the states 4) ~ 6), the system can be entered to the MPG interruption state, instead of generating the enabled MPG interruption value.

CHAPTER 6 ZERO RETURN OPERATION

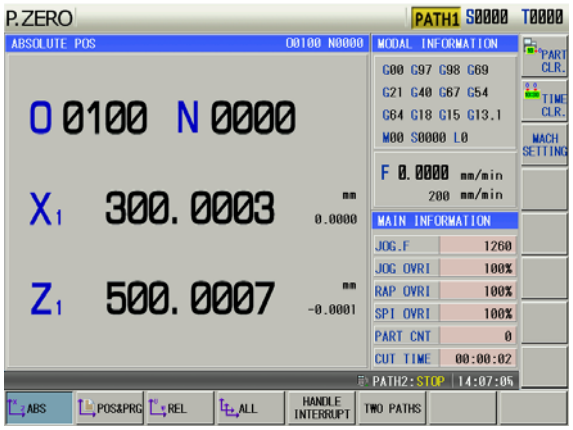
6.1 Program Zero Return

6.1.1 Program Zero

While the part is fixed on the machine, absolute coordinate of current tool position may be set by G50 code according to the relative position between the tool and the part, so a workpiece coordinate system is setup. The tool current position is called **program zero**, and this is the program zero return position.

6.1.2 Program zero return steps

1.Press  key, it enters the Program zero return mode, the bottom line of the window displays “P. ZERO”:



2.Press the direction key of X, Z , Y, 4th, C axis, it returns to the program zero of X, Z , Y, 4th or C axis;

3.The machine axis moves toward the program zero return, and the axis stops with the program zero return completion indicator ON after the axis returns to the program zero.

Note 1:The tool offset is not changed for the program zero return operation, if there is offset, the return position is the point set by G50.

Note 2:Whether the key is held on at program zero return is defined by the bit parameter No.011 BIT2 (zero return is locked automatically).

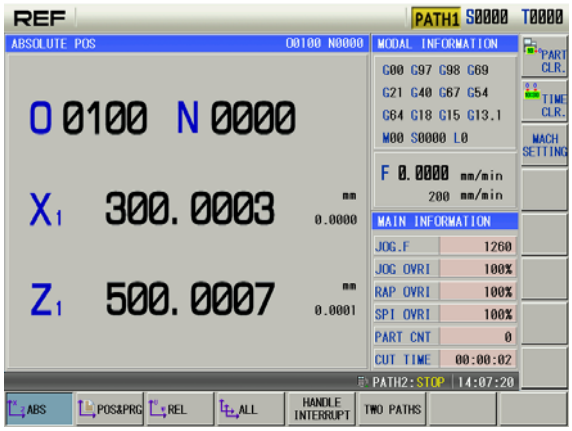
6.2 Machine Zero Return

6.2.1 Machine Zero

The **machine coordinate system** is a reference coordinate system for CNC coordinate operation. It is an inherent coordinate system of the machine. The origin of the machine coordinate system is called machine zero (or machine reference point). It is defined by the zero or zero return switch fixed on the machine. Usually this switch is fixed at the positive stroke point of each axis.

6.2.2 Machine Zero return steps

1) Press  key, it enters the Machine zero mode, the bottom line of the window displays "REF":



2) Press  ,  ,  ,  or  key to return to the machine zero of X, Z, Y, 4th or C axis;

3) Carrier moves along with the machine tool zero; when it matches with the incremental encoder motor, return to the machine tool zero after passing the deceleration signal or zero signal detection. When it matches with the absolute encoder motor, directly position the set mechanical zero based upon the velocity of the mechanical zero turn; in this case, the axis stops, and the zero return indicator is ON.

Note 1:If there is no machine zero on the machine, the machine zero operation is forbidden;

Note 2:The machine zero finish indicator is gone out on condition that: 1)The axis is moved out from machine zero; 2) CNC is powered off;

Note 3:After the machine zero operation, the tool length compensation is cancelled by CNC;

Note 4:Parameters related to machine zero return are referred to INSTALLATION and CONNECTION;

Note 5:After the machine zero return is executed, the original workpiece coordinate system is set again with G50.

CHAPTER 7 MDI OPERATION

In MDI mode, the operations of parameter setting, code words input and execution can be performed.

7.1 Block Input

Select MDI mode to enter PROGRAM->MDI page, to input an block G50 X50 Z100, the steps are shown below:

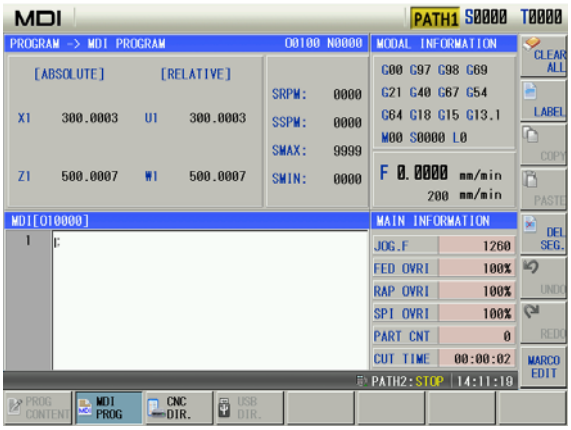
1)Press 

key to enter MDI mode;

2)Press 

, and then press 

to enter MDI program page below:



Note: Up to 10-block programs can be input of the MDI program。

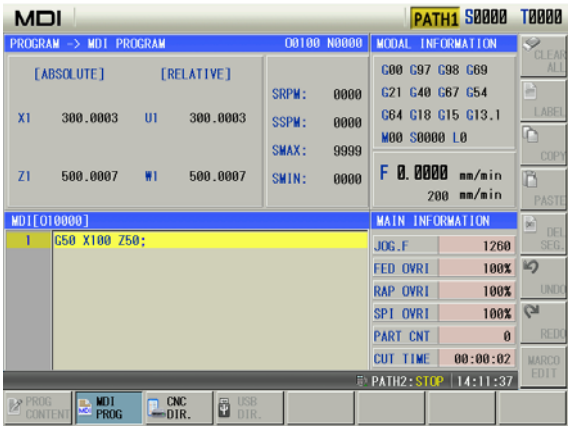
7.2 Block Execution

After the code words are input and 

is pressed to display the block, 

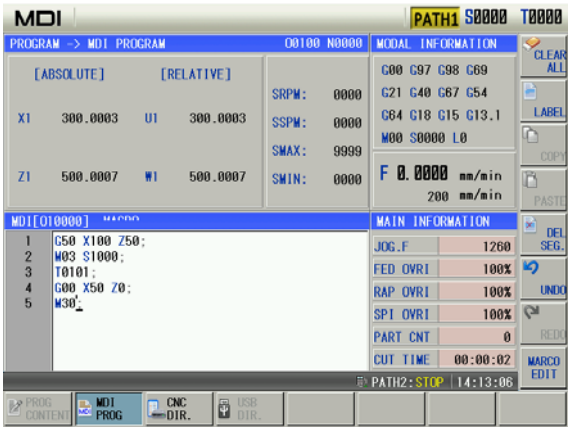
key is pressed to execute the input block. During the execution, 

, 
and Emergency Stop button may be pressed to terminate these code words execution. the display is as follows:



7.3 Data Alteration

There are CLEAR ALL, LABEL, COPY, PASTE, DEL SEG., UNDO, REDO and MARCO EDIT program edit operation items on the MDI program page.



CLEAR ALL soft button: Clear the overall contents of the MDI program input frame.

CHAPTER 8 AUTO OPERATION

8.1 Automatic Run

1) Set coordinate system G54~G59,select coordinate system in the program,when no select，default is G54;
2) Set tool offset and tool wear, select tool offset and tool wear in the program;
3) In edit mode,open the selected block.in prg content page, move the cursor to the block to be executed,refer to the following fig.1;
4) Press  key to enter the Auto mode, refer to the following fig.2;

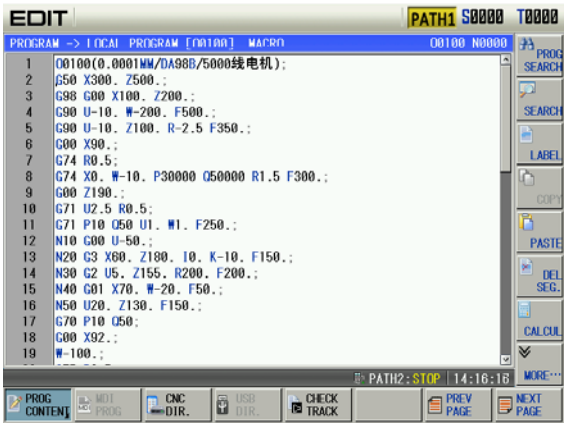


Fig 1

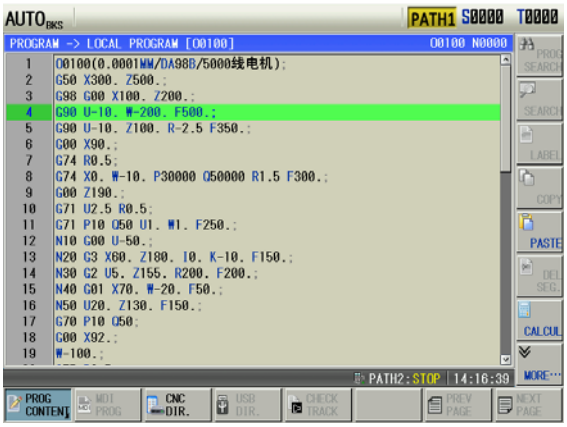


Fig 2

5) Press  to start the program, and the program automatically runs;
6) Necessity,by pressing  or external pause key pause program,The program execution continues after pressing the  key.;press  ,the automatic run end；After pressing emergency button, the CNC system enters into emergency state, and the machine moving is stopped immediately, all the output (such as spindle rotation, cooling) are all cut off. If the Emergency button is released, the alarm is cancelled and CNC system enters into reset mode.

Note:When Auto mode is switched to the Machine zero, MPG/Step, Manual, Program zero mode, the current block “dwells” immediately; when the Auto mode is switched to the Edit, MDI mode in Auto mode, the “dwell” is not displayed till the current block is executed.

8.2 Running State

8.2.1 Single Block Execution

When a program is executed firstly, the system selects the single block run mode to avoid the program error to cause the unexpected.

Press  to select the single block run function in Auto mode; in single block mode, after the current block is executed, the CNC stops run; press  to execute the next block. Such repetition is executed till the program is done.

Note 1:The single block stops at the mid point of G28 code.

Note 2:For the single block state in the execution of canned cycle codes G90, G92, G94, G70～G76, refer to the 1st part PROGRAMMING.

Note 3:While the subprogram calling (M98__), or subprogram calling return (M99)is being executed, the single block is inactive. But it is active except for N, O, P addresses in the block that contains M98 or M99 code.

8.2.2 Dry Run

Before the program is to be executed automatically, in order to avoid the programming errors, it may select the Dry run mode to check the program.

In Auto mode, press  to enter the dry run state; in Dry run state, the machine feed and miscellaneous functions are both active (as machine lock, MST lock are both OFF), that means the dry run switch has nothing to do with the machine feeding, MST functions, so the feedrate by program is inactive and the CNC system runs at the speed described in the following table:

	Program command	
	Rapid traverse	Cutting feed
Rapid traverse switch ON	Rapid traverse	Max. manual feedrate
Rapid traverse switch OFF	Manual feedrate or rapid traverse(see note)	Manual feedrate

Note 1:The rate by manual feedrate or rapid rate is set by the BIT6 of the CNC system parameter No.004.

Note 2:The shift of rapid switch in Dry run mode doesn't affect the rate of the current block being executed, but that of the next block.

Note 3:The switch operation of Dry run is inactive if the ladder of this GSK980TTi is defined to be in auto running state (Auto, MDI mode).

8.2.3 Machine Lock

In Auto mode, press  to enter the machine lock; the machine lock and MST lock are usually used together to check the program. While the machine is in the lock run state:

1.The machine carriage doesn't move, the "MACHINE" in the INTEGRATED POS window of the Position interface doesn't vary too. The RELATIVE POS and ABSOLUTE POS, DIST TO GO

are refreshed continuously, which is the same as that the machine lock switch is OFF.

2. M, S, T commands can be executed normally.

3. Press the  on the system operation panel to close the machine lock; the system is then automatically recovered the absolute coordinate value along each axis; that is, the coordinate system automatically restores.

Precautions:

1) The machine lock open/close system automatically restores the absolute/relative coordinate along each axis;
2) The tool state is regardless of the automatic recovery of the workpiece coordinate system before/after the machine lock. For example, the tool state before the machine lock is T0101; after is T0202;
3) The state of the cutter compensation C is regardless of the automatic recovery of the workpiece coordinate system before/after the machine lock. For example, the state of the cutter compensation C before the machine lock is G40; after is G42;
4) The system ON-OFF state is regardless of the automatic recovery of the workpiece coordinate before/after the machine lock. For example, the machine is locked before the power-off; the system can be automatically restored the workpiece coordinate system after the power is turned on again.
5) To avoid the tool interval motion during the automatic operation, it is better not to change the state of the machine tool after starting the program.

8.2.4 MST Lock

In Auto mode, press 

to enter the miscellaneous function lock state; The machine carriage moves without the M, S, T code being executed. The machine lock and MST lock are usually used together to check the program.

Note: When the MST lock is active, it takes no effect to the execution of M00, M29, M30, M98, M99.

8.2.5 Block Skip

When a block in program is not needed to be executed and not to be deleted, this block skip function can be used. When the block is headed with "/ " sign and Block skip indicator lights up (panel key active or external skip input active), this block is skipped without execution in Auto mode.

In Auto mode, press  to enter the state that the program skip is valid.

Note:While the block skip switch is off, the blocks headed with "/" signs are executed normally in Auto mode.

CHAPTER 9 EDIT OPERATION

In Edit mode, a program can be created, selected, altered, copied and deleted, and the bidirectional communication of CNC to CNC, or CNC to PC can also be done.

To prevent the program to be altered or deleted accidentally, a program switch is set up for this GSK980TTi system. And it must be turned on before program editing. Please see details in Section 12.1 of the part.

The system provides multi-level user permission to facilitate the management. More than 4 levels (4 level, 3 level) can open the program switch to edit programs. See **OPERATION, Section** 12.2 **.**

9.1 Program Creation

9.1.1 Creating a block number

Press SETTING , CNC SETTING and PROG SWITCH, it automatically generates the next block number by pressing EOB or DATA INPUT in editing. The block number increment is set by the CNC data parameter No.042. (See details in **Section** 12.1 of this part.) .

9.1.2 Program Creation

Press EDIT to enter the Edit mode. Press PROGRAM to enter the Program interface, create a machining program before inputting a machining program. The creating method is shown below:

1) Pres CNC DIR. to enter the local directory page .Press NEW& OPEN, input orderly O, O, O, 1 in the pop-up dialog box(taking example of creating O0001, the leading zero can be omitted when it is input);

2) Press EOB (or DATA INPUT) to create the new program. The current page is automatically switched into the program content page.,refer to the following Fig 2;

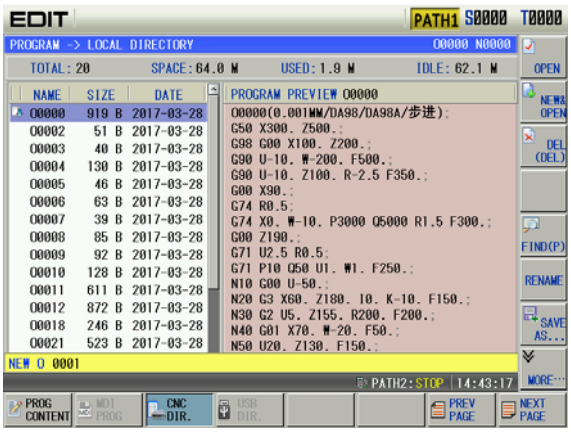


Fig 1

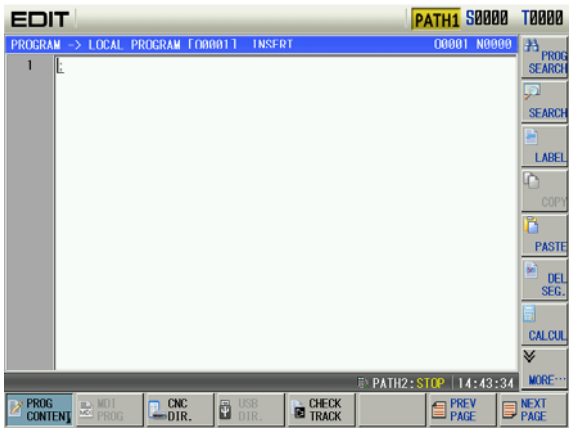


Fig 2

Note: When a program is needed to create, the input program name exists, the system opens the file, otherwise, it automatically create a new one.

9.1.3 Program opening

1) Press PROGRAM to enter the Program interface,press CNC DIR. softkey ,or after inserting u disk CNC displays USB DIR. softkey;

2) press ↑、↓、▤、▤ key,move the cursor to the place where you want to open the program name,press OPEN softkey or EOB key or DATA INPUT key ,enter into program.

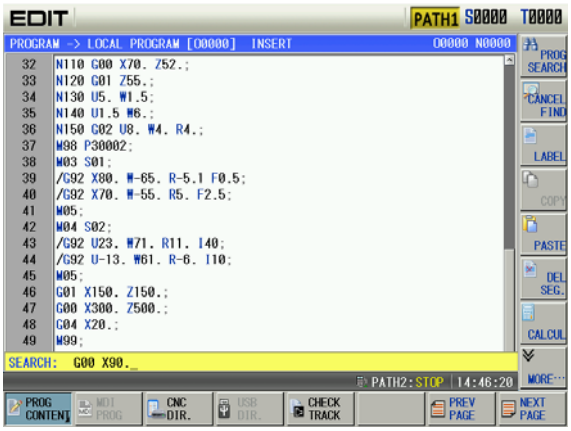
9.1.4 Inputting a program

After opening a program，the character will be displayed on the screen immediately as it is input(as for compound key, press this key repeatedly for alternate input), after a block is finished, press EOB or DATA INPUT to terminate it.

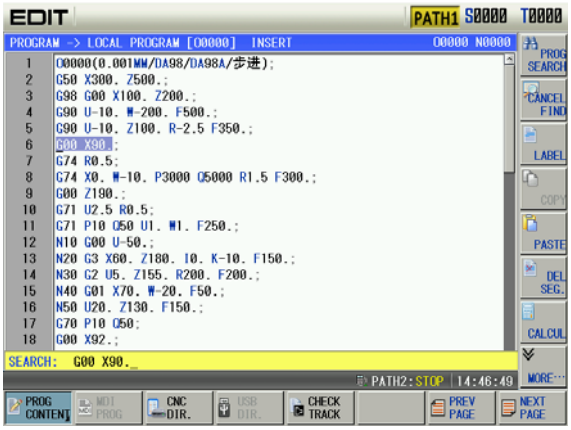
Note: The unexpected power-off when the program is input, the program being edited cannot be saved.

9.1.5 Searching a character

Press SEARCH, and input the required characters to be searched(can input one block). For example, finding “G00” page is shown below:



1) Press ↑ key (↑or ↓ is determined by the location of the character searched to the character where the cursor locates), it displays as follows:



2) After the finding, the CNC system is still in FIND state, press ↑ or ↓ key again, the next character can be found. Or press CHANGE key to exit the FIND state.

3) If the character is not found, the prompt of “Search fail” will be displayed.

Note: During the searching, it doesn’t search the characters in the called subprogram, and the character in subprogram is searched in subprogram.

9.1.6 Positioning the cursor to the specified line

Press SEEK, and input the line number (the line number of block is the left of one row label). For example, the cursor positions to the 10th line, the display page is shown below Fig 1;Press DATA INPUT (or EOB),and the display page is shown below Fig 2;

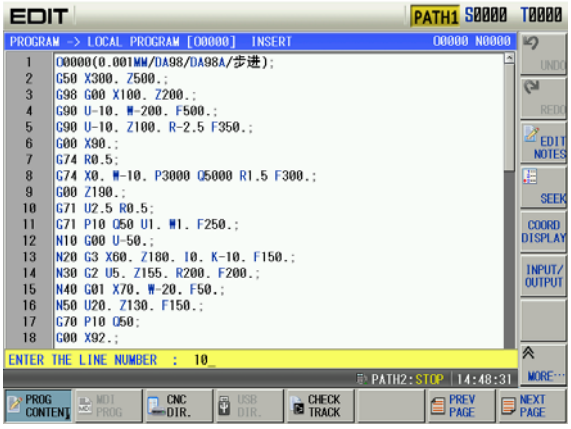


Fig 1

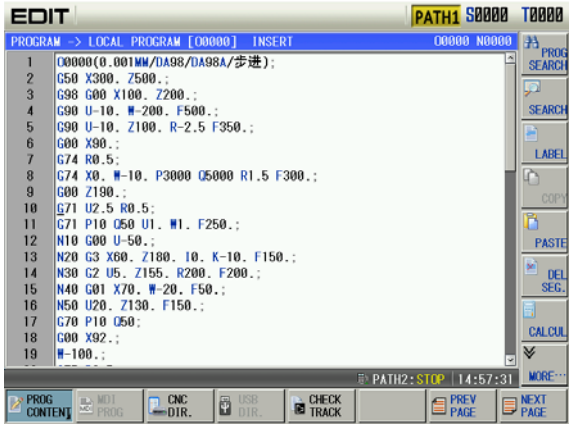


Fig 2

1. The method to return to the program head

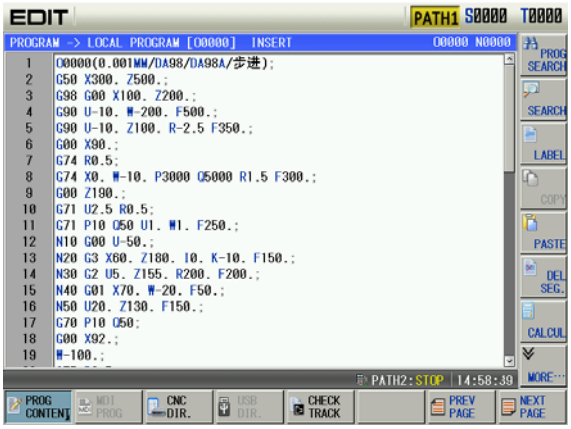
1) In Program interface of the Edit mode, press key, the cursor returns to the program head;

2) Search the program head character by the methods in OPERATION, Section 9.1.5.

9.1.7 Inserting a character

Press  key to enter the INS mode (the cursor is an underline, and the “Insertion” shows at the title column), the window is as follows: Fig 1;

Input the character to be inserted (to insert G98 code before G50 in the above figure, input G, 9, 8); it displays as follows Fig 2.



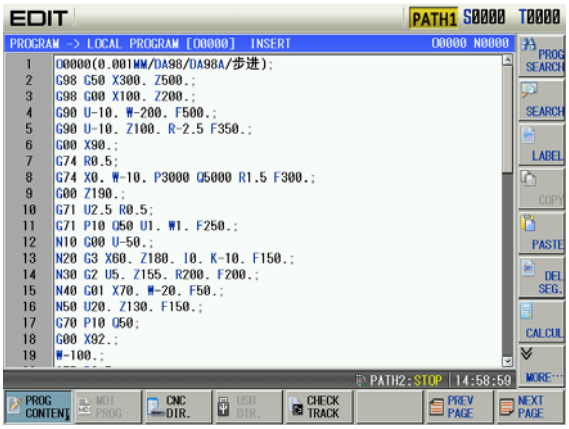


Fig 1 Fig 2

Note 1:In the Insert mode, if the cursor is not located at the row head, a space will be automatically generated when inserting the code address; if the cursor is located at the row head, the space will not be generated, and it should be inserted manually.

Note 2:In the Insert mode, if the previous bit before the cursor is a decimal point and the cursor is not located at the row end, input an address word, the “space” will be added automatically following the decimal point.

9.1.8 Deleting a character

Press  key to delete the character before the cursor.

9.1.9 Altering a character

Press  key to enter the ALT state (the cursor is a backlight rectangle, and the “Modification” shows at the tile column), the display page is as follows Fig 1; Input the modified character(as the above figure, alter X300 to X250, input 2, 5, 0), it displays as follows Fig 2:

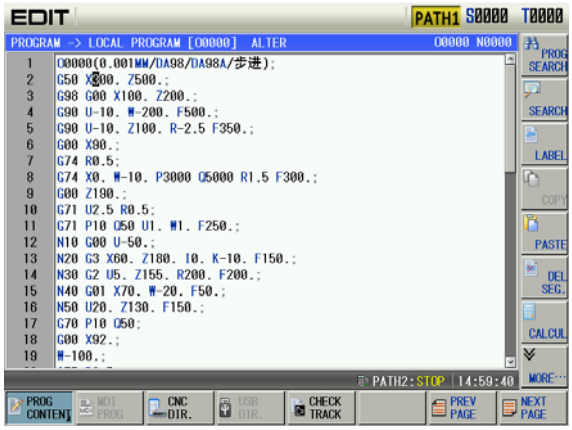


Fig 1

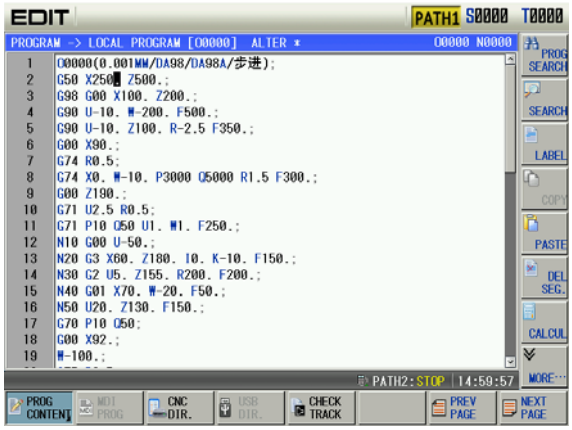


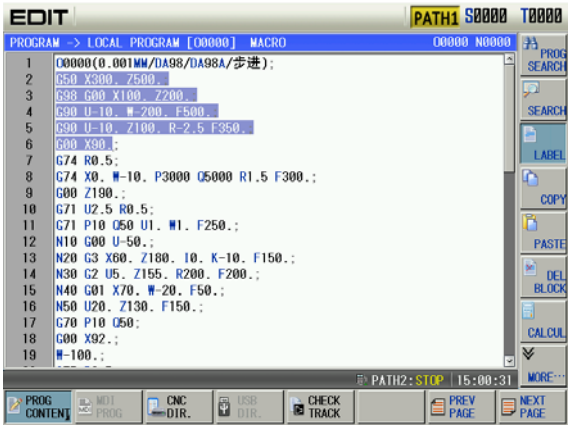
Fig 2

Note:In ALTER state, the current character where the cursor locates is altered for the input one, and the cursor moves a byte forward.

9.1.10 Deleting a single block

Single block deletion: Move the cursor to the head of the block to be deleted, then press DEL SEG. key.

Multi-block deletion: Move the cursor to the beginning of the block to be deleted, press the LABEL soft button, and then move the selected block by cursor, Delete the selected block by DELETE,the page is as follows:

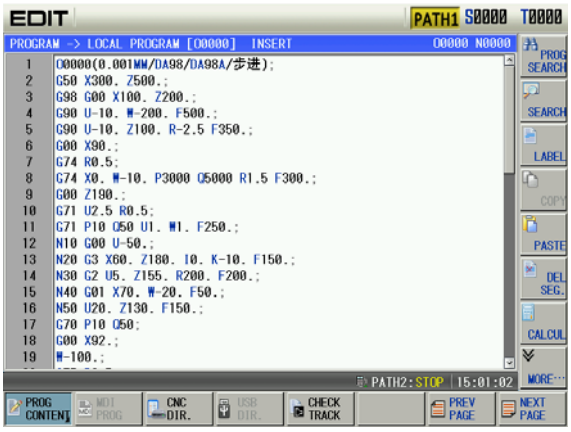


9.1.11 Copying and pasting a block

1) Move the cursor to the beginning of the block to be copied, press the LABEL soft button, and then move the selected block by cursor;
2) Copy the selected block by COPY soft button;
3) Move the cursor to the required paste position, press PASTE, and the copied block is pasted.

9.1.12 Canceling and recovering a program

1) Press MORE··· to enter the next menu, which is shown below:



2) When the current content is changed(after modification, copy or deletion), UNDO is pressed to recover the previous content before the copy;

3) Recover and cancel are matched. Press UNDO, REDO to cancel the previous operation.

Note: Up to 10 previous steps can be retreated by using the retraction function.

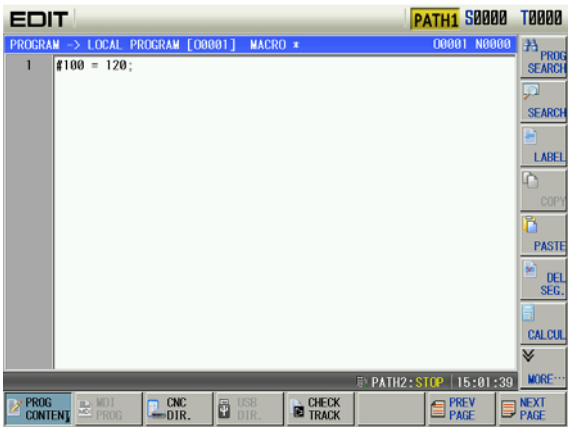
9.1.13 Program Save

In Edit mode, after the program is edited or modified, although the system automatically save programs at regular time, shift to other working mode from edit or switch the current display page and the CNC will automatically save the current program.

Note: "*" in the title bar is displayed when the program is not saved.

9.1.14 Macro Program Edit

Edit operation method selection, enter to the program page set by PROGRAM and then shift to the program content page，press PROG CONTENT soft key，enter to the program content page，press MARCO EDIT soft button to open the macro edit function menu, which is shown below:



In this case, automatically enter to the macro edit state, some special characters, such as ([,], ＝, ＞,＜, ＋ and ＊) can be input. The macro program command can be inserted by the function button at the right side, so that the macro edit is more simple and enabled.

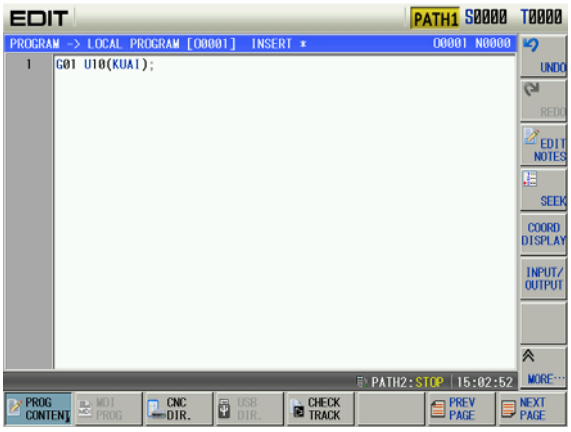
Note: In the macro program edit state, program index or scan operation is unavailable.

9.1.15 Creating & Modifying a Program Annotation

GSK980TTi can add annotation to each line, and its steps are shown below:

1) Press 

and the CNC automatically insert "( )" into the end of line where the cursor is, Input annotation character into "( ). For example, input "KUAI" character string, and the display page is shown below:



2) The modification of annotation is the same that of program content.

Note :The annotation added to the CNC could only be character. Actually, the CNC takes all content in "()" as annotation, so, adding other annotation can be executed by using PC editing, and then be transmitted to the CNC.

9.2 Deleting Programs

9.2.1 Deleting a Program

1) Select Edit mode, press 

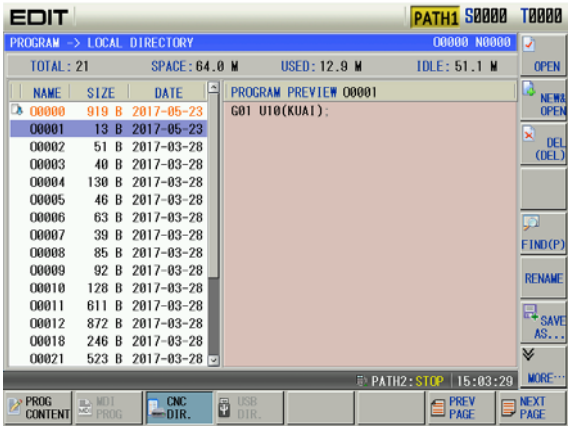
to enter program interface, press to 

enter the local program page;

2) Press  、  、  、  （ 

、 

）to select the required deleted program.,For example, select O0001 program and the display is shown below:



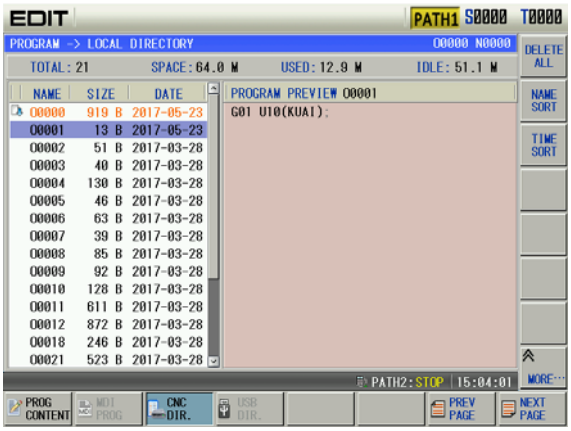
3) Press DEL (DEL) soft key，pop-up the dialog box,Press DATA INPUT and O0001 program is deleted; press CANCEL and the deletion is cancelled.

9.2.2 Deleting all Programs

Steps:

1) Refer to Section9.2.1 Method 2 to enter the local program page;

2) Press MORE··· to enter the next menu and the display is shown below:



3) Press DELETE ALL, and then press DATA INPUT，the all program are deleted.

9.3 Selecting a Program

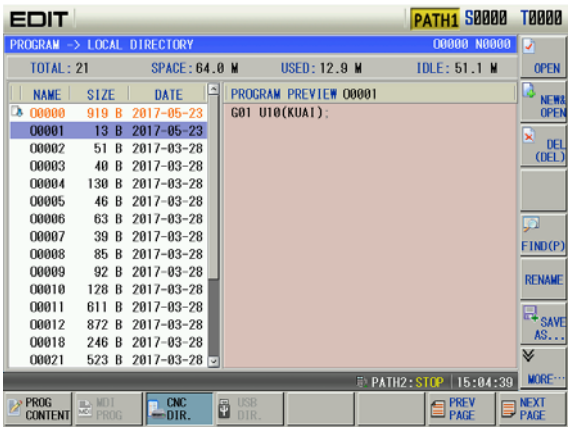
After the program to be executed is selected by the method in OPERATION, Section 7.1.3 select the Auto mode, then press CYCLE START key (or external cycle start key), the program will be executed automatically.

9.4 Renaming a Program

1) Select Edit mode, press 

to enter the program interface, and in necessity press 

to enter the local directory page:



2) Move the cursor to the program which is to be modified, press 

and input a new program name in the pop-up diaglog box, and at last press 

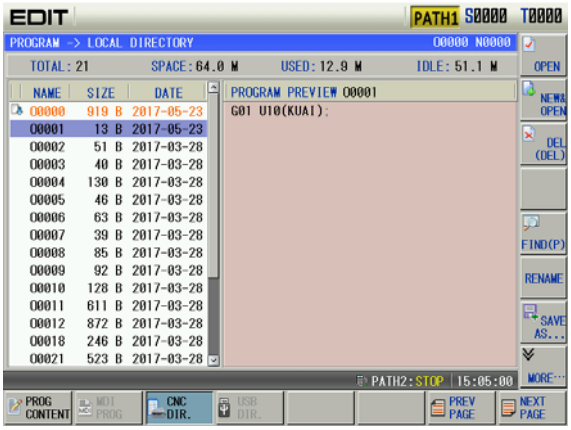
.

9.5 Copying a Program

1) Select Edit mode, press 

to enter the program interface, and in necessity press 

to enter the local directory page:



2) Move the cursor to the program which is to be modified, press 

and input a new program name in the pop-up diaglog box, and at last press 

.

9.6 Counter Function

1) press PROGRAM and PROG CONTENT to enter the program content page.

2) Spring out the counter window by CALCUL soft button; Alternatively, directly press the DATA OUTPUT at the program content page, the calculator may appear, refer to the following figure:

3) Control the corresponding button function on the MDI panel to perform the number input and calculation based upon the content of the calculator. The calculated result can be transmitted to the program cursor on the MDI panel based upon the displayed OUT SEND from calculator, and then retreat from the calculation function.

4) Press the CANCEL to retreat from the calculation function, alternatively, it can be carried out by CALCUL soft button.

9.7 Teaching function

Teaching is a programming method by the artificial guiding machining operation,980TTi adds a teaching function, so that the teaching programming is more simply and efficient.

Select the Manual or MPG operation method and press the PROGRAM to enter the program page set, and then enter to the program content page by pressing PROG CONTENT if necessary; refer to the following figure 1，Open the next optional menu by pressing the PROG CONTENT again, refer to the following figure 2:

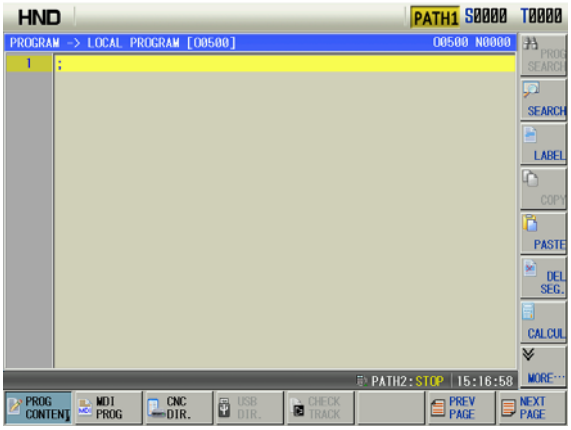


Fig 1

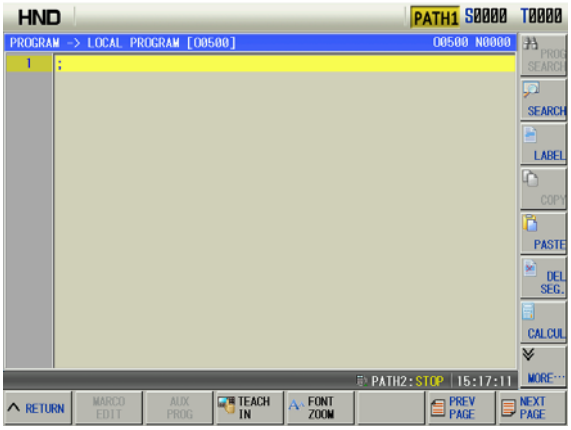
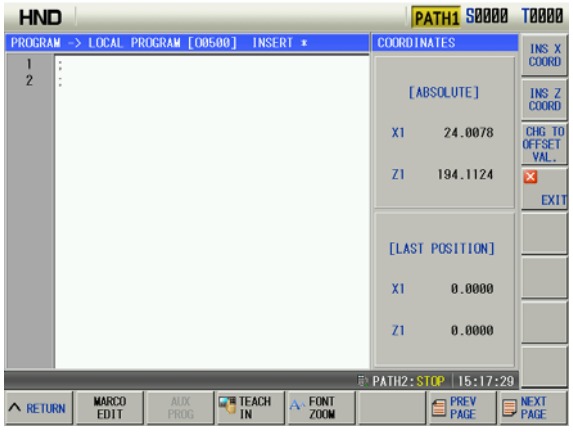


Fig 2

Enter to the Teaching function page by TEACH IN, refer to the following figure:



INS X COORD、INS Z COORD: Insert the current absolute coordinate (original value input state) or the sum (offset input state) of the absolute coordinate and offset value input for the corresponding axis at the cursor position.

CHG TO OFFSET VAL.: Shift the system to the retyping offset value when the absolute coordinate value along each axis is inserted; simultaneously, it becomes CHG TO CURRENT VAL. after controlling this button.

CHG TO CURRENT VAL.: Shift the system to state that only the absolute coordinate along each axis can be inserted; simultaneously, it becomes CHG TO OFFSET VAL. after controlling this button.

EXIT: Retreat from Teaching mode.

CHAPTER 10 TOOL OFFSET AND SETTING

The actual location of tool can be overlooked in programming for simplifying programming. Three methods including positioning tool-setting, trial tool-setting and machine zero tool-setting are available in this GSK980TTi system. The tool offset data are obtained from this tool-setting operation.

10.1 Tool Positioning Setting

Steps:

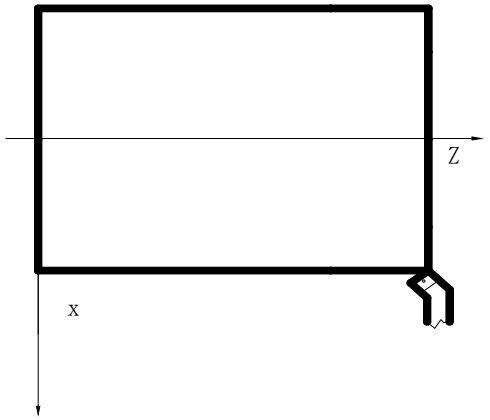


Fig. A

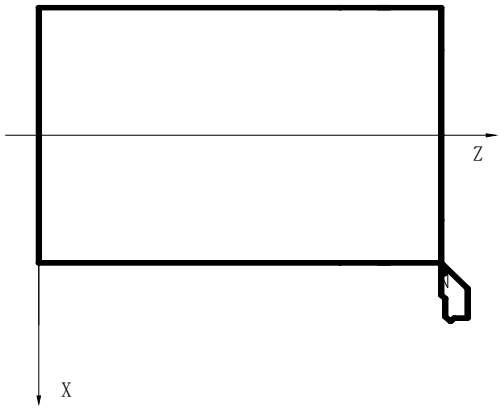


Fig. B

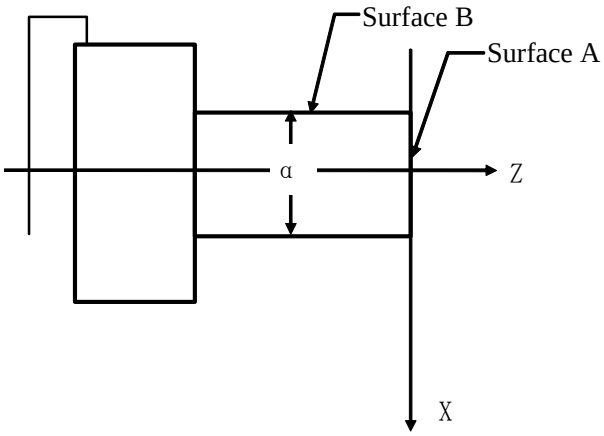
1. Firstly determine if the offset values are zero in X, Z, if not, clear all the tool number offset values;
2. Set the offset No. for 00 (i.e. T0100,T0300), as for the offset value: (method: execute a move code or perform the machine zero return in T0100 state, then clear the offset value automatically as returning to the machine zero);
3. Select a tool by random (usually the 1st tool, this tool will be used as the reference tool);
4. Position the tool nose of the reference tool to a point (tool-setting point), as shown in Fig. A;
5. In PRG STATE page of the MDI mode, set up the workpiece coordinate system by the command G50 X__ Z__;
6. Clear the relative coordinate (U, W);
7. After the tool is moved to a safety height, select another tool and move it to the setting point, as shown in Fig. B;
8. Press OFFSET key and move the cursor by ↓, ↑ key to select the corresponding offset number of that tool;
9. Press address key U, then press DATA INPUT key to input the tool offset value of X axis into the corresponding offset number;
10. Press address key W, then press DATA INPUT key to input the tool offset value of Z axis into the corresponding offset number;
11. Repeat the steps from 7 to 10 to perform the tool-setting operation for other tools.

Note:For the fixed tool-setting, the original system tool offset should be cleared, input the new offset one time instead of many times by pressing U, W. Refer to II OPERATION, Section 10.5.4 about the tool compensation clearing method.

10.2 Trial Tool-setting

Whether the method of trial tool-setting is inactive is defined by the system parameter No.012 Bit5.

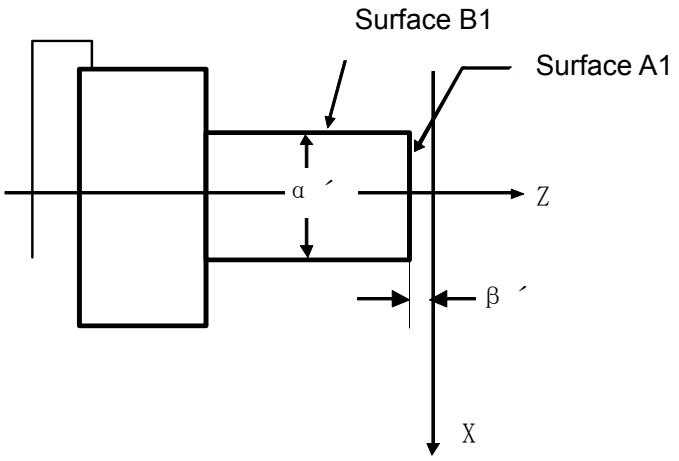
Steps (workpiece coordinate system by part end surface):



1) Select a tool by random and make it cut on Surface A;
2) Retract the tool along X axis without Z axis moving and stop the spindle; directly press RECORD X POS. and RECORD Z POS., and the CNC records the absolute values of the position, at the moment, directly move the tool;

Press OFFSET key to enter the Offset interface, select the TOOL OFFSET page, then move

3) the cursor by pressing ↑, ↓ key to select the corresponding offset number;
4) Key in by sequence the address key Z, number key 0 and DATA INPUT key;
5) Make the tool cut along Surface B;
6) Retract the tool along Z axis without the movement of X axis, and stop the spindle; directly press RECORD X POS. and RECORD Z POS., and the CNC records the absolute values of the position, at the moment, directly move the tool;
7) Measure the diameter "α" (supposing α=15) ;
8) Press OFFSET key to enter the Offset interface, select the TOOL OFFSET window, then move the cursor by pressing ↑, ↓ key to select the corresponding offset number;
9) Key in the address key X by sequence, number key 1, 5 and DATA INPUT key;
10) Move the tool to a safety height to change for another tool;



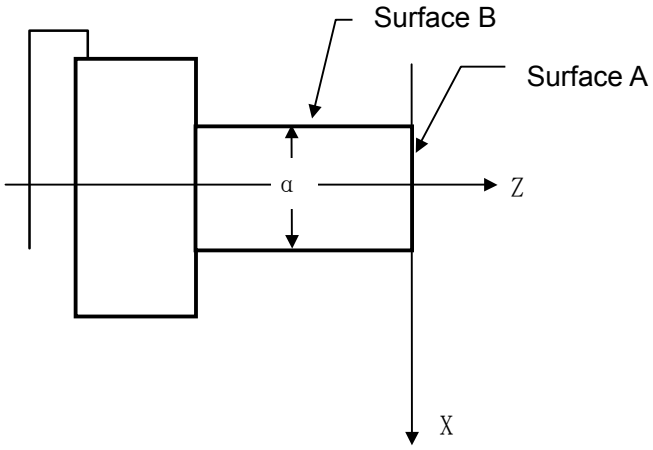
11) Make the tool to cut on Surface A1;
12) Retract the tool along X axis without Z axis moving and stop the spindle;
13) Measure the distance "β´" between the Surface A1 and the workpiece coordinate origin(supposing β´=1) ;
14) Press OFFSET key to enter the Offset interface, select the TOOL OFFSET page, then move the cursor by pressing ↑, ↓ to select the corresponding offset number;
15) Key in by sequence the address key Z, sign key +/-, number key 1, and INPUT key;
16) Make the tool to cut on Surface B1;
17) Retract the tool along Z axis without the movement of X axis, and stop the spindle;
18) Measure the distance "α' " (supposing α'=10);
19) Press OFFSET key to enter the Offset interface, select the TOOL OFFSET window, then move the cursor by pressing ↑, ↓ to select the corresponding offset number;
20) Press orderly the address key X, number key 1, 0 and INPUT key;
21) Repeat the execution from Step 10 to Step 20 to perform the tool-setting operation for other tools.

Note:The offset value may be large by this tool-setting method, so the tool compensation should be done by the coordinate offset by the CNC system. (set the BIT4 of the CNC parameter No.003 to 1). Moreover, the tool lengths compensation should be performed by using the T code in the 1st block, or the 1st move block should contain the T code for the tool length compensation.

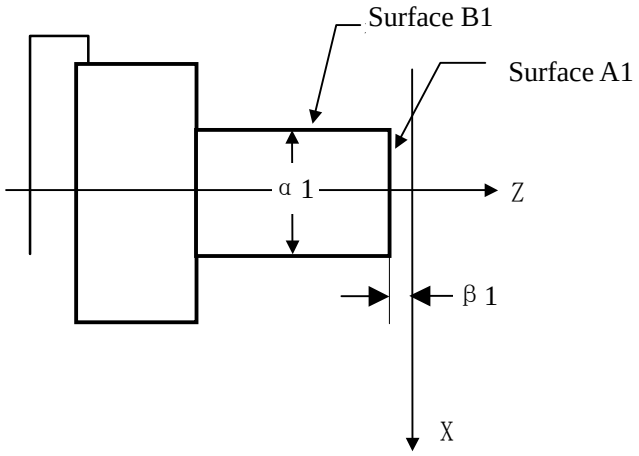
10.3 Tool-setting by Machine Zero Return

There is no reference tool in this tool-setting methods, when the tool is worn or to be adjusted, it only needs to be set again, and a machine zero return should be done before the tool-setting. The machining could be continued by performing a machine zero return at power on after power-off, which is very convenient for the operation.

Steps (workpiece coordinate system by part end surface):



1) Press  key to enter Machine Zero mode, move axes to machine zero;
2) Select a tool by random and set the offset number of the tool to 00 (e.g. T0100, T0300) ;
3) Make the tool to cut on Surface A;
4) Retract the tool along X axis without the movement of Z axis, and stop the spindle; directly press RECORD X POS. and RECORD Z POS., and the CNC records the absolute values of the position, at the moment, directly move the tool;
5) Press  key to enter the Offset interface, select the TOOL OFFSET window, then move the cursor by pressing ↑, ↓ key to select the corresponding offset number;
6) Key in by sequence the address key Z, number key 0 and DATA INPUT key to set the offset value of Z axis;
7) Make the tool cut along Surface B;
8) Retract the tool along Z axis without the movement of X axis, and stop the spindle;
9) Measure the distance "α"(supposing α=15);
10) Press  key to enter the Offset interface, select the TOOL OFFSET window, then move the cursor by pressing ↑, ↓ to select the corresponding offset number;
11) Key in by sequence the address key X, number key 1, 5 and DATA INPUT key to set the offset value of X axis;
12) Move the tool to a safety height for tool change;
13) Change for another tool, and set the tool offset number to 00 (i.e. T0100, T0300);



14) Make the tool to cut on Surface A1;
15) Retract the tool along X axis without Z axis moving and stop the spindle; measure the distance "β1" between the Surface A1 and the workpiece coordinate system origin (supposing β1=1) ;
16) Press OFFSET key to enter the Offset interface, select the TOOL OFFSET window, then move the cursor by pressing ↑, ↓ to select the corresponding offset number;
17) Key in by sequence the address key Z, sign key -/+, number key 1, and DATA INPUT key to set Z offset value;
18) Make the tool to cut on Surface B1;
19) Retract the tool along Z axis without the movement of X axis, and stop the spindle;
20) Measure the distance "α1" (supposing α1=10);
21) Press OFFSET key to enter the Offset interface, select the TOOL OFFSET window, then move the cursor by pressing ↑, ↓ key to select the corresponding offset number;
22) Key in by sequence the address key X, number key 1, 0, and DATA INPUT key to set X offset value;
23) Move the tool to a safety position;
24) Repeat the execution from Step 12 to Step 23 to perform the tool-setting operation for other tools.

Note 1:Machine zero switch must be fixed for the tool-setting operation by machine zero return.

Note 2:The workpiece coordinate system setting can't be done by G50 code after tool-setting by machine zero return.

Note 3:The tool compensation should be done by coordinate offset by the CNC system (the system parameter No.003 Bit4 is set to 1), further more, the tool lengths compensation should be performed by using the T code in the 1st block, or the 1st move block should contain the T code for the tool length compensation.

Note 4:The corresponding parameters should be set as follows:

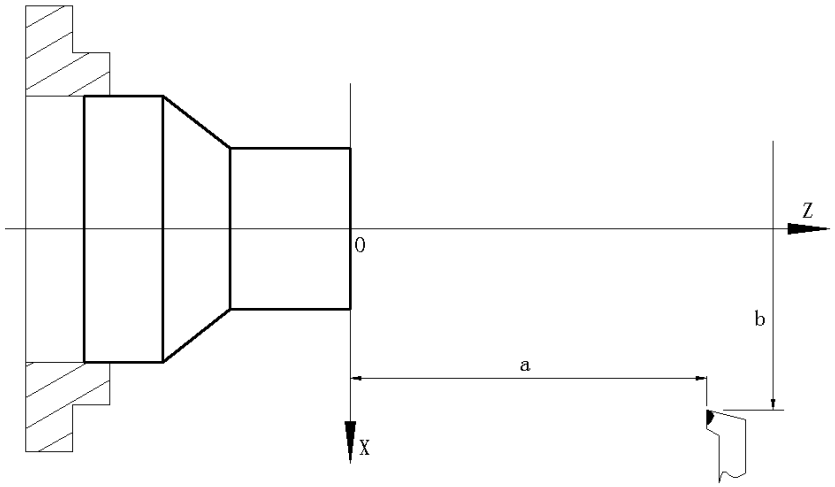
Bit7=0 of the system parameter No.004;

Bit5=1 of the system parameter No.012;

Bit7=1 of the system parameter No.012.

Note 5:The setting values of the system parameter No.047 should be close to the absolute coordinate values of machine zero in workpiece coordinate system XOZ, as is shown in the following figure:

As shown below:



Example:After machine zero returning, when the absolute coordinate of the tool in workpiece coordinate system is (a,b), the setting value of system parameter No.047 should be close to a,b.

10.4 Coordinates Record

When the trail-cut tool-setting or machine zero return tool-setting is executed, the coordinates record function can be used to get the convenient tool-setting operation.

When the tool-setting is executed and after the tool cuts along X (or Z), the following steps are executed to directly move the tool to the safety position as follows:

1) Press 

to enter the tool offset interface, and then press 

to enter the tool offset page，refer to the following Fig.1.

2) Press 

and 

, absolute corrdinates of X and Z（without including current tool offset values）are recorded in the CNC. After the absolute values are recorded correctly, there is a flash “Record coordinates” in the upper of the screen, and there is a prompt “Current X、Z Absolute coordinates have been recorded” in the bottom of the screen , refer to the following Fig.2.

3) Retract the tool to the safety position, and stop the spindle. When the diameter is measured for the outer circle and the distance from the reference plane to it is done, the measured value is input in the tool offset page; the system counts the new tool offset value and sets it to the selected tool offset number according to the input value and the previous recorded absolute coordinates.

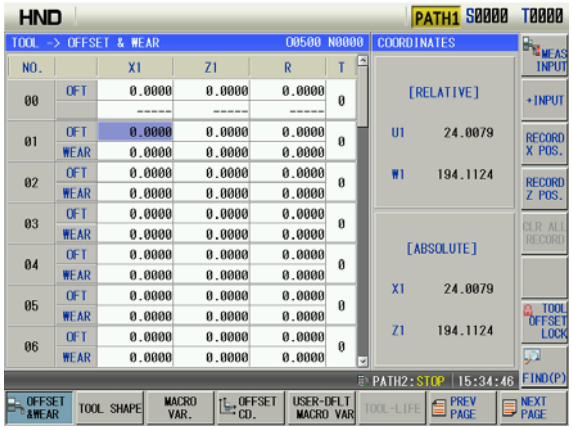


Fig.1

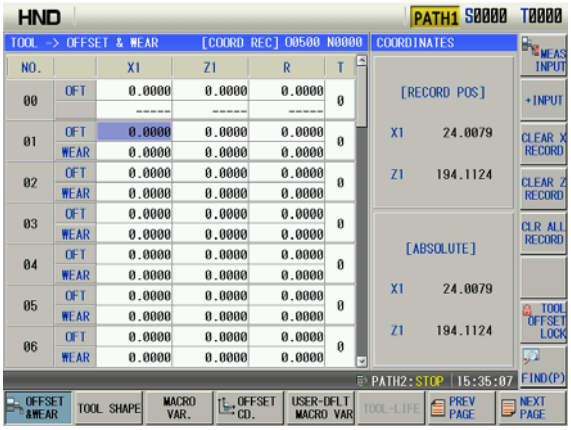


Fig 2

Use the following methods to clear the recorded coordinates when the above methods are used to execute the tool-setting:

1) In the tool offset page, after the absolute coordinates are input in some tool offset(for example: Z0 is input in the No.01 tool offset number) and then the axis moves, the recorded coordinates are cleared;

2) Press CLEAR X RECORD or CLEAR Z RECORD or CLR ALL RECORD in the tool offset page and the recorded coordinates are cleared.

10.5 Setting and Altering the Offset Value

10.5.1 Offset Setting

1) Press OFFSET key to enter the offset setting page,select the desired window by pressing the , ;

2) Move the cursor to the location of the tool offset, wear number to be input.

Scanning: Press , to move the cursor in sequent

Searching: use the following press keys to directly move the cursor to the input position:

Press P + offset number + DATA INPUT or FIND(P) and input the offset number in the pop-up dialogue box, press DATA INPUT and the cursor directly positions the target offset number.

3) Numerical value input.

Method 1: Input number after pressing the address button (the corresponding axis name button for the X, Z, etc.);

Method 2: Input number after pressing the MEAS INPUT soft button;

4) Press  and the CNC calculates the offset value automatically and displays the result in the window. Its range:-9999.999~9999.999（IS_B）/-9999.9999~9999.9999（IS_C）(unit：mm)

10.5.2 Offset alteration

1) Refer to OPERATION, Section 10.5.1, and move the cursor to the location of the offset number that is to be altered;
2) If the offset value of X axis is to be altered, key in U; as for that of Z axis, key in W; alternatively, select the data position to be changed by 

soft button;
3) Then key in the incremental value;
4) Press  to add the current offset value to the value keyed in, the operation result will be displayed as a new offset value.

Example: The set X axis offset value is 5.688
The increment keyed in is U 1.5
Then the new offset value is 8.188(=5.688+1.5)

Example: The external diameter of the workpiece to be machined is Φ55.382, and the No.01 offset value is applied in the machining. Before machining, the values in No.01 are shown as follows table:

No.	X	Z	T	R
00	0	0	0	0
01	16.380	-24.562	0	0

In machining, the actual external diameter measured of workpiece is Φ55.561, so the offset value of No.01 can be altered as follows table:

No.	X	Z	T	R
00	0	0	0	0
01	16.201	-24.562	0	0

16.380-（55.561-55.382）

Note 1:While changing the offset value, the new offset value takes effect after the T code is executed.

Note 2:If the actual workpiece dimensions doesn't conforms to that of the part drawing, subtract the error from the original offset value for the oversize workpiece, add the error to the original offset value for the undersize workpiece.

10.5.3 Offset Values Clearing

1) Move the cursor to the offset number to be cleared;
2) When the offset value of X axis is to be cleared,  is pressed, then  is done, the offset will be cleared. Other axis clearing is done like that;

Note: The offset clearing in the tool offset page doesn't mean that the system in under the state with no

offsetting, if the system is needed to be in this state, the offsetting is required to be executed, which is shown as follows:

Execute a positioning command in T0100 state or perform a machine zero return. After the offsetting is finished, the "□□"in "T○○□□" shown at the right bottom of the screen will not be backlighted.

10.5.4 Setting and altering the tool wear

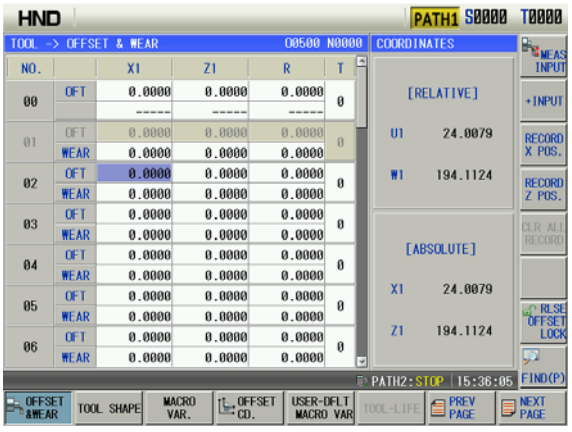
The setting and alteration methods for the tool wear are approximately identical to that of the tool offset, and the wear value is input by U(X axis), W(Z axis), V(Y axis). The wear input range is defined by the data parameter No.140, and they are saved even at power down.

10.5.5 Locking and unlocking the offset value

In order to protect the offset value to be operated by mistaken, the offset values can be put into lock, the operation steps are as follows:

1) Move the cursor to the location of the offset number to be locked;

2) Press the CHANGE or TOOL OFFSET LOCK key, the current offset value will be backlighted for locking, which is forbidden to alter, press the CHANGE or RLSE OFFSET LOCK key again, the locking may be cancelled. No.1 tool offset is locked as follows:



Note:The tool wear values can't be locked.

10.5.6 No.0 tool offset moving workpiece coordinate system

When No.012 Bit 6 is set to 1, No. 0 tool offset moving workpiece coordinate system is valid. After the value is input in No. 0 tool offset, the workpiece coordinate system executes the offset based on the input value.

Example: after X100, Z100 in No.0 tool offset is input, the workpiece coordinate system offsets X100, Z100.

Note: When No. 0 offset modification is valid real-time, No. 0 tool offset must be set before the system runs a program, otherwise, the run path will offset.

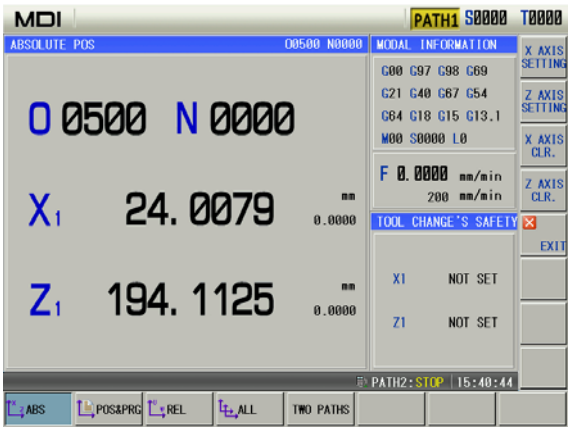
10.6 High Efficiency Tool-Change

it can be reached to the high-efficiency tool-change. It is better to set the Bit 7 of the state parameter to "1".

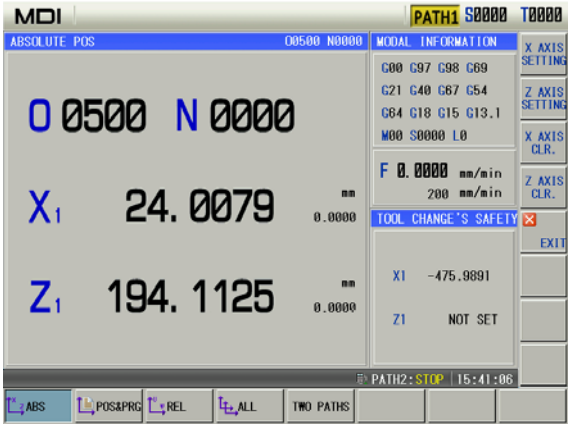
10.6.1 Tool-Change Safety Area Setting

To carry out the high-efficiency tool-change in the actual machining, the machine operator should be performed within the safety area setting based upon the following steps:

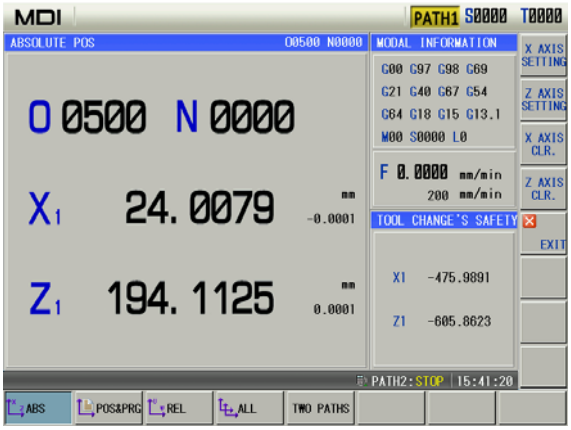
1) In the absolute coordinate page, press TOOL SAFETY SETTING soft key, the tool-change safety point setting information window is shown and covered the integrated information window. It will show a unset state when X or Z axis does not set; it will show the coordinate value of setting when X or Z is set. The initial state is as follows:



2) Manually move the tool-post at the side of + X of the workpiece, and the optimized position of the impact will not occur when the tool-change executes at this side; record the X axis coordinate at this position by pressing the X AXIS SETTING; refer to the following figure:



3) Manually move the tool-post at the side of + Z on the surface of the workpiece, and the optimized position of the impact will not occur when the tool-change executes at this side; record the Z axis coordinate at this position by pressing the Z AXIS SETTING; refer to the following figure:



In the above-mentioned interface, shift the interface or press the EXIT automatically conceal the small window of “Tool-change safety point setting”, this area recovers and displays the small window of the “Integral information”.

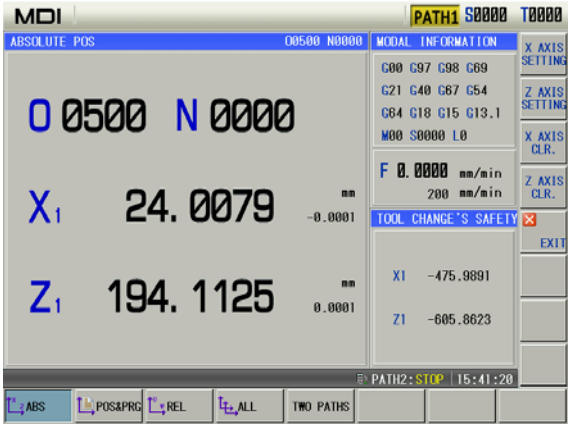
The high-efficiency tool-change function can be enabled after the X and Z are set; the setting value at the tool-change safety area is memorized after the power is turned off. After the high efficiency tool-change is enabled, the tool mark is then shown at the right corner of the page “”.

Note: The record in the setting of the tool-change safety point is the machine coordinate.

10.6.2 Setting Clear in Tool-Change Safety Area

The setting value of the set tool-change safety area can be eliminated by the following methods:

1)The set data along with the X or Z can be eliminated by soft button X AXIS CLR. or Z AXIS CLR. after the tool-change safety point setting window is shown;



2) The setting data set by X or Z axis on the tool-change safety area will be automatically eliminated after the G50 command sets the coordinate system;

3) The setting data set by X or Z axis on the tool-change safety area will be automatically eliminated after the machine zero return is operated;

4) On the tool offset page, if the tool-setting operation is performed by the measure input method or trial cutting tool-setting method; the current tool is regarded as to be changed; the setting data set by its corresponding data on the tool-change safety area will be automatically eliminated.

10.6.3 High-Efficiency Too-Change Programming Format

In order to realize the high-efficiency tool-change, user should use the shared programming format both the positioning G code and Tool-change T code when the machining procedure is programmed.

G00 X_ Z_ T_ or

G01 X_ Z_ T_

T code is performed sharing a same block with G00 or G01 code, simultaneously, the T code is performed while the positioning code executes; and then deliver the tool-change strobe signal TF to PLC. When the plate moves to the tool-change safety area, CNC delivers the tool-change allowable signal TCEN. PLC program triggers the tool-change operation by identifying the tool strobe signal TF and tool-change allowable signal TCEN.

When T code does not share a same block with the movement code, simultaneously output tool strobe signal TF and tool-change allowable signal TCEN when T code is performed.

10.6.4 Precaution of High-Efficiency Tool-Change

1) When the tool-change code does not performed with a same block to the axis positioning code; the tool-change allowable signal will be output before performing the tool-change;
2) When the tool-change code and axis positioning code (G00 or G01) are performed based upon a same block; the tool-change allowable signal TCEN output after the carriage plate enters to the safety area. When the tool-change code and other movement codes are performed with a same block; the tool-change allowable signal TCEN outputs after the movement ends;
3) When the high-efficiency tool-change is performed, It is necessary to prevent the tool rotation from impacting to other machine components during that the plate moves to the tool-change point within the tool-change safety area;
4) High-efficiency tool-change function is suitable for the tool-change safety detection within the XZ panel, if additional axes are placed at the machine tool, it is necessary to confirm that tool-change will not interfere to other additional axes when programming;
5) After the G50 command setting the coordinate system ,the machine zero return being operated or the current tool being regarded as to be changed ,the setting data set by X or Z axis on the tool-change safety area will be automatically eliminated，the high-efficiency tool-change function can be enabled after resetting the tool-change safety area operation.

CHAPTER 11 COORDINATE SYSTEM

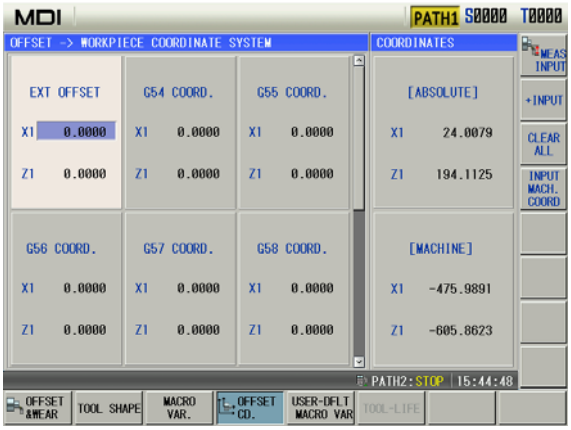
GSK980TTi owns workpiece coordinate system selection function, which can be contented several workpiece coordinate system during the machining; specifying one of the G54~G59 code in the block can be selected in the workpiece coordinate systems 1~6.

11.1 Coordinate System Setting

Press 

, 

to enter its corresponding page:

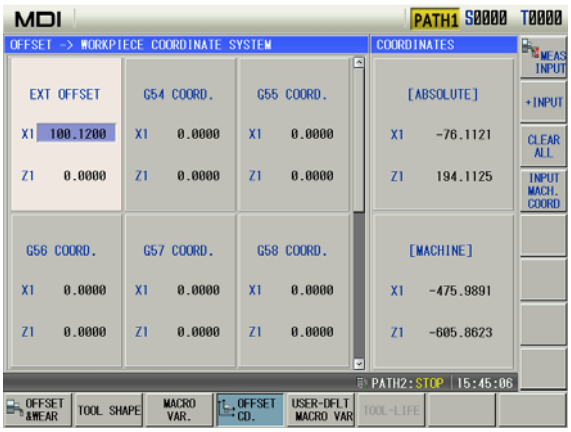


11.1.1 Directly Input

Enter to the workpiece coordinate system page, press 

Move the cursor to the inputted position, input the numerical value by the number buttons, and then directly input the numerical value by the 

button.

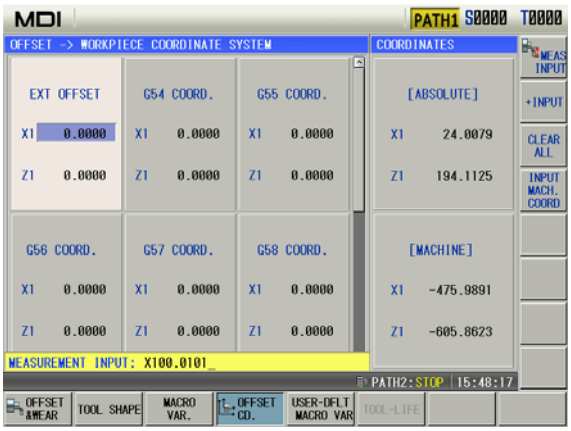


11.1.2 Measure Input

CNC automatically calculates the data such as the cutter compensation or zero offset, etc. when using the measure input, so that the input data becomes the absolute coordinate of the new coordinate system at the machine's current position.

Method 1: Press  or  Move the cursor to the inputted position, input the coordinate value after pressing the  soft button，the CNC will automatically calculate to input the data by controlling the  button;

Method 2: Move the cursor to the selected coordinate system by pressing  or  , key the coordinate value by address button  or  ; CNC automatically calculates and then inputs the data by  ;



Note: The input data from measure will become the absolute coordinate of which the current point locates at the setting coordinate system, for example:

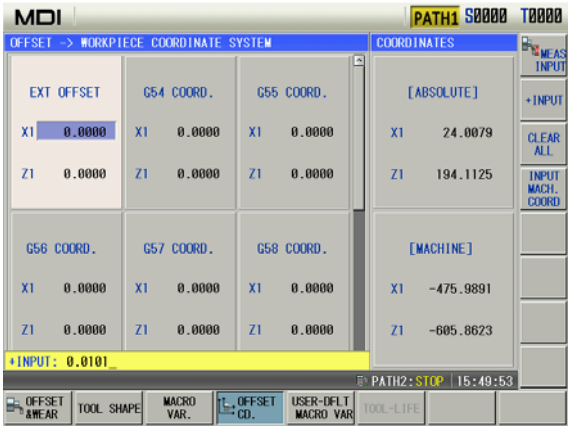
System executes G54 command, input X100; Z100 in G54 coordinate system by measure input method. CNC automatically calculates the G54 offset value, so that the absolute coordinate at the current position will become X100; Z100.

11.1.3 Machine Tool Coordinate Record

Enter to the workpiece coordinate system setting page, move the cursor to the inputted position by  or  ; directly input the machine tool coordinate by  soft button.

11.2 Coordinate System Modification

Enter to the workpiece coordinate system setting page，press  、  ,move the cursor to the data position to be modified; input the incremental value by the  soft button  ,press  , add the current workpiece coordinate system offset value and the inputted incremental value, and its result will show as a new workpiece coordinate system offset value.



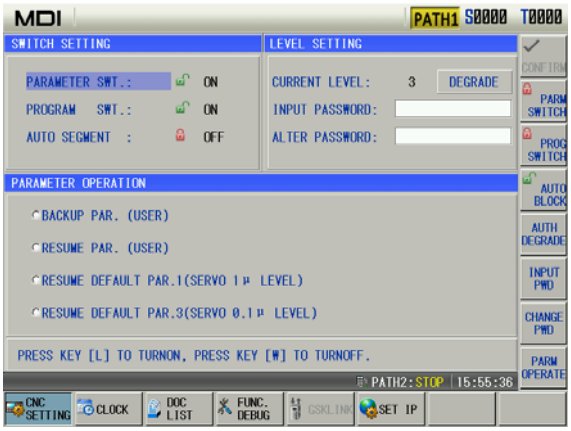
11.3 Coordinate System Clearing

Enter to the workpiece coordinate system setting page; the overall workpiece coordinate offset values will clear after the confirmation by pressing the CLEAR ALL soft button; alternatively, Use the directly input method of the workpiece coordinate system in this chapter to set the axis offset value of any coordinate system as "0".

CHAPTER 12 DATA SETTING

12.1 Switch setting

Press  and then  to enter the CNC setting page which includes the switch setting, level setting and parameter operation function as follows:



press the  、  、  、  to move the cursor to the setting item，the system can set parameters, programs, automatic sequence number ON/OFF state.

12.1.1 Switch explanation

Parameter switch: parameters can be modified when the parameter switch is ON; they are forbidden to alter when it is OFF.

Program switch: programs can be edited when the program switch is ON; they are forbidden to edit when it is OFF.

Automatic block number: the block number is automatically generated when the automatic sequence number switch is ON and a program is edited; when the switch is OFF, the block number is input manually instead of being automatically generated.

12.1.2 Switch setting method

a).Move the cursor to the setting item;

b) Directly press the  ,  or  soft button to open the switch, if it is ON, press the corresponding  ,  or  soft button to close it; simultaneously, the cursor will automatically move to the switch item.

Note:When the parameter switch is shifted from “OFF” to “ON”, an alarm will be issued by CNC system. The alarm can be cancelled by pressing  . If the PARM SWT is shifted again, no alarm is issued. For security it should set the PARM SWT for “OFF” after the parameter alteration is finished.

12.2 Level setting

To prevent programs, CNC parameters from being modified at will, GSK980TTi provides the level setting function, and its password grade is divided into five, from high to low grade: 2nd grade (machine manufacturer), 3rd grade (device management), 4th grade (engineer), 5th grade (machining operation), 6th grade(operation limit). The current operation grade is displayed in “Current Level:_” in the level setting page.

2nd grade:it is for the machine manufacturer, which permits the machine manufacturer to alter the state parameters, data parameters, pitch compensation data, tool compensation data, to edit programs, to transmit PLC and so on.

3rd grade: it permits to alter state parameters, data parameters, tool compensation data and to edit programs.

4th grade: it permits to alter tool compensation data (tool-setting operation), macro variables, to edit programs, but not to alter state parameters, data parameters and pitch compensation data.

5th grade: it permits to alter tool compensation data, not to select and edit programs, and not to alter state programs, data programs and pitch compensation data.

6th grade: it has no level to operate only the machine operation panel, not to alter tool compensation data, not select and edit programs, not to alter state parameters, data parameter and pitch compensation data.

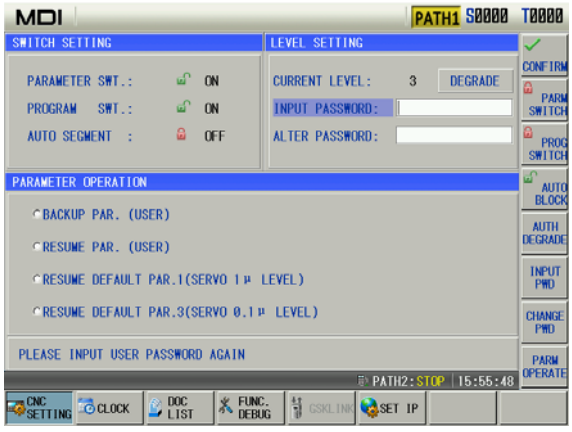
12.2.1 Operation grade entry

a).Move the cursor to the line “INPUT PASSWORD” in the CNC setting page;

b)Input the operation password(the system displays one “*” as soon as a number is input;

c).Press  and the operator can enter the corresponding grade operation;

d).After the operator enters the corresponding operation grade page, the prompt column prompts the system’s current grade, and the corresponding operations as follows:



Note 1: The defined password length corresponds to the operation grade, and the user cannot increase or reduce the length at will, which is shown below:

Operation grade	Length	Initial password
3rd grade	5 digits	12345
4th grade	4 digits	1234
5th grade	3 digits	123

6th grade	None	None

Note 2: When the operation level is less than or equal to the 3rd grade (3rd, 4th, 5th or 6th grade) and the system is turned on again, the operation level does not change. When the level is the 2nd grade and the system is turned on again, the operation level defaults to the 3rd grade.

12.2.2 Password modification

Steps:

a) The operator enters the grade to alter the password according to the method "Operation grade entry" after the system enters the setting page;

b) Press the CHANGE PWD soft button or manually move the cursor; Move the cursor to the line "ALATER PASSWORD";

c) Input the new password, and then press DATA INPUT;

d) The CNC prompts " Input the new password again";

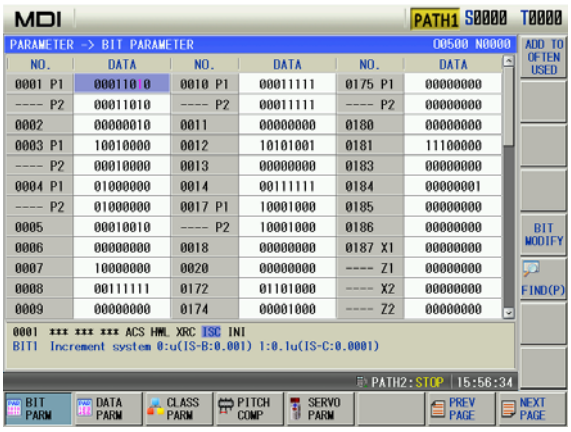
e) Input the new password again and then press DATA INPUT. When the two input passwords are identical, the CNC prompts "Password has been altered, please save the new password." The password alteration is done successfully.

Note:The system prompts, "The new password is not identical with the confirmation" when the two input passwords are not identical. At the moment, input the new password again.

12.3 Parameter Setting

Parameter setting can adjust related characteristics of the drive unit and machine.

Press PARAMETER to enter Parameter interface which includes BIT PARM, DATA PARM, CLASS PARM, PITCH COMP, SERVO PARM as follows:

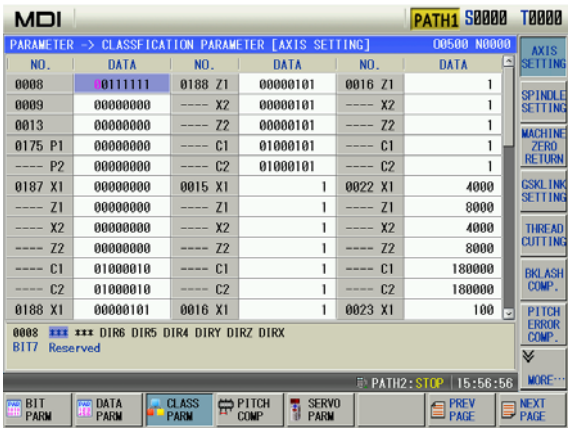


12.3.1 Parameter Lookup

Method 1:Press  or to enter the page containing the parameter to be set, press  or  key to move the cursor to the parameter number to be set;

Method 2:Press  (or ) , and input the parameter number in the pop-up dialogue box, at last, press  ;

Method 3:Press  and the parameter can be displayed based upon its function.The current displayed classification parameter can be shifted by its corresponding soft button.



12.3.2 parameter alteration

1) Turn on the parameter switch;
2) Press  to enter the MDI mode;
3) Move the cursor to the parameter number to be set;
4) Alteration methods

   Method 1:Straight number keys enter new parameter values or entire bytes directly;

   Method 1:Press  （or  ）to enter the bit alteration mode, at the moment, some bit of the parameter is backlighted. Press  or  to move the cursor to the bit to be altered, then key in 0 or 1;
5) Press  key, the parameter value is entered and displayed;
6) For security, the PARM SWT needs to be set to OFF after all parameter settings are finished.

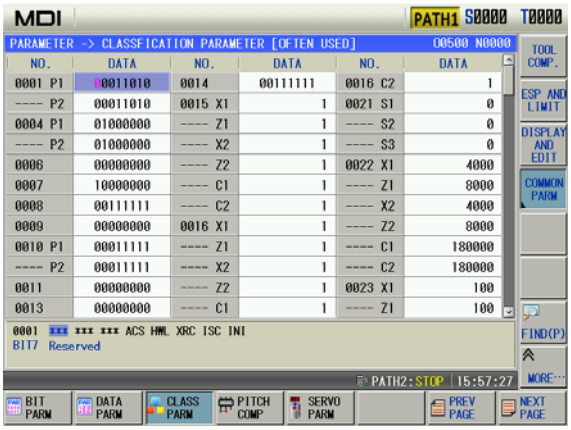
Note1:After the system enters some bit of the parameter,  or  is pressed to escape from the alteration mode, and the cursor stops the parameter number .

Note2:The screw-pitch parameter can only be altered under the 2 level password authority

Note3: The operation of setting and alteration methods for the PLC data like above steps。

12.3.3 Often used parameters

Press PARAMETER to enter parameter interface, and press COMMON PARM to enter the often used parameter page. The page is shown below:



The user can add the often used state parameter, data parameters to the often used parameters to get the convenient search and alteration.

12.3.3.1 Adding often used parameters

In the state, data or classification parameter page, move the cursor to the parameter number where is desired to add as the common parameter; press the ADD TO OFTEN USED soft button, the current parameter is then add as the common one.

12.3.3.2 Sorting often used parameters

In the common parameter page, enter to the next menu of the common parameter by pressing the COMMON PARM soft button again; refer to the following figure:

Pressing MOVE UP, MOVE DOWN can move the parameter where the current cursor is. Pressing AUTO ORDER sorts all often used parameter in sequence number.

12.3.3.3 Removing often used parameters

Move the cursor to the required parameter in the often used parameter page. Press 

and the current often used parameter is removed. Press 

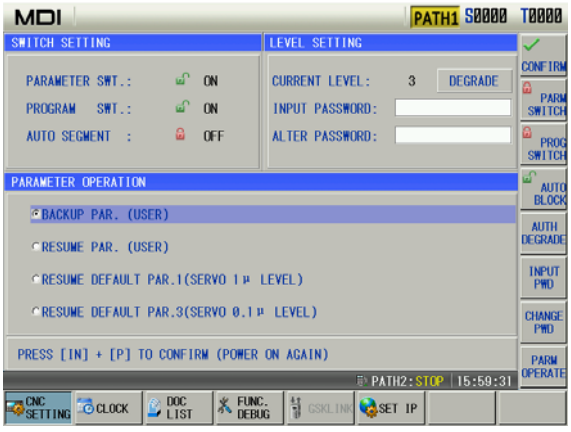
and all often used parameters are removed.

12.3.3.4 Altering often used parameter

Parameters can be altered directly in the often used parameter page. Its alteration method is the same that of state parameter and data parameter. Its value is also refreshed in the state parameter or data parameter page.

12.3.4 Data Copy and Resume

The parameter data (state parameter, data parameter) can be backup (stored) and recovered (read). Programs can be stored in the CNC when the backup or recovery is executed. Press  and the display is shown below:



1) Option explanation:

Backup current parameters (user): the user to backup the CNC data (store).

Resume backup parameters(user): the user resumes the backup data (read).

Resume parameter (servo 1u level): the user reads the original parameter data matched with the servo drive with the precision 1um.

Resume parameter (servo 0.1u level): the user reads the original parameter data matched with the servo drive with the precision 0.1um.

Note: When the level is the 2nd grade, the backuping current parameters (user) and resuming backup parameters (user) become backuping current parameters (machine manufacturer) and resuming backup parameters (machine manufacturer), which is used to backup and resume the manufacturer's data.

2) Operation methods:

a) Open the parameter switch;

b) Press 

to enter the MDI mode, move the cursor the required item;

c) Press 

, and the system prompts whether to confirm the current operation;

d) Press  , and the system prompts the backup or resume is done successfully. When the recovery is done, the system prompts power-on again.

Note: Don't turn off the system and execute other operations when the backup or resuming is executed.

12.4 Macro variable note

Macro variable note display is mainly administrated for the macro note and macro note file. The administration of the macro note file includes the leading-in, leading-out, rename, overloading and remark, etc. of the macro note file.

12.4.1 Macro variable note page

Press the 

soft button to enter its corresponding customization page ,after entering the cutter compensation page by controlling the 

button, The page is shown below Fig.1:

Press the 

button to enter the macro variable note list after entering the macro variable note page, refer to the following fig.2.

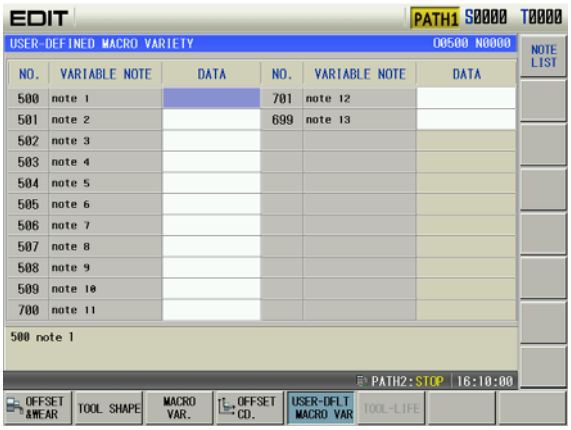


Fig.1

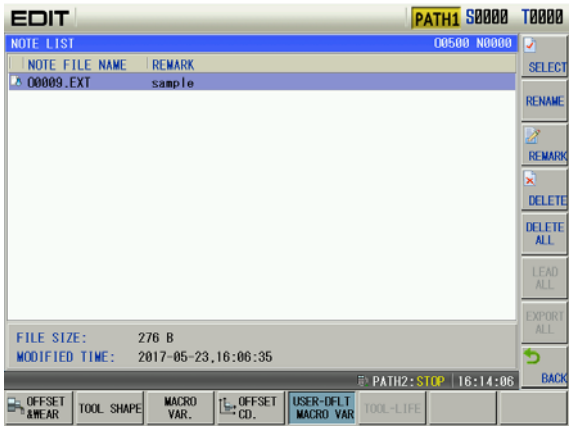


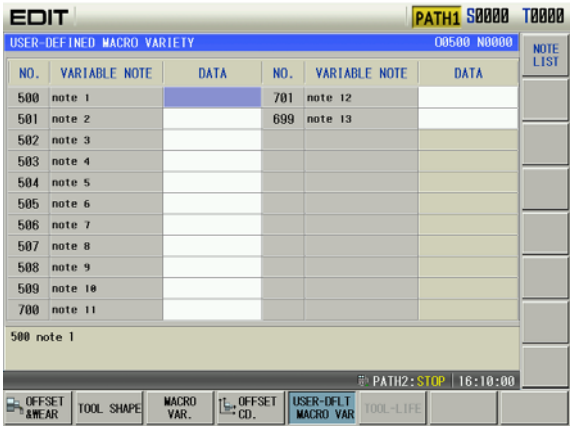
Fig. 2

Note: The overloaded file data can be set on the "Custom macro variable" page, only 4 and above operation authorities can be performed.

12.4.2 Macro Variable Note File

1、File content format

1）Create a TXT file，the surfix is renamed as .EXT
2）The file content format is as follows：
0, Remark the display content.
Variable number, custom content \t Note content.
For example: Set up a O0009.EXT file, its content is as follows:
0, Note file example
500, Note 1\t Detailed note 1
501, Note 2\t Detailed note 2
……
701, Note 12\t Detailed note 12
699, Note 13\t Detailed note 13



2、File leading-in/out operation

Leading-in operation

1）Create a new file MACROEXT under the root list of the U-disk;
2) The performed system custom file is delivered to the folder;
3) Set the system authority as 2-level or above, and then insert to U-disk;
4) The *.EXT file under the U disk MACROEXT list can be entirely lead out to the system note list by LEAD ALL soft button.

Leading-out operation

1) Insert an U-disk;
2) The overall files in the system note list are lead out to the U disk MACROEXT folder by EXPORT ALL soft button; if the note file name is same as the system list under the U disk MACROEXT; the same name note file will be covered under the systme root list.

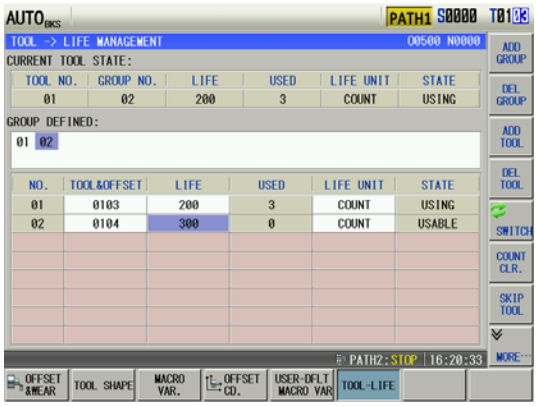
CHAPTER 13 TOOL LIFE MANAGEMENT

13.1 Tool life management display window

Press repetitively 

into the tool life management display window 

, The current window displays the life management data of current used tool life management and defined tool groups. The window is shown below:



Current tool state: display the current used tool life management data;

Tool number: current used tool and tool compensation number;

Group: the group where the tool is;

Life: tool life data, the specified value can be time or number of times according to the different count N value; press 

to record the time or clear the times.

Used: used tool life data.

Life unit: count unit of tool life, N1 is the used time (unit: minute), N0 is the used times (unit: times) . Press 

, 

to change the count unit of the tool.

State: display the tool state(0—not be used,1—being used ,2—used,3—skip).Press 

to change the tool state to skip when the tool is not used in the current window.

Defined group: It only display defined group number and the undefined group number is not displayed. The displayed group back lighted means that all tool life in this group is over.

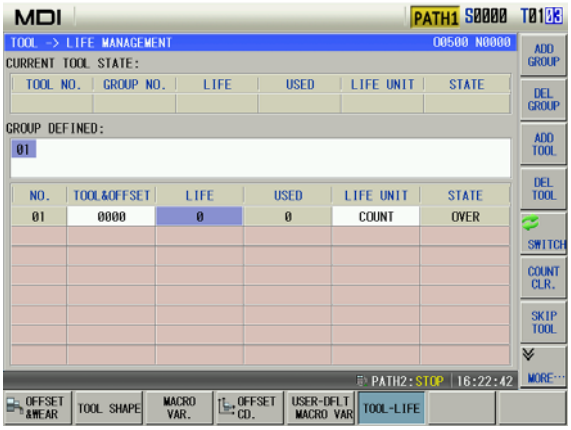
13.2 Creating and displaying tool group number

In tool group state display window, press 

, group number, 

and the system display the tool group life data, and when the group does not exist, it is defined to the new group number(the parameter switch is ON in MDI mode).

1) Note: After the new group is defined, GSK980TTi will automatically define the first tool, for example, the new defined group number is 01, the display is as follows:



2) Press SWITCH to move the cursor to Defined Group Number;

3) Press ← or → to select the group number in Defined Group Number, and to gradually display content of each group number.

13.3 Definition of tool life data

There are two ways to set the tool life data: 1) compile NC programs and run the program setting.

13.3.1 Compile NC programs and run program setting

Command format: G10 L3
Command function: set to be the tool life manage data input mode
Command format: G11
Command function: cancel the tool life management data input mode

Program	Meaning	Remark
O0020 (O0020)		T_: tool and tool offset number;
G10 L3;	Set to be the tool life management data input mod	
P01;	Tool group number, too group number setting range(1～32)	N_: tool life count mode, N0 is the used time(minute) to count tool life and N1 is the used number of times to count the tool life(unit: min)
T0101 L500 N0;	Tool number, life, mode(number of times) setting	
T0201 L600 N1;	Tool number, life and mode(min) setting	
P02;	Another tool group number	
T0303 L200 N0;		L_: tool life data, the specified value can be time or number of times according to the different count N value
T0304 L300 N0;		
G11;	Cancel the tool life management data input mode	
M30;		

Note 1:The tool group numbers specified by P may not be the continuous, but it is better to gradually increase the sequence number to easily search the tool group number.

Note 2:The tool life is 0 when the life data L_ is omitted, and the tool mode is 0 (minute) when the specified mode N_ is omitted, at the time, the system only counts and doest not alarm for the output.

Note 3:Other commands between G10 L3 and G11 are ignored.

Note 4:Running the tool life preset program (such as O0020) completely clear all previous life data and preset the life data according to the program requirements.

Note 5:Prevent the life data from being modified manually when the part programs are running until the run state stops (except for running the tool life preset programs).

Note 6:All tool life data is stored when power-down.

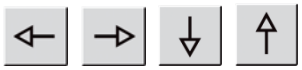
13.3.2 Input tool life management data in the tool life management window

Input directly the tool life management data in tool compensation- >tool life window in MDI mode, parameter switch ON and 3-level operation limit.

1) alter data:

In tool compensation -> tool life window, press 

to move the tool to the life position, input directly data, such as 500, press 

to confirm the input. Press  to complete data input (tool offset, life) .

2) Inserting data:

Insert any serial number In tool compensation -> tool life window, press 

—> [01～08] ——> 

, insert a new line.

3) Delete data:

a）Delete data in all groups. In tool compensation-> tool life window, press 

to delete all defined data.

b）Delete data in any group: Press 

—>input serial number [02] —> 

.

c）Delete any serial number in the current page(the parameter switch is ON): Press 

—> input serial number [04] —> 

.

13.4 Tool life function use

Command format:

Txx99: end of current used tool group, start the tool and execute the life management in XX group.

Txx88: cancel the tool offset in XX group

Two examples are as follows:

Application example:

```
O0000(O0000)
    …
  T0199;      End of previous tool group, and start the tool in 01 group
    …
  T0188;      Cancel tool offset in 01 group(current used tool offset)
    …
  T0508;      Use No.05 tool and 08 tool offset without life management
    …
  T0500;      Cancel No.05 tool offset
    …
  T0299;      End of No.05 tool and start the tools in 02 group
    …
  T0199;      End of tools in No.02 group, start tools in No.02 group, and start the next
                tool when there are many tools in No.01 group.
    …
```

13.5 Tool life count

When the count result is for the used life value being more than or equal to the setting value of the life data, the next tool group number selects the standby tool in the commanded selection tool group and the new selected tool will be counted, the count will be continuously executed and the system alarms to output to PLC when all tool life in the tool groups reaches and there is no standby tools. Executing the counting in MDI mode is determined by No.002 Bit3 (MDITL).

there are two ways to count tool life: times and minute.

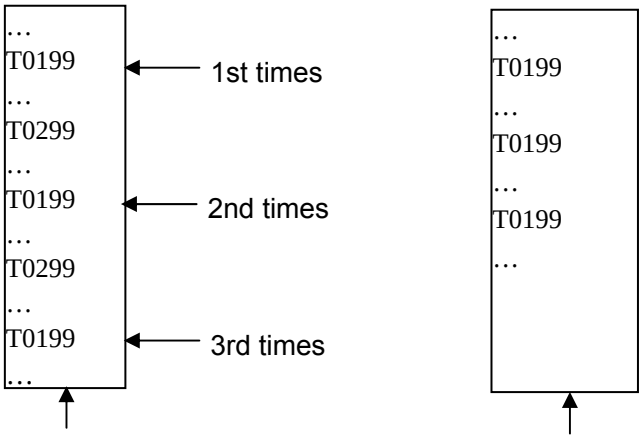
13.5.1 Time count

Use the time (minute) to count the tool life in cutting feed mode(such as G01, G02, G03, G32, G33, G34 and so on), and do not count it in G00, G04, single block stop, machine lock, auxiliary lock, dry run and so on.

13.5.2 Number of times count

There are two methods about the number of times count which is determined by No.002 Bit2 (LIFC). Execute the tool select (Txx99) to change the tool number, execute the count in the cutting feed mode (except for machine lock, auxiliary lock, dry run state). The count is not executed when the tool number is changed and the system is not in the cutting feed mode.

Application example:



Use 3 times in 01 tool group use once in 01 tool group

LIFC＝1 Tool life management count 2

The tool group adds when the machining program runs to M30(M99), when the system resets halfway, the number of time does not add, the count is not executed in machine lock, auxiliary lock and dry run mode.

CHAPTER 14 COMMUNICATION

14.1 GSK980TTi Communication Software GSKComm

GSKComm is an allocated project administrator. It can execute the upload/download between PC and CNC, conveniently transmit batch with easy operation, high communication efficiency and reliability.

The followings introduce communication transmission and setting between GSK980TTi and GSKComm. Refer to GSKComm explanations about GSKComm communication software functions.

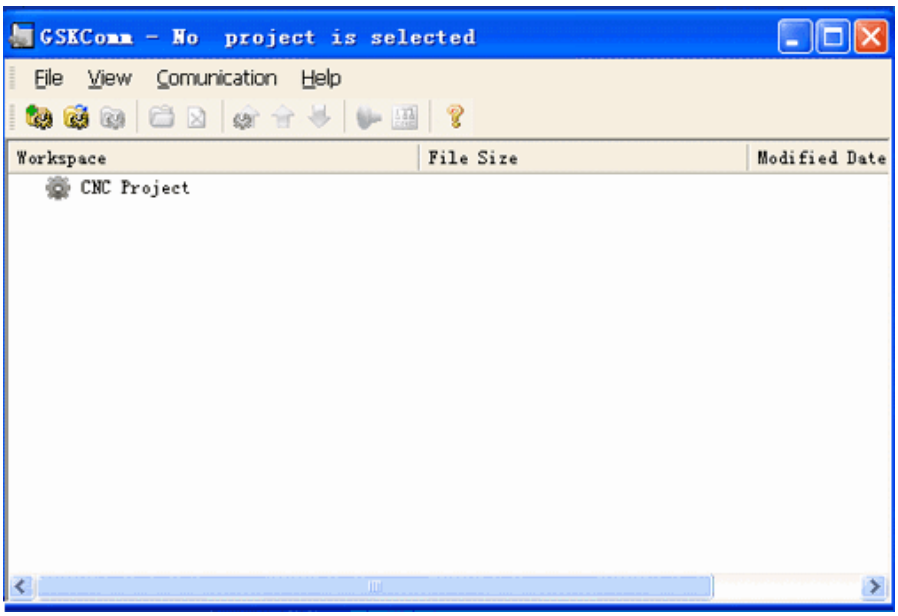
14.1.1 GSKComm's System (PC) Requirements

Hardware: general PC with RS232 serial port, serial communication cable (3-wire).

Operation system: Microsoft Windows 98/2000/XP/2003.

14.1.2 Software Interface

GSKComm software takes an administrator as a unit to complete management. When it run firstly, it does not upload project files, at the moment, the user should create an project program or upload project to execute the communication operation. The display is shown below after the an project is created:



At the mode, the user can receive the files from the CNC to the current project or send files in the current project to the CNC.

14.1.3 Receiving Files (CNC→PC)

①Click the option [Receive files from CNC] in [Communication] and GSKComm can receive files from the CNC after the current project is selected as follows Fig.1;

②After [Receive Files] is clicked, a dialog box about receiving files from the CNC, the required files from the CNC including part programs, tool compensation (TOFF. CMP), pitch compensation (WOFF.WMP), parameters (PARAM.PAR) as follows Fig.2;

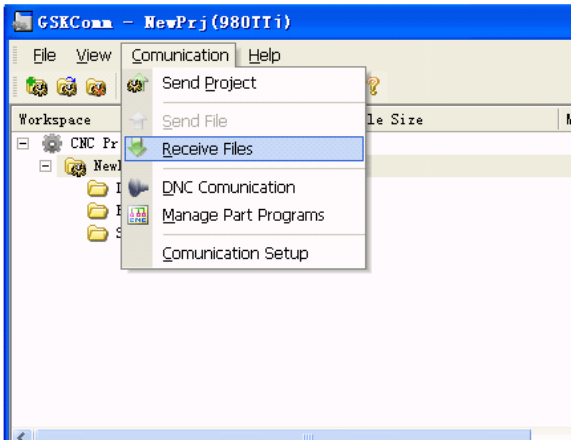


Fig.1

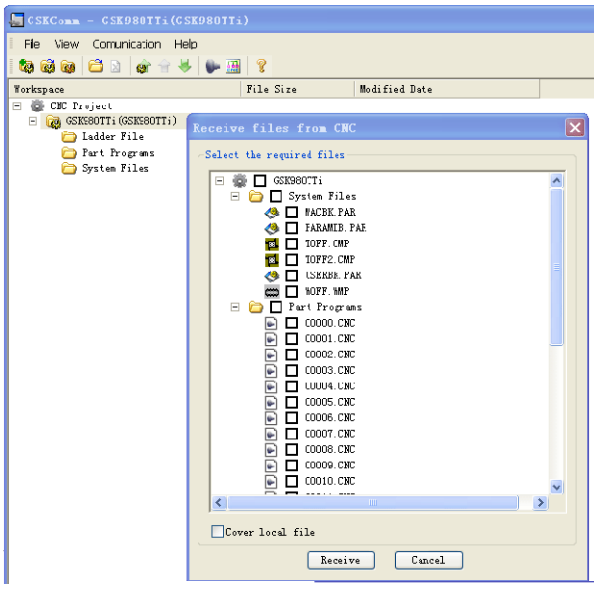


Fig.2

③After the user selects the required file, [Start Receiving] is pressed and the system pops up a path selection dialog box to save the received files as follows Fig.3;

④Select the save path, start to receive the file from CNC to PC by pressing [i.e. Confirm] as follows:, as follows Fig.4;

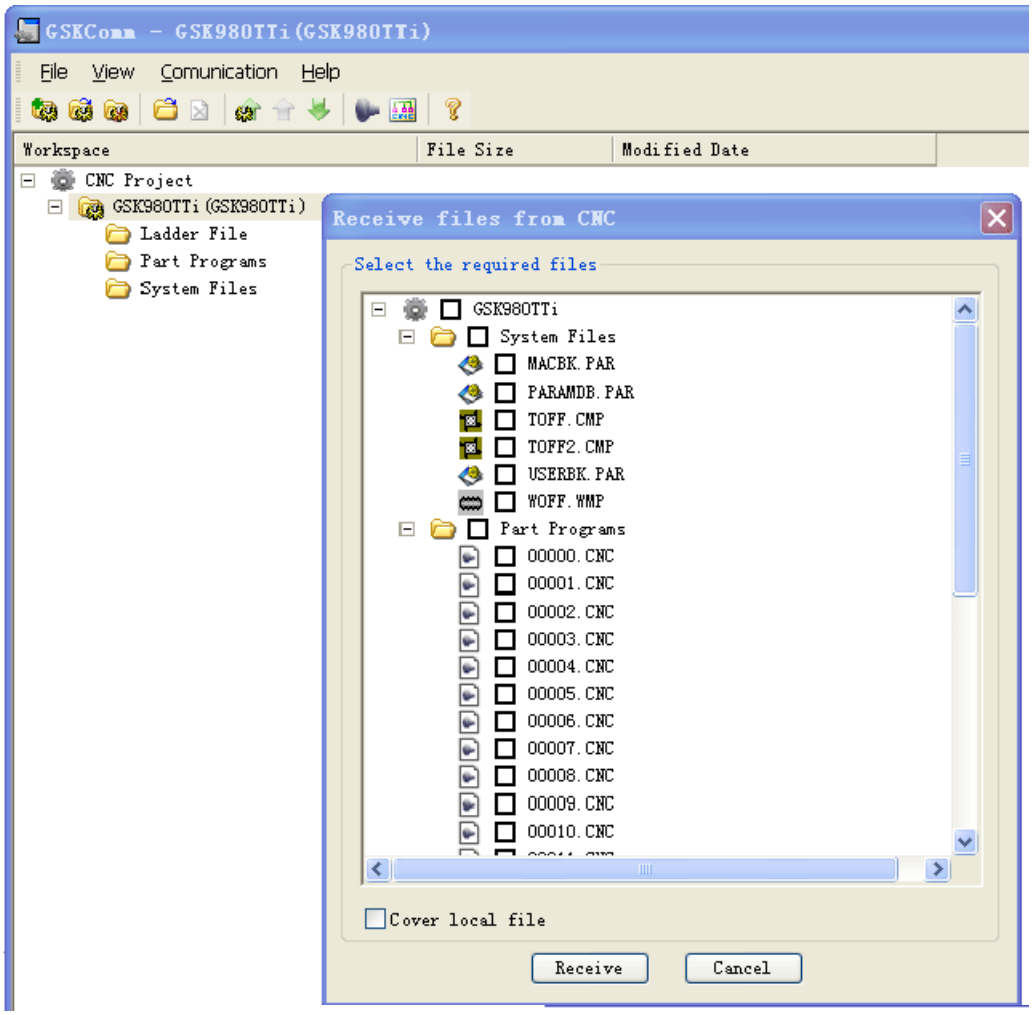


Fig.3

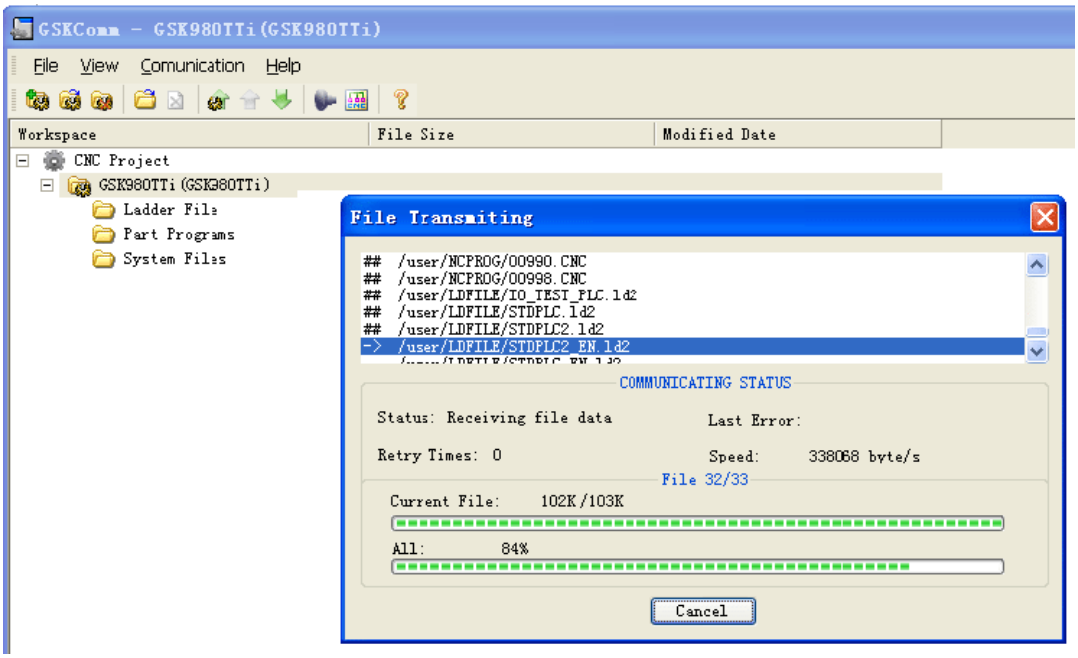
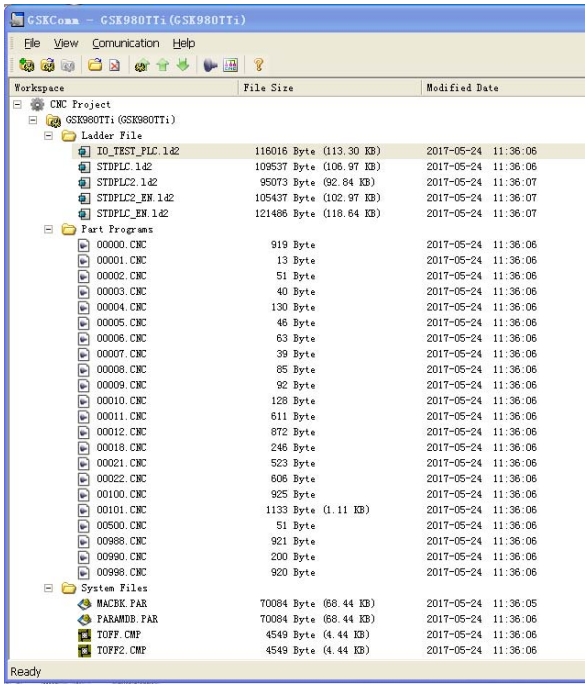


Fig.4

After the receiving is done, the system displays below:



14.1.4 Sending Files (PC→CNC)

14.1.4.1 Sending multiple files

After the current project file is selected, the user clicks [Send projects to CNC] in the menu [Communication], and so the receiving file from CNC is done.

①A dialog box to send files to CNC is popped up as follows Fig.1;

②then press [start sending], and the file is started to send to the CNC as follows Fig.2;

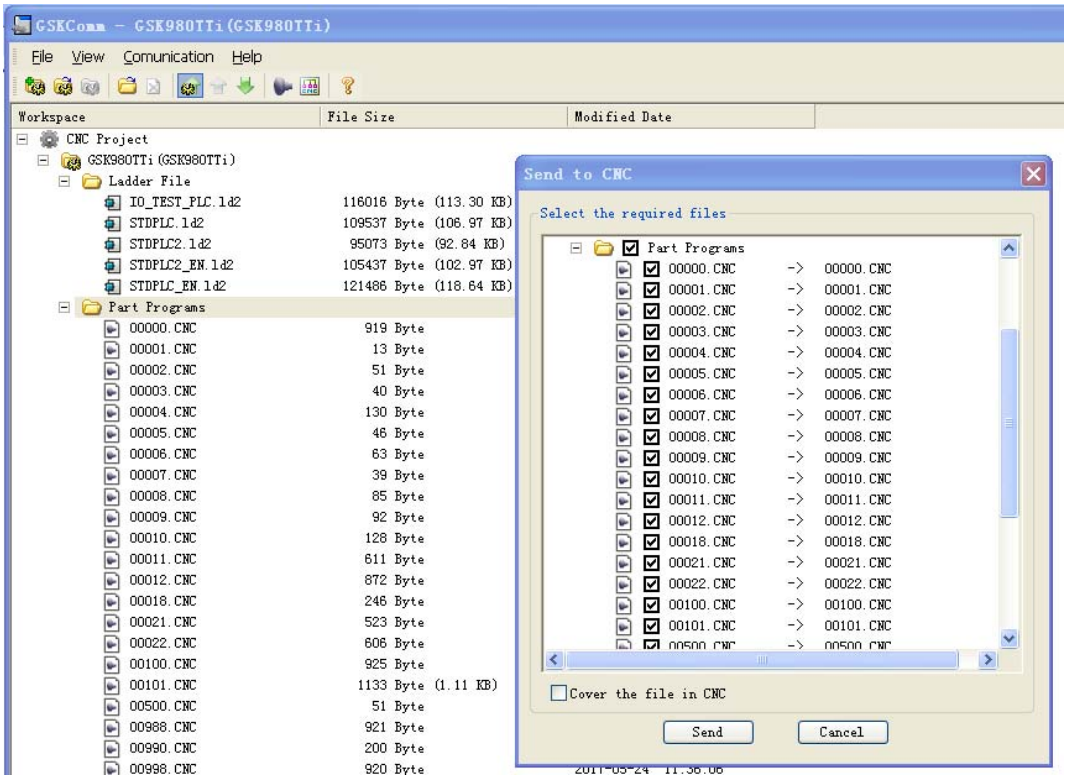


Fig.1

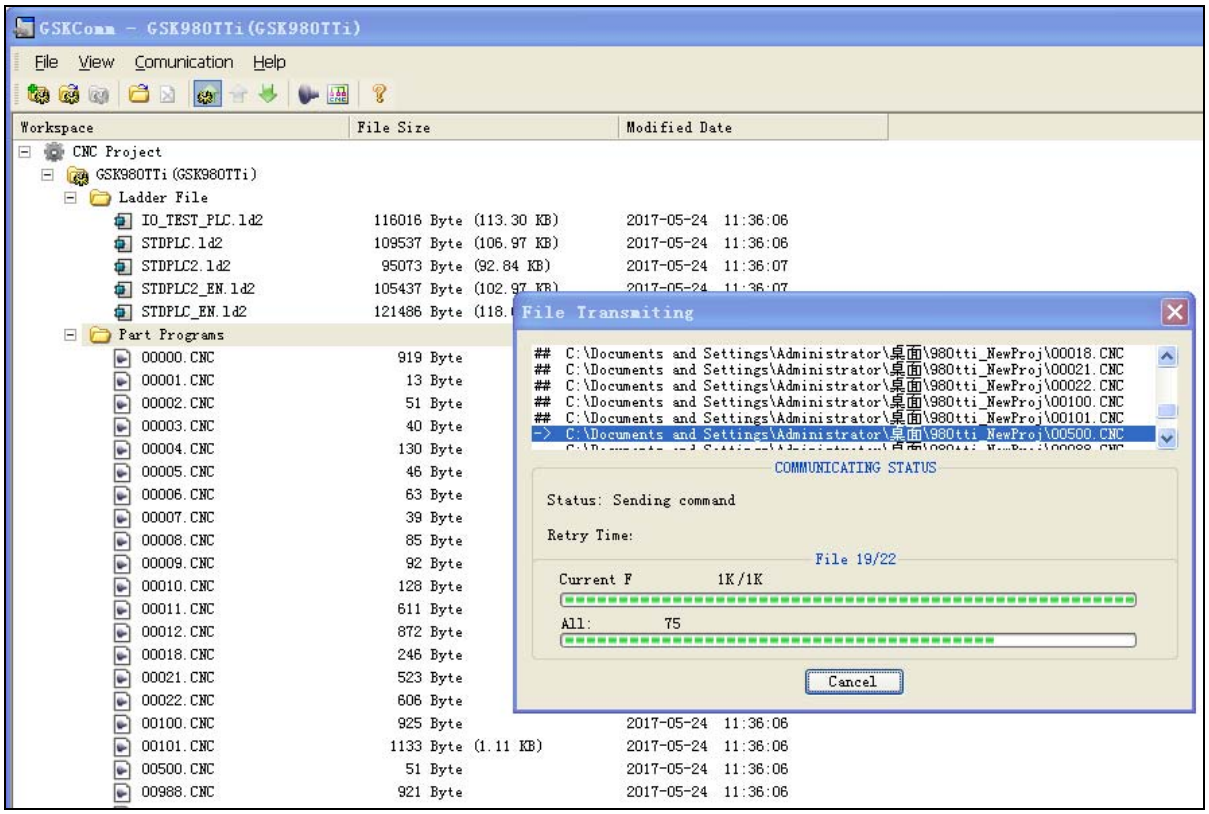


Fig.2

③When a file name in the CNC is the same that of the transmitting, a dialog box is popped up to select the covering, skipping the file or cancelling transmission as follows Fig.3;

④When the part program name is the same that of the transmitted file, the covering cannot be

done, otherwise, the system pops up the wrong prompt as follows Fig.4;

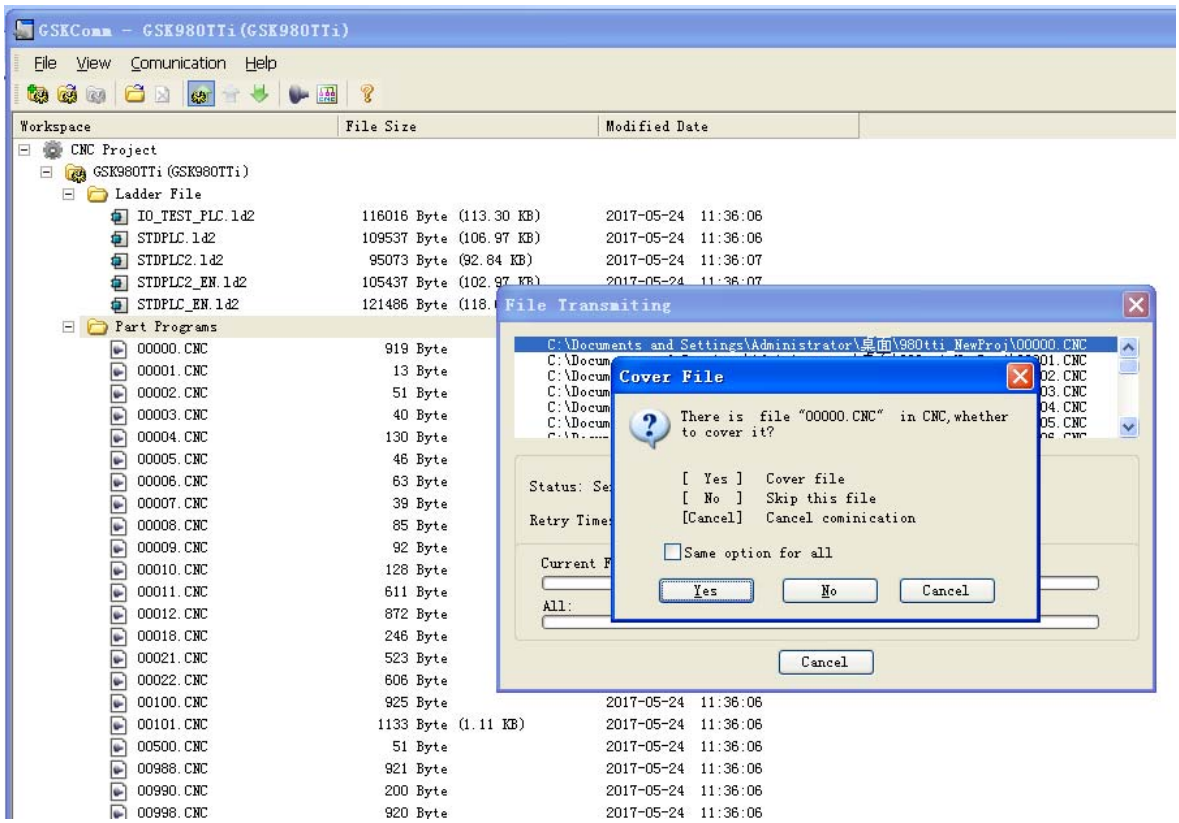


Fig.3

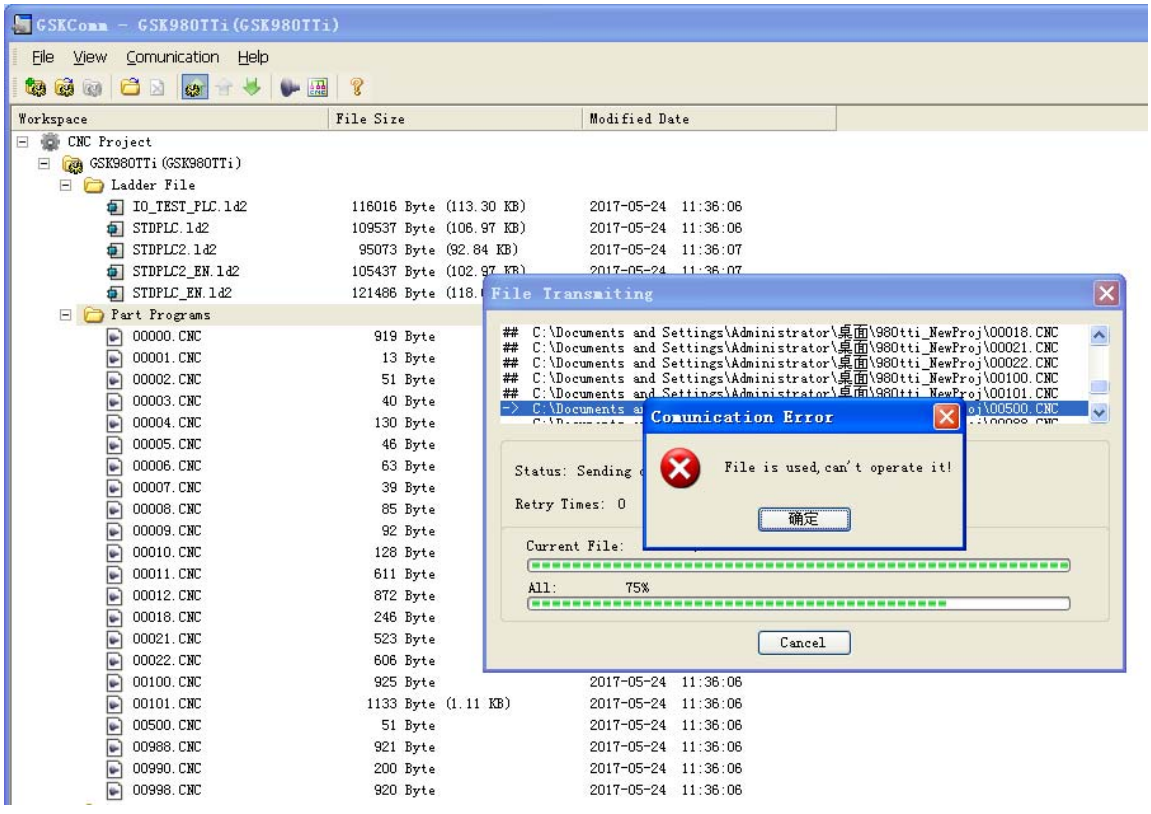


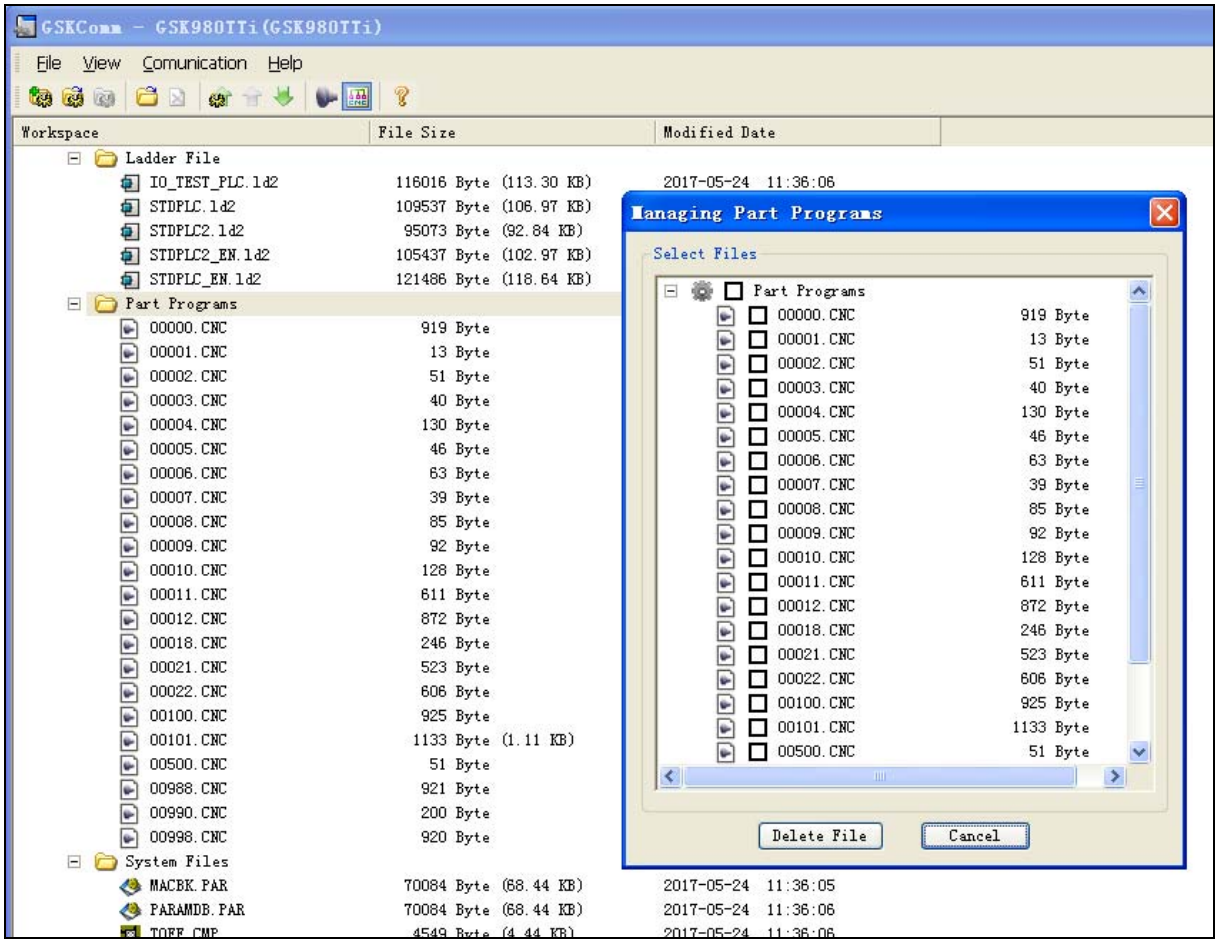
Fig.4

14.1.4.2 Sending single files

Hereby, the user can select the single required receiving files from CNC including part programs, tool compensation (TOFF.CMP), pitch compensation (WOFF.WMP), parameters (PARA.PAR),PLC. The user clicks [Send files to CNC] in the menu [Communication], the system pops up a dialog box [Save CNC file name],after change the file name, click [confirm] to save.

14.1.5 Part Program Management

Select the current project file, then click [Manage Part Programs] in the menu [Communication] , and the part programs in the CNC are managed. The system pops up a dialog box to manage part programs in the CNC as follows:



Press [Delete File] and the selected part programs are deleted in the CNC after the required programs are selected.

Note: The program being used in the CNC cannot be deleted.

14.2 Preparatory Before Communication

14.2.1 RS-232 serial communication

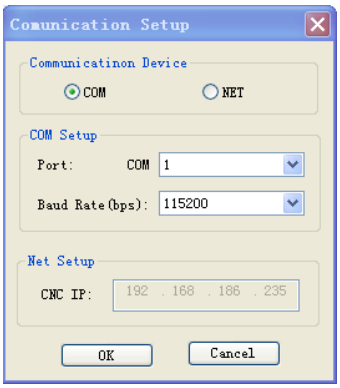
14.2.1.1 Communication Cable Connection

DB9 male plug is inserted into CN51 communication interface of the CNC, DB9 female plug is inserted in to DB9-male serial port (COM0 or COM1) of PC. Connect the communication cable when PC and CNC are turned off.

14.2.1.2 Communication Setting Baud Rate

Setting the baud rate of communication ensures that the baud rate of PC is the same that of the CNC communication.

The communication baud of the serial port is set by No.044;in GSKComm,click [Communication Setup] in the menu [Communication] and the system pops up a dialog box as follows:



Set the serial port and baud rate. The baud rate should be not less than 4800 when the data is transmitted between CNC and PC, the factory setting is :115200.

14.2.2 Network Connection

14.2.2.1 Communication Cable Connection

Use the common network cable, one terminal connects with the internet access of GSK980TTi, the other one directly connects the Computer or Router access.

14.2.2.2 Communication IP Setting

Set the IP address of communication, so that the IP address between PC and CNC communication are consistant.

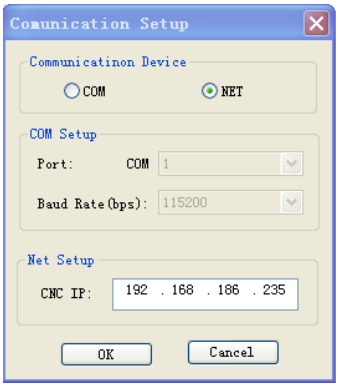
The IP setting of the GSK980TTi is as follows: 

—> 

,and then set the IP address and sub-net mask, etc. after entering the system IP setting page, refer to the following figure:

Wherein, the IP address can not be identical with the other PC computer in the network or CNC. The subnet mask should be same with the PC computer or the subnet mask of the router; it is not necessary to alter the default subnet.

In the GSKComm, click the [Communication Setting] in the [Communication] menu, its corresponding dialog frame is shown below, refer to the following figure:



Network selection of communication equipment, write the IP address set to CNC in the network setting.

Note 1: Stop machining when the machining is being done to get the stable communication.

Note 2: Must not turn off during the data transmission, otherwise, it causes the mistaken data transmission.

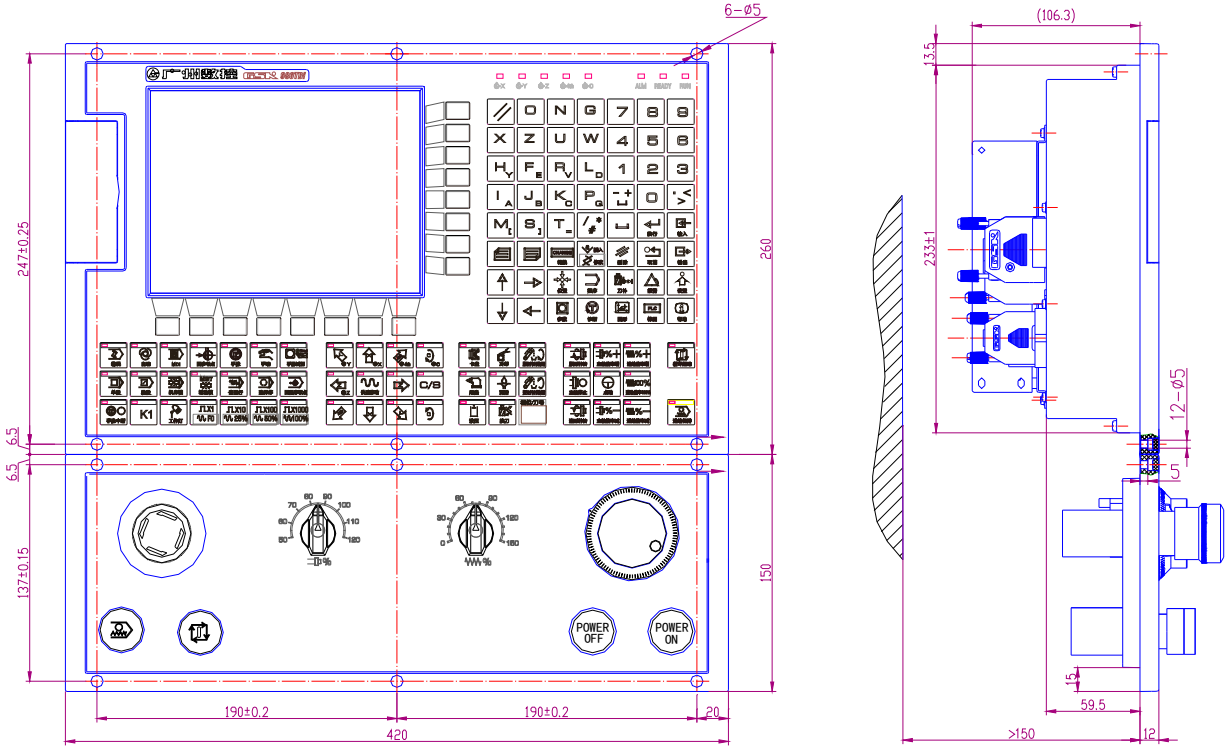
Ⅲ Installation & Connection

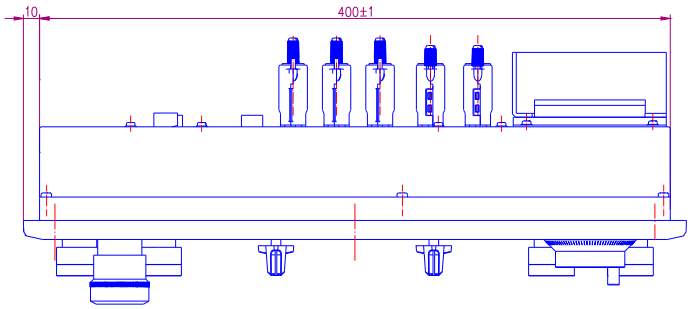
CHAPTER 1 INSTALLATION LAYOUT

1.1 GSK980TTi contour dimension

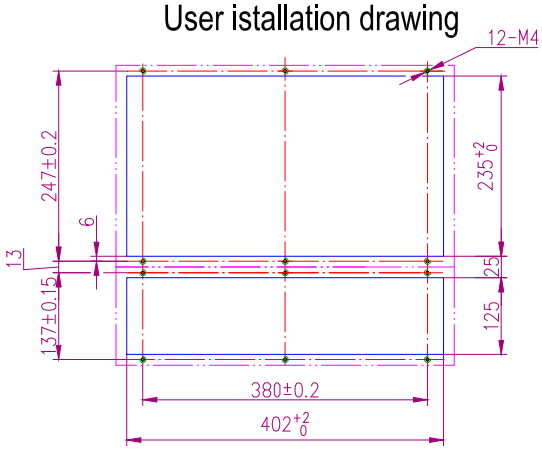
GSK980TTi to distinguish between horizontal panel and vertical panel. Their electrical interface and application functions are identical, the difference in the contour dimension.

1) GSK980TTi horizontal panel contour dimension



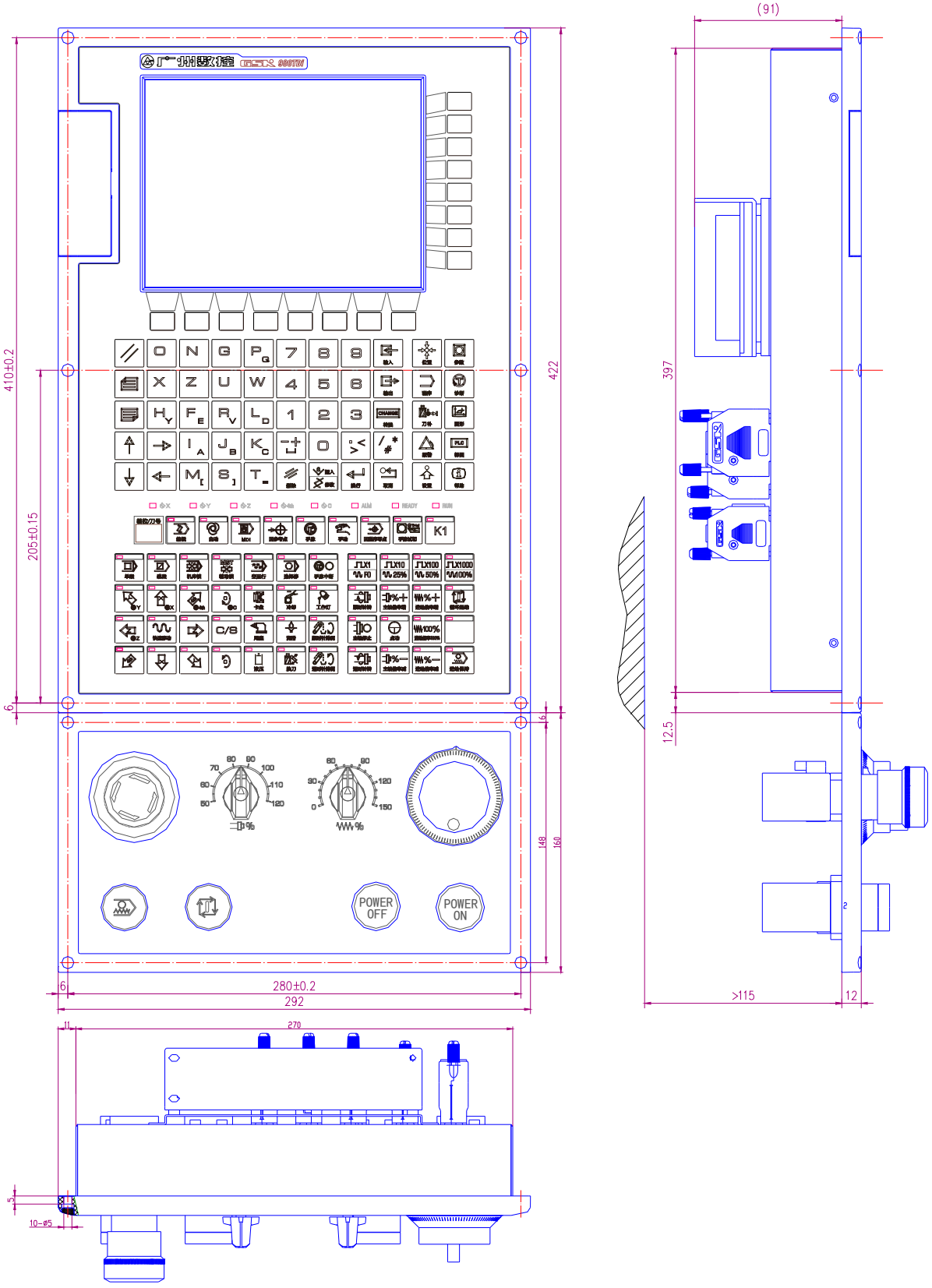


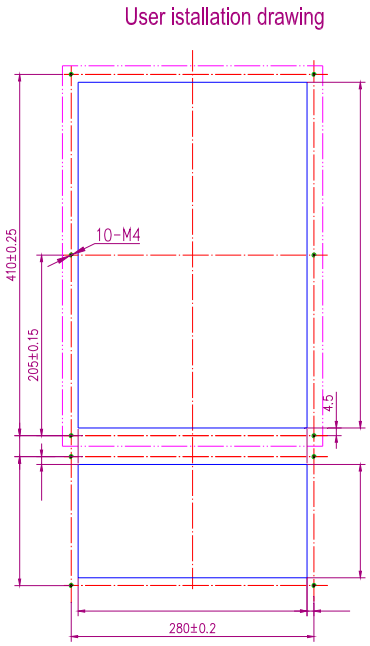
User istallation drawing



III Connection

2) GSK980TTi vertical panel contour dimension





1.2 GK980TTi System Connection

1.2.1 GK980TTi rear cover interface layout

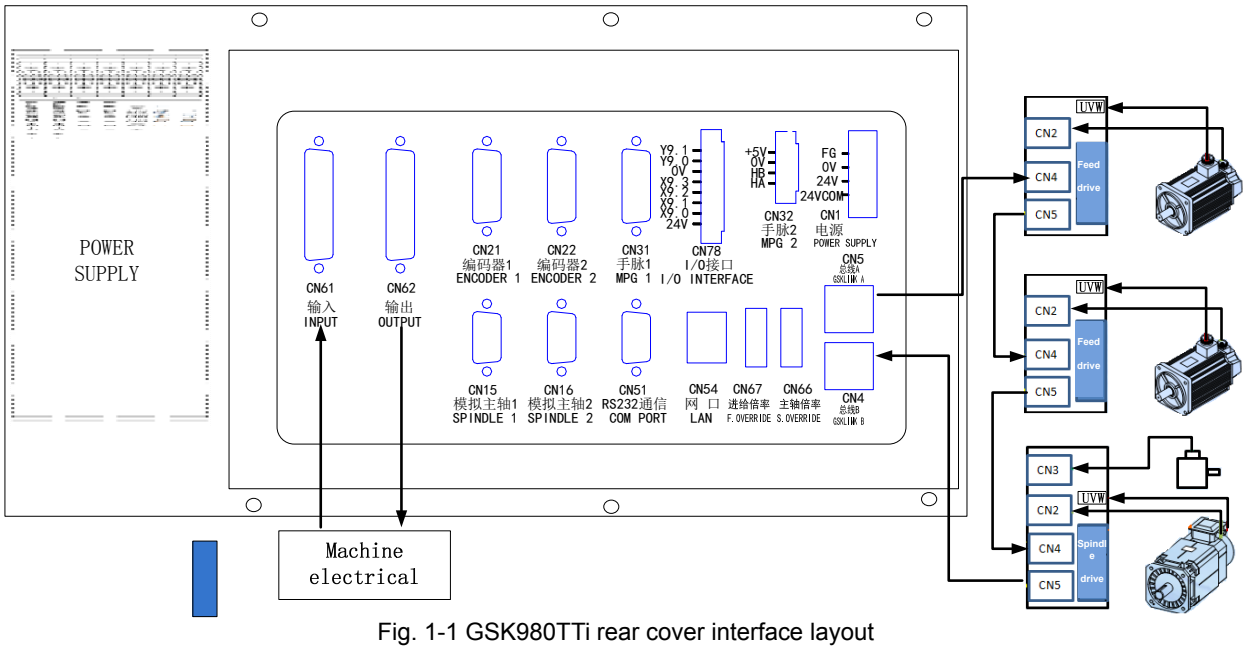


Fig. 1-1 GSK980TTi rear cover interface layout

1.2.2 Interface explanation

- CN1: power supply interface
- CN4, CN5:GSKLINK bus connection interface B,A are connected with the servo unit and IO unit
- CN15:Analogy spindle 1,male socket of 9-cord in type D,connect with analogy spindle 1
- CN16:Analogy spindle 2,male socket of 9-cord in type D,connect with analogy spindle 2
- CN21: encoder, pin15 D male,connect with spindle encoder 1

- CN22: encoder, pin15 D male,connect with spindle encoder 2
- CN31: MPG, pin26 D male, connect with MPG
- CN51: communication, pin9 D female, connect PC RS232 interface
- CN54:Ethernet interface, plug are connected with the PC to transmit data
- CN61: input, pin44 D male, connect with machine input
- CN62: output, pin44 D female, connect with machine output
- CN66:the spindle override input signal, 4 input interfaces are connected with the additional panel
- CN67:the feedrate override input signal, 5 input interfaces are connected with the additional panel
- CN78:4 input/2 output interfaces, 8 connection terminals, machine input,output

CHAPTER 2 DEFINITION & CONNECTION OF INTERFACE SIGNALS

2.1 GSKLink Bus Interface

Bus interfaces of GSK980TTi are CN4 and CN5 (GSKLinkB and GSKLinkA), which is connected with the GSKLink bus communication function, the feedrate servo drive unit, spindle drive unit and extended I/O communication. GSKLink bus communication uses a loop control, and therefore, CN4 AND CN5 interfaces should be used together.

GSKLINK bus connection is shown below:

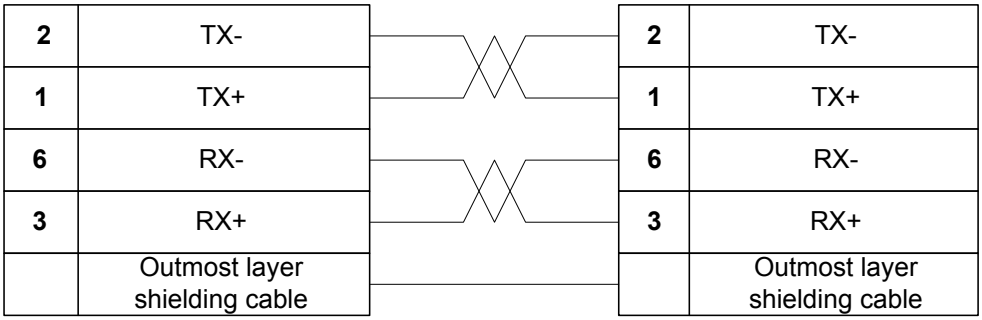


Fig. 2-1 GSKLink communication connection

2.2 Being Connected with Spindle Encoder

2.2.1 Spindle encoder interface definition

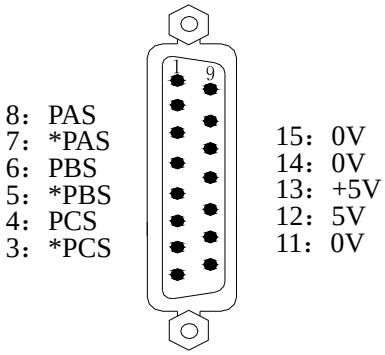


Fig.2-2 CN21 encoder interface
(15-core D type male socket)

Name	Explanation
*PAS/PAS	Encoder A phase pulse
*PBS/PBS	Encoder B phase pulse
*PCS/PCS	Encoder C phase pulse

2.2.2 Being connected with spindle encoder interface

GSK980TTi is connected with spindle encoder shown in Fig. 2-3, and it uses twisted pair cables. (exemplified by CHANGCHUN YIGUANG ZLF-12-102.4BM-C05D encoder):

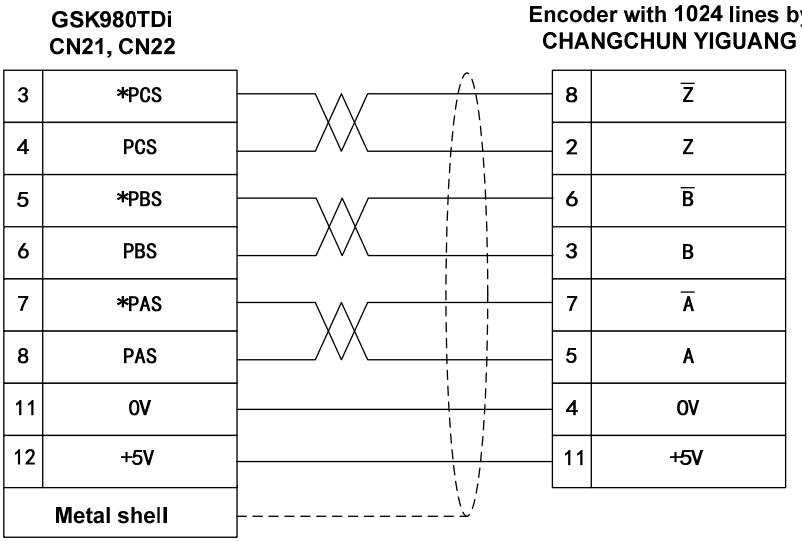


Fig. 2-3 GSK980TTi is connected with the encoder

2.3 Being Connected with MPG (Manual Pulse Generator)

2.3.1 MPG interface definition

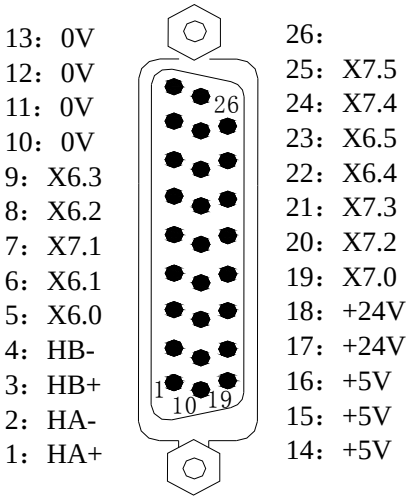
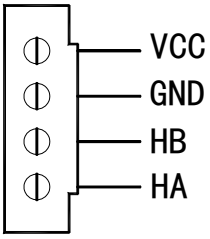


Fig.2-4 CN31 MPG interface(26-core DB type male socket)

Signal	Explanation
HA+、HA-	MPG A phase signal
HB+、HB-	MPG B phase signal
+24V	DC power supply
+5V、0V	

Standard PLC Definition	
Address	**Function**
X6.0	X MPG axis selection
X6.1	Y MPG axis selection
X6.2	Z MPG axis selection
X7.0	4TH_axis selection signal
X7.1	5TH_axis selection signal
X6.3	Increment0.001
X6.4	Increment0.01
X6.5	Increment0.1



Signal	Explanation
HA	MPG A phase signal input
HB	MPG B phase signal input
GND	0V
VCC	5V

Fig. 2-5 CN32 MPG interface
(4-point wiring terminal)

2.3.2 GSK980TTi connected with MPG

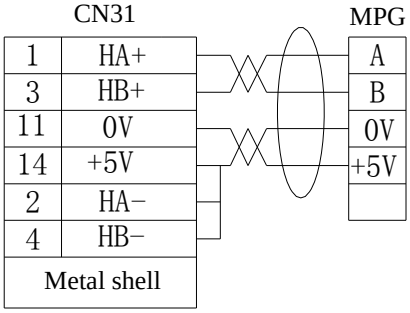


Fig. 2-6 CN31 connects with the single-port MPG input

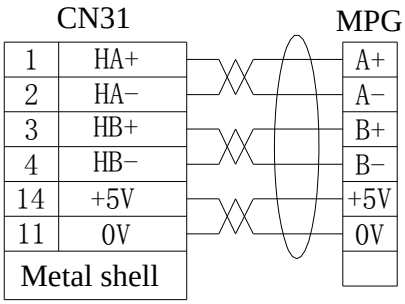


Fig. 2-7 CN31 connects with the difference MPG input

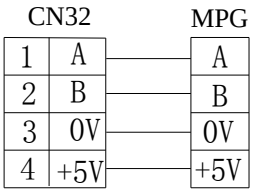


Fig. 2-8 CN32 connects with MPG

Note: Refer to the Fig. 3-13 and Fig. 3-14 for the connection of CN31 interface and MPG cabinet.

2.4 Analog Spindle Interface

2.4.1 Analog spindle interface definition

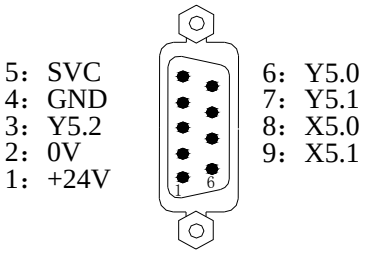


Fig.2-9 CN15 analogy spindle interface (male socket of 9-cord in type D)

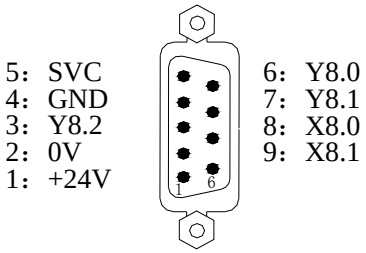


Fig.2-10 CN16 analogy spindle interface (male socket of 9-cord in type D)

Signal	Explanation
SVC	0~10V Analog voltage
GND	Signal grounding

Standard PLC Definition	
Address	**Function**
X5.0	Spindle 1 alarm signal
X5.1	
Y5.0	series spindle 1 rotation signal CW(positive)
Y5.1	series spindle 1 rotation signal CCW(negative)
Y5.2	
X8.0	Spindle 2 alarm signal
X8.1	
Y8.0	series spindle 2 rotation signal CW(positive)
Y8.1	series spindle 2 rotation signal CCW(negative)
Y8.2	

2.4.2 Connection to inverter

GSK980TTi is connected with the inverter shown in Fig. 2-11:

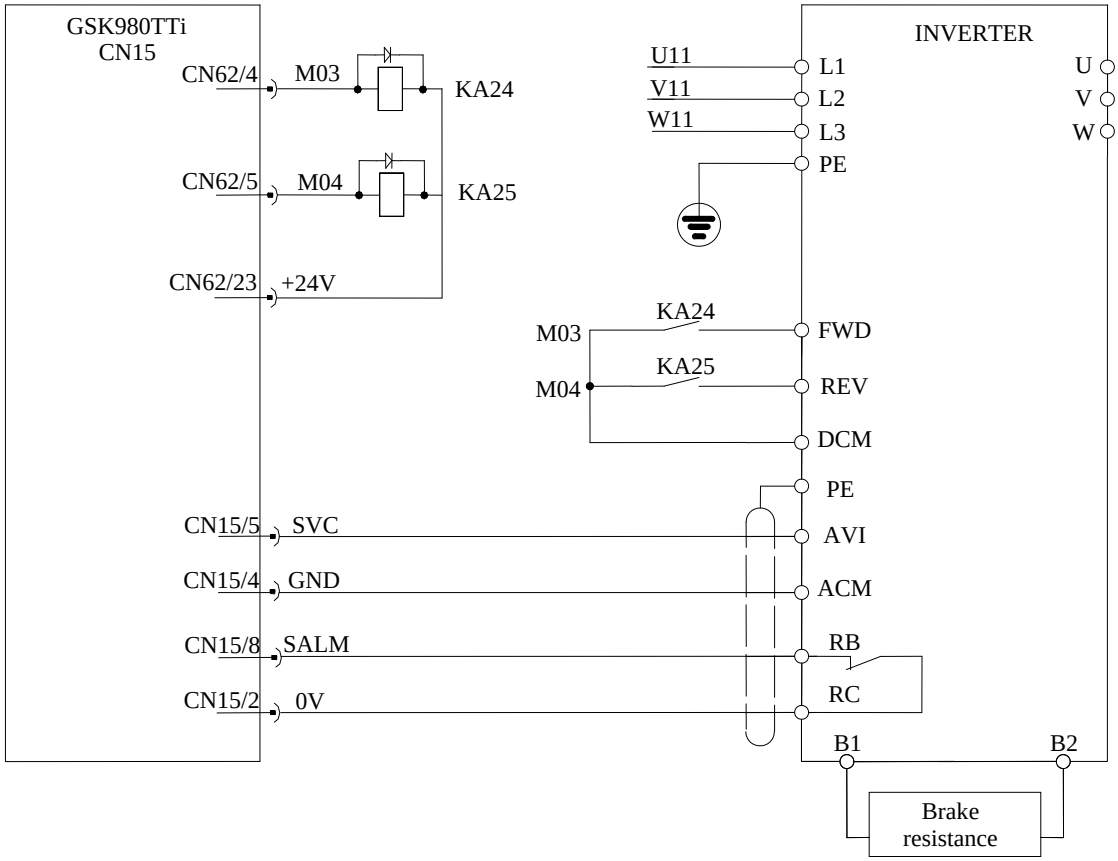
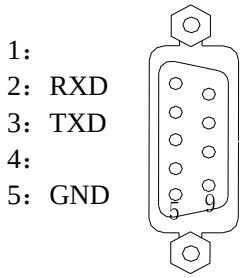


Fig. 2-11 GSK980TTi is connected with the inverter

2.5 Connected with PC

2.5.1 Communication interface definition



Signal	Explanation
RXD	Receiving data
TXD	Transmitting data
GND	Signal grounding

Fig. 2-12 CN51 communication interface (DB9-female)

2.5.2 Communication interface connection

GSK980TTi can perform the communication by CN51 and PC(optional communication software). GSK980TTi is connected with PC shown in Fig 2-13:

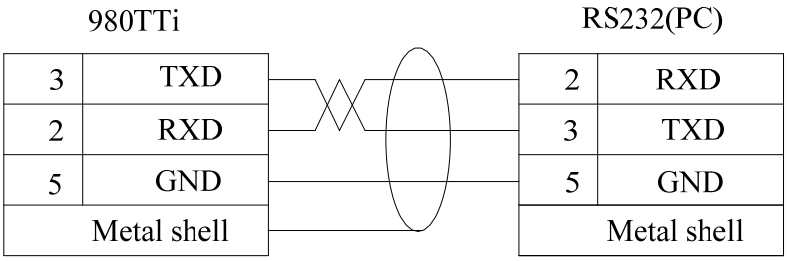


Fig. 2-13 GSK980TTi is connected with PC

2.6 Network interface

Network interfaces of GSKTTi CN54 are standard and their definitions are shown below:

PIN NO.	Signal	PIN NO.	Signal
1	TXDLAN+	9	LINK_LED
2	TXDLAN-	11	LAN_LED
3	RXDLAN+	10、12	VDD33
6	RXDLAN-	13、14	Shell grounding

Note: TXD+ and TXD- are difference signal, as well the **RXD+ and RXD**-, which are required to connect with twisted-pair.

2.7 Power Interface Connection

The power box is applied in GSK980TTi system, which involves 2 groups of voltage: +24V. The connection of power box to GSK980TTi CN1 interface has been done for its delivery from factory, and the user only need to connect it to a 220V AC power in using.

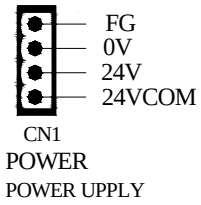


Fig. 2-14 system power interface CN1

24VOM is connected with the output interface CN62 24V. When GSK980TTi's power supply box supplies power supply for an external electron device, the supply box's +V is connected with 24VCOM. When an external power supply is used, its 24V is connected with 24VCOM and its 0V must be connected with the system's supply box's-V.

2.8 I/O Interface Definition

Note!

The I/O function significances of the unlabelled fixed addresses of this GSK980TTi/GSK980TTi-V turning machine CNC system are defined by PLC programs (ladders), and they are defined by the machine builder when matching with a machine, please refer to the manual by the machine builder.

The fixed address I/O function not be marked are described for GSK980TTi PLC. The described contents without other special explanation are also applied to GSK980TTi-V.

2.8.1 CN61 Input signal

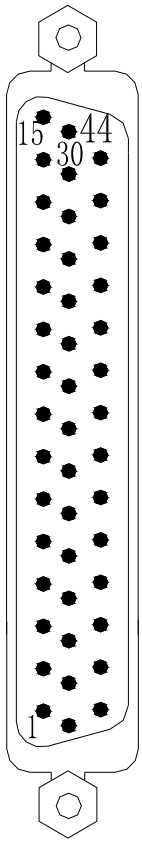


Fig.2-15CN61 inputinterface (male socket of 44-cord in type D)

Pin	Address	Function	Explanation
21～24	0V	0V	Power supply interface
17～20 25～28	Floating	Floating	Floating
1	X0.0		
2	X0.1		
3	X0.2		
4	X0.3	DECX(DEC1)	X deceleration signal(fixed address)
5	X0.4	PRES	Pressure check signal
6	X0.5	ESP	External emergency stop signal (fixed address)
7	X0.6	LMIX1	Path1 X overtravel
8	X0.7	T11	Path1 Tool signal 1
9	X1.0	T21	Path1 Tool signal 2
10	X1.1	T31	Path1 Tool signal 3
11	X1.2	T41	Path1 Tool signal 4
12	X1.3	DECZ(DEC2)	Z deceleration signal(fixed address)
13	X1.4	TCP1	Path1 Tool post clamping signal
14	X1.5	SAGT	Guard door check signal
15	X1.6	DIQP1	Path1 Chuck input signal
16	X1.7	DITW	Tailstock control signal
29	X2.0		
30	X2.1	LMIX2	Path2 X overtravel
31	X2.2	LMIZ2	Path2 Z overtravel
32	X2.3	DECY(DEC3)	Y deceleration signal(fixed address)
33	X2.4	DEC4	4th deceleration signal(fixed address)
34	X2.5	DEC5	5th deceleration signal(fixed address)
35	X2.6	LMIZ1	Path1 Z overtravel
36	X2.7	T12	Path2 Tool signal 1
37	X3.0	T22	Path2 Tool signal 2
38	X3.1	T32	Path2 Tool signal 3
39	X3.2	T42	Path2 Tool signal 4
40	X3.3	TCP2	Path2 Tool post clamping signal
41	X3.4	DEC6/LUBS	6th deceleration signal(fixed address)/lubricating oil detection
42	X3.5	SKIP	G31 skip signal
43	X3.6	DIQP2	Path2 Chuck input signal
44	X3.7		

Note: Partial input interface is the fixed address for some functions, for example: the signal of ESP, DECX, DECZ or SKIP, etc. If this function does not use, it will be redefined in PLC.

2.8.2 CN62 Output signal

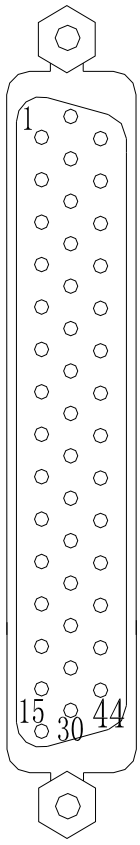


Fig.2-16 CN62output interface(female socket of 44-cord in type D)

Pin	Address	Function	Explanation
17～19 26～28	0V	0V	Power supply interface
20～25	+24V	+24V	Power supply interface
1	Y0.0	COOL1	Path1 Cooling output
2	Y0.1	LUBR	Lubricating output
3	Y0.2	SCLP	Path1 Spindle clamped output
4	Y0.3	SFR1	Path1 Spindle rotation (CCW)
5	Y0.4	SRV1	Path1 Spindle rotation (CW)
6	Y0.5	DOTWJ	Tailstock forward
7	Y0.6	DOTWS	Tailstock backward
8	Y0.7	SPZD1	Path1 Spindle brake
9	Y1.0	S11/M411	Path1 Spindle machine gear output 1
10	Y1.1	S21/M421	Path1 Spindle machine gear output 2
11	Y1.2	S31/M431	Path1 Spindle machine gear output 3
12	Y1.3	S41/M441	Path1 Spindle machine gear output 4
13	Y1.4	DOQPJ1	Path1 Chuck clamping output
14	Y1.5	DOQPS1	Path1 Chuck releasing output
15	Y1.6	TL1+	Path1 Tool post CCW rotation
16	Y1.7	TL1-	Path1 Tool post CW rotation
29	Y2.0	COOL2	Path2 Cooling output
30	Y2.1	HPST	Hydraulic control output
31	Y2.2	WKLP	Work lamp
32	Y2.3	SFR2	Path2 Spindle rotation (CCW)
33	Y2.4	SRV2	Path2 Spindle rotation (CW)
34	Y2.5	CHRO+	Chip cleaner (CCW)
35	Y2.6	CHRO-	Chip cleaner (CW)
36	Y2.7	SPZD2	Path2 Spindle brake
37	Y3.0	S12/M412/ CLPR	Path1 Spindle machine gear output 1/ Three-color lamp -red
38	Y3.1	S22/M422/ CLPG	Path1 Spindle machine gear output 2/ Three-color lamp -green
39	Y3.2	S32/M432/ CLPY	Path1 Spindle machine gear output 3/ Three-color lamp-yellow
40	Y3.3	S42/M442	Path1 Spindle machine gear output 4
41	Y3.4	DOQPJ2	Path2 Chuck clamping output
42	Y3.5	DOQPS2	Path2 Chuck releasing output
43	Y3.6	TL2+	Path2 Tool post CCW rotation
44	Y3.7	TL2-	Path12 Tool post CW rotation

2.9 I/O Function and Connection

2.9.1 CN78 I/O Interface

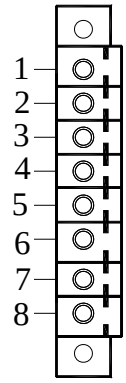


Fig.2-17 CN78 I/O

Pin.No	Address	Function	Explanation
1	Y9.1	SPLAMP	Extended pause lamp
2	Y9.0	STLAMP	Extended start lamp
3	0V	0V	Power supply 0 V
4	X9.3	EXESP	Extended emergency stop signal
5	X9.2	PATH	reserved
6	X9.1	SP	Extended pause
7	X9.0	ST	Extended cycle start
8	+24V	24V	Power supply ＋24V

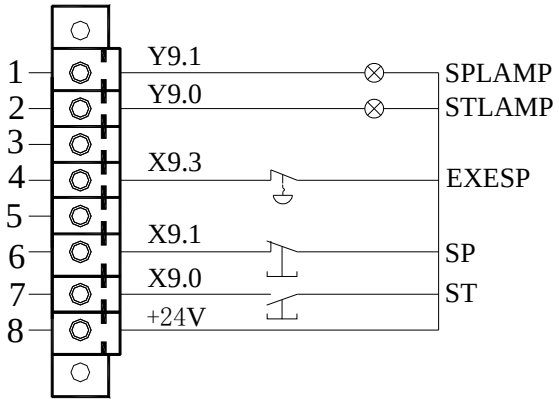


Fig.2-18 I/O interface connection

2.9.2 CN66 External Spindle override

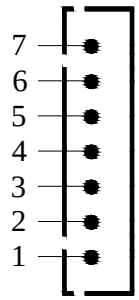


Fig.2-19 CN66

Pin.No	Address	Function	Explanation
1	X29.4	BAUD1_A	Spindle override wave-band switch signal 0
2	X29.6	BAUD1_B	Spindle override wave-band switch signal 2
3	Floating	Floating	Floating
4	+24V	24V	Power supply ＋24V
5	X29.7	BAUD1_E	Spindle override wave-band switch signal 3
6	X29.5	BAUD1_F	Spindle override wave-band switch signal 1
7	INH	BAUD1_INH	INH

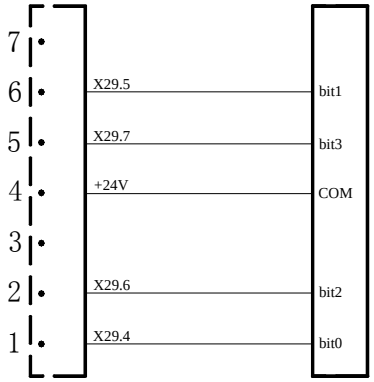


Fig.2-20 external Spindle override interface connection

2.9.3 CN67 External feedrate override

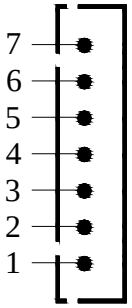


Fig.2-21 CN67

Pin.No	Address	Function	Explanation
1	X29.0	BAUD2_A	Feedrate override wave-band switch signal 0
2	X29.2	BAUD2_B	Feedrate override wave-band switch signal 2
3	Floating	Floating	Floating
4	+24V	24V	Power supply ＋24V
5	X29.3	BAUD2_E	Feedrate override wave-band switch signal 3
6	X29.1	BAUD2_F	Feedrate override wave-band switch signal 1
7	INH	BAUD2_INH	INH

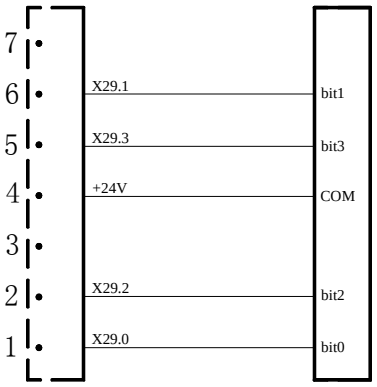


Fig.2-22 external feedrate override interface connection

Note 1: Various functions can be defined to some of the input and output interfaces, and they are indicated by "/"sign in the table above.

Note 2: If output function is active, the output signal is through on to 0V. If output function is inactive, the output signal is cut off by high impedance.

Note 3: If input function is active, the input signal is through on to +24V. If input function is inactive, the input signal is cut off with it.

Note 4: The effectiveness of +24V, COM terminals are equivalent to those of the GSK980TTi/GSK980TTi power box terminals that have the same names.

2.10 Machine panel

GSK980TTi

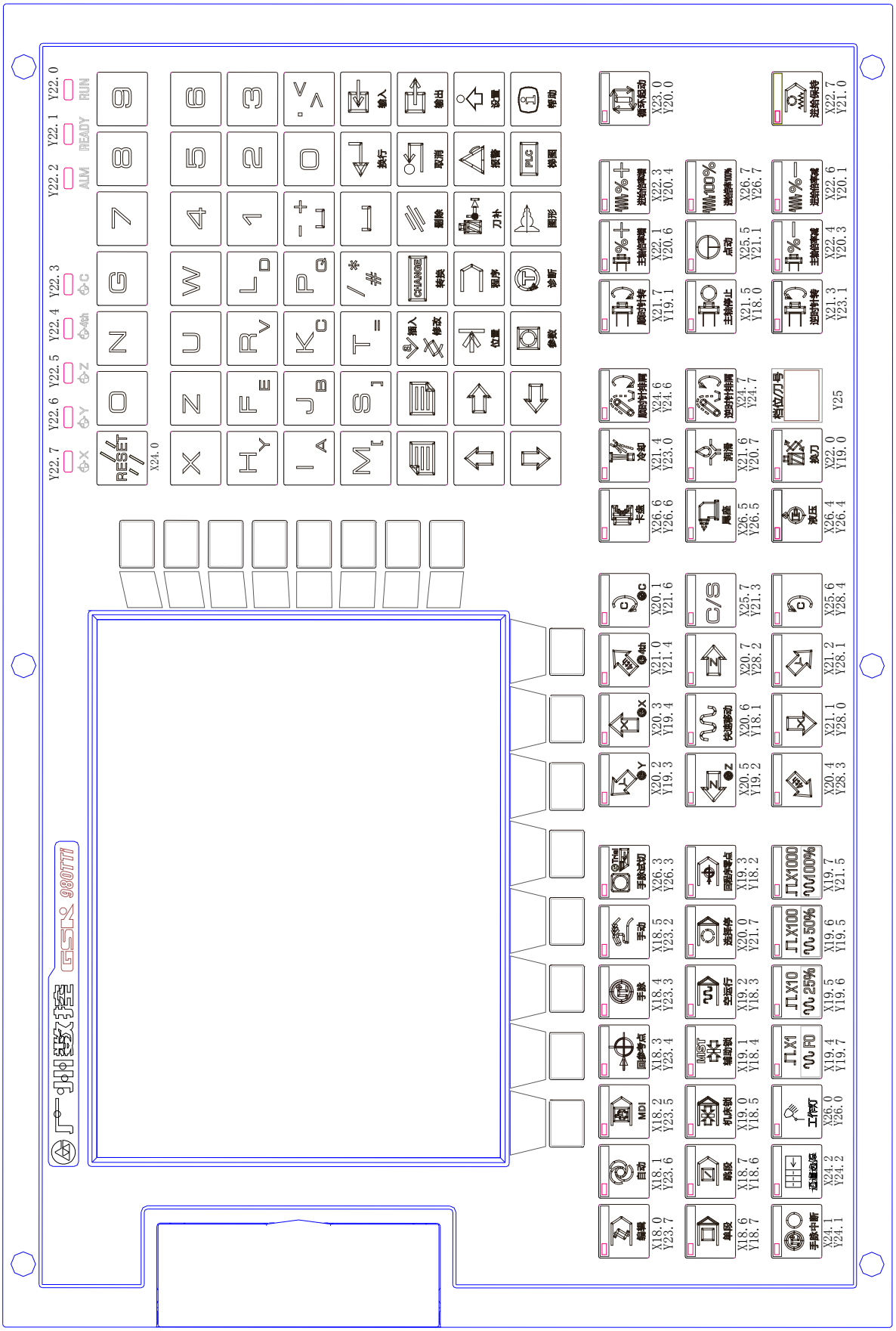


Fig.2-23 GSK980TTi

GSK980TTi-V

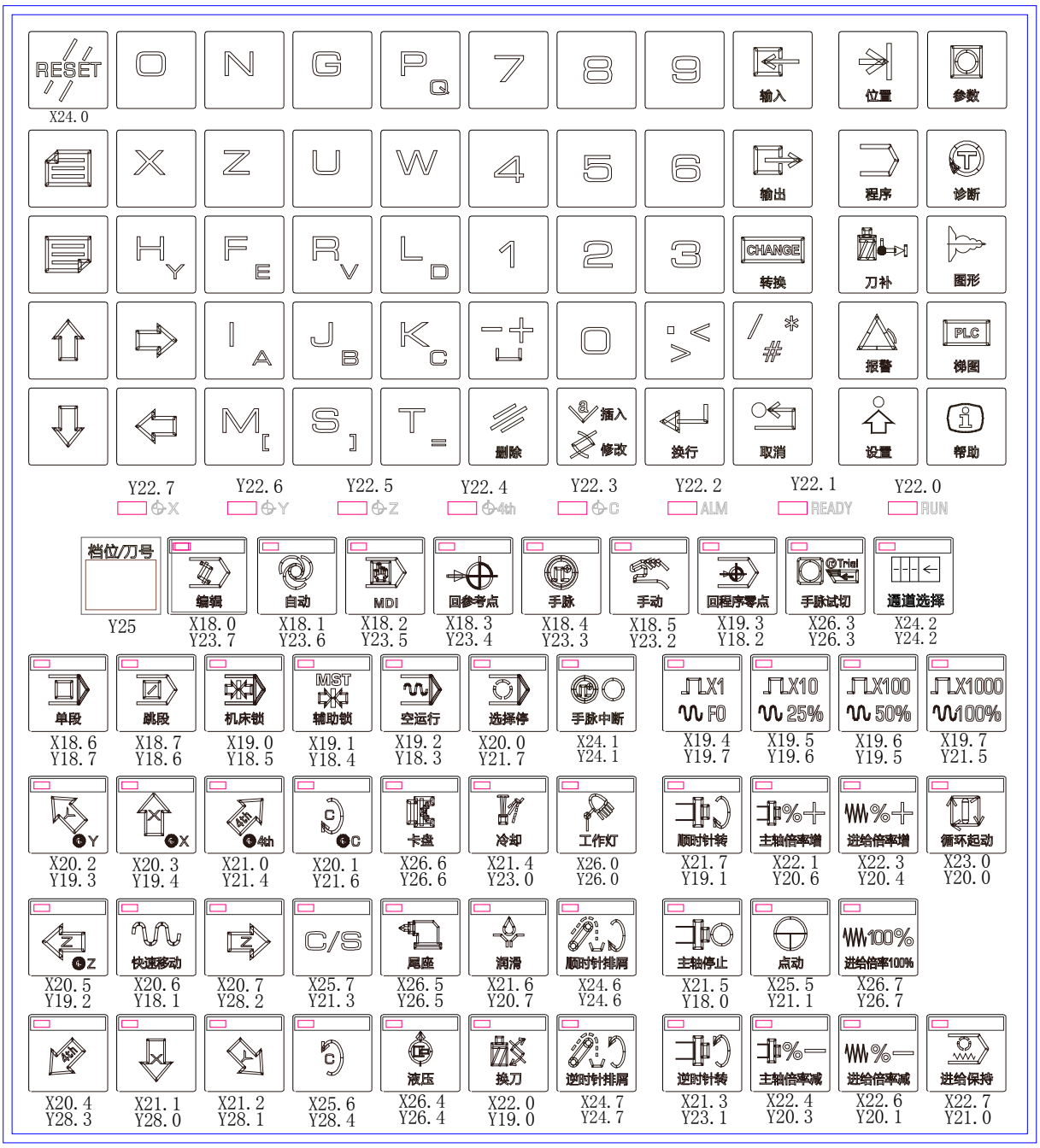


Fig.2-24 GSK980TTi-V

CHAPTER 3 PARAMETERS

3.1 Bit parameter

The state parameter is expressed as follows:

Parameter No.	BIT7	BIT6	BIT5	BIT4	BIT3	BIT2	BIT1	BIT0
0 0 1	***	***	***	ACS	MPG	PROG	ISC	INI

[Data type]: Bit channel type

Bit4 1: Spindle analog voltage control
0: Spindle switching volume control

Bit3 1: MPG mode
0: Step mode

Bit2 1: Programming by radius
0: Programming by diameter

Bit1 1: IS-C incremental system
0: IS-B incremental system

Increment system	Least input increment, least command increment	Abbreviation
0	0.001mm,0.0001inch	IS-B
1	0.0001mm, 0.00001inch	IS-C

Bit0 1: Inch input
0: Metric input

0 0 2	***	***	***	LIFJ	MDITL	LIFC	TNRC	TLIF

Bit4 1: Tool life management group skip active
0: Tool life management group skip inactive

Bit3 1: Tool life management active in MDI mode
0: Tool life management inactive in MDI mode

Bit2 1: Tool life counting type 2, by times
0: Tool life counting type 1, by times

Bit1 1: Tool nose radius offset active
0: Tool nose radius offset inactive

Bit0 1: Tool life management active
0: Tool life management inactive

0 0 3	HTCHG	***	PCOMP	TCOMP	CDCOMP	***	***	OIM

[Data type]: Bit channel type

Bit7 1: High efficiency tool-change function enabled
0: High efficiency tool-change function disabled

Bit5 1: Pitch error offset active

0: Pitch error offset inactive

Bit4 1: Tool offset by coordinate offset

0: Tool offset by move

Bit3 1: Tool compensated by the shift of coordinate system do not memorize the unexecuted tool compensation

0: Tool compensated by the shift of coordinate system memorize the unexecuted tool compensation

Bit0 1: Offset automatically change in metric and inch conversion

0: Offset not change in metric and inch conversion

0	0	4

ABOT	RDRN	DECI	ORC	***	***	PROD	SCW

[Data type]: Bit channel type

Bit7 1: do not memorize absolute coordinate in power-down

0: memorize the absolute coordinate in power-down

Bit6 1: G00 is rapid traverse speed in dry run mode

0: G00 is manual feedrate in dry run mode

Bit5 1: DEC signal is high level for machine zero return

0: DEC signal is low level for machine zero return

Bit4 1: Tool offset by radius

0: Tool offset by diameter

Bit1 1: Relative position for programming in POSITION page

0: Relative position involving offset in POSITION page

Bit0 1: Inch system for min. code unit, active after repowering

0: Metric system for min. code unit, active after repowering

0	0	5

***	***	SMAL	M30	***	POSRE	PPD	***

Bit 5 1: Manual spindle gear change during the execution of S code

0: Automatic spindle gear change during the execution of S code

Bit 4 1: Cursor returns to the head of the program after the execution of M30

0: Cursor does not return to the head of the program after the execution of M30

Bit 2 1: Position recording signal PRC is used during tool setting

0: Position recording signal PRC is not used during tool setting

Bit 1 1: G50 sets relative coordinate value

0: G50 does not set relative coordinate value

0	0	6

***	***	ZM6	ZM5	ZM4	ZMY	ZMZ	ZMX

Bit5~Bit0 1: 6^{th}, 5^{th}, 4^{th}, Y, Z, X zero return type C

0: 6^{th}, 5^{th}, 4^{th}, Y, Z, X zero return type B

0	0	7

DISP	SMZ	ZC6	ZC5	ZC4	ZCY	ZCZ	ZCX

Bit7 1: Enter ABSOLUTE POS page after power on

0: Enter RELATIVE POS page after power on

Bit6 1: Execute the next block after all motion block exactly are executed to the in-position

0: Smooth transition between two blocks

Bit5~Bit0 1: In machine zero return, 6^{th},5^{th}, 4^{th}, Y, Z ,X deceleration signal (DEC6~DEC1) and

one-revolution signal (PC6~PC1) are parallel(use one proximity switch as the deceleration signal and zero signal)

0: In machine zero return, 6th, 5th, 4th, Y, Z ,X deceleration (DEC6~DEC1) and one-revolution signal (PC6~PC1) is separately connected (need the separate deceleration signal and zero signal)

Note: In the zero-return B method; when it is set to 0 and connected independently, it is necessary to detect the PC signal after releasing the block during the zero-return; when it is set to 1 and connected in parallel, the PC signal detects with block after the pressing block decelerates to zero during the zero-return method.

0	0	8	***	***	DIR6	DIR5	DIR4	DIRY	DIRZ	DIRX

Bit5~Bit0 1: The motor's rotation diretion of the 6th, 5th, 4th, Y, Z and X axes are positive

0: The motor's rotation diretion of the 6th, 5th, 4th, Y, Z and X axes are negative

Note: The movement direction of the machine tool is inconsistent with the actual direction, which can be modified by adjusting this parameter.

0	0	9	***	***	6ALM	5ALM	4ALM	YALM	ZALM	XALM

Bit4~Bit0 1: Servo not ready alarm without detection of the6th, 5th, 4th, Y, Z or X axis.

0: Servo not ready alarm detection of the 6th, 5th, 4th, Y, Z or X axis.

0	1	0	***	***	CPF6	CPF5	CPF4	CPF3	CPF2	CPF1

[Data type]: Bit channel type

Bit0~ Bit5: Pulse frequency setting values of backlash offset (by BCD code)

The set frequency =(setting value+1) Kpps

CPF5	CPF4	CPF3	CPF2	CPF1	Set frequency (Kpps)
0	0	0	0	0	1
0	0	0	0	1	2
0	0	0	1	0	3
0	0	0	1	1	4
0	0	1	0	0	5
0	0	1	0	1	6
0	0	1	1	0	7
0	0	1	1	1	8
0	1	0	0	0	9
0	1	0	0	1	10
0	1	0	1	0	11
0	1	0	1	1	12
0	1	1	0	0	13
0	1	1	0	1	14
0	1	1	1	0	15
0	1	1	1	1	16
1	0	0	0	0	17
1	0	0	0	1	18
1	0	0	1	0	19
1	0	0	1	1	20
1	0	1	0	0	21
1	0	1	0	1	22
1	0	1	1	0	23

CPF5	CPF4	CPF3	CPF2	CPF1	Set frequency (Kpps)
1	0	1	1	1	24
1	1	0	0	0	25
1	1	0	0	1	26
1	1	0	1	0	27
1	1	0	1	1	28
1	1	1	0	0	29
1	1	1	0	1	30
1	1	1	1	0	31
1	1	1	1	1	32

0	1	1	BDEC	BD8	***	***	NORF	ZNIK	REFC2	***

Bit7 1: Backlash offset type B, the offset data are output by ascending or descending type and the set frequency is inactive

0: Backlash offset type A, the offset data are output by the set frequency (by bit parameter No.010) or 1/8 of it

Bit6 1: Backlash offset is done by the 1/8 of the set frequency

0: Backlash offset is done by the set frequency

Bit3 1: Manual machine zero return inactive

0: Manual machine zero return active

Bit2 1: Direction key locked during zero return, homing continues to end by pressing direction key once;

0: Direction key unlocked, which is held on during zero return

Bit1 1: Ref.C mode, the feeding slowdown when pressing down switch, reverse after decelerating to zero.

0: Ref.C mode, the feeding slowdown when releasing the switch, reverse after decelerating to zero.

0	1	2	APRS	WSFT	DOFSI	***	EAL	RF0	HDINT	ISOT

Bit7 1: Automatic ABSOLUTE set after reference return

0: ABSOLUTE not automatically set after reference return

Bit6 1: Workpiece coordinate offset active, defined by offset No.000

0: Workpiece coordinate offset inactive

Bit5 1: Trial tool setting active

0: Trial tool setting inactive

Bit3 1: Program editing allowed during CNC alarming

0: Program editing unallowed during CNC alarming

Bit2 1: When the feedrate override is 0% during rapid traverse, tool stop moving

0: When the feedrate override is 0% during rapid traverse, tool does not stop moving

Bit1 1: Handle interrupt is valid (power on)

0: Handle interrupt is invalid (power on)

Bit0 1: Prior to machine zero return after power on, manual rapid traverse active

0: Prior to machine zero return after power on, manual rapid traverse inactive

0	1	3	RTH	RPH	HW6	HW5	HW4	HWY	HWZ	HWX

Bit7 1: When system reset, the handle interrupt move amount is cancelled

0: When system reset, the handle interrupt move amount is not cancelled

Bit6 1: When MPG feedrate exceeds the rapid movement speed,the exceeding part is not ignored and saved in CNC

0: When MPG feedrate exceeds the rapid movement speed,the exceeding part is ignored and saved in CNC

Bit5~Bit0 1: Coordinates increase in 6^{th}, 5^{th}, 4^{th}, Y, Z or X MPG (CCW) rotation

0: Coordinates increase in 6^{th}, 5^{th}, 4^{th}, Y, Z or X MPG (CW) rotation

0 1 4	***	***	ZRS6	ZRS5	ZRS4	ZRSY	ZRSZ	ZRSX

Bit5~Bit0 1: 6^{th}, 5^{th}, 4^{th}, Y, Z, X have the machine zeroes. The system needs to check the deceleration signal and zero signal in executing the machine zero return

0: 6^{th}, 5^{th}, 4^{th}, Y, Z, X have no machine zeroes. The system does not check the deceleration signal and zero signal till it returns to the zero of machine coordinate in executing the machine zero return

0 1 7	SCRD	***	RSCD	***	TSD	QNI	TSL	TS1

[Data type]: Bit channel type

Bit7 1: Coordinate system is stored before power-off

0: Coordinate system is not stored before power-off; the default system is set by G54 after power-on

Bit5 1: Coordinate system is set by G54 after reset

0: Coordinate system is not changed after reset

Bit3 1: Movement direction discrimination is valid during offset measurement 4 point input

0: Movement direction discrimination is invalid during offset measurement 4 point input

Bit2 1: Tool offset number is set through G signal

0: Tool offset number is set through the current cursor;

Bit1 1: Tool offset measurement detection signal is valid at low level

0: Tool offset measurement detection signal is valid at high level

Bit0 1: Tool offset automatic measurement detection signal 1 point input

0: Tool offset automatic measurement detection signal 4-point input

0 2 0	TRIP	DSB	***	***	***	***	IAL	RST

Bit7 1: Puls trial cutting valid in all channels

0: Puls trial cutting valid only in current channel

Bit6 1: When one channel single stop, the other dwell

0: When one channel single stop, the other not affected

Bit1 1: When one channel alarms, the other not affected

0: When one channel alarms, the other dwell and forbides program restart

Bit0 1: Press reset button in MDI pane, valid for all chanels

0: Press reset button in MDI pane, valid only for selected channel

1 7 2	***	MST	MSP	MOT	ESP	***	LZR	MOTDEC

Bit6 1: External cycle start signal (ST) inactive)

0: External cycle start signal (ST) active

Bit5 1: External SP signal inactive

0: External SP signal active with external dwell switch needed, otherwise CNC shows “Pause”

Bit4 1: Not detect software stroke limit
0: Detect software stroke limit
Bit3 1: Emergency stop inactive
0: Emergency stop active
Bit1 1: Software limit is inactive after power-on and before machine zero return
0: Software limit is active after power-on and before machine zero return
Bit0 1: When Arriving stored travel position, CNC is stopped slowly
0: When Arriving stored travel position, CNC is stopped immediately

1	7	4

PARTSET	***	TRPMOD	TRP	KEY1	THDTLMOD	THDA	CKSPD

Bit7 1:Continue to run part program while current machining workpiece counts reach to max. counts.and machining workpiece counts accumulate
0:Continue to run part program while current machining workpiece counts reach to max. counts.and machining workpiece counts restart
Bit5 1: Screw repair mode use follow the spindle with encoder
0: Screw repair mode use interpolation with servo spindle
Bit4 1: Thread repair function is valid
0: Thread repair function is invalid
Bit3 1: Program switch ON at power on
0: Program switch OFF at power on
Bit2 1: Thread tailing mode tailling according angle;
0: Thread tailing mode rapid tailing.
Bit1 1: Thread cut is exp. acc&dec
0: Thread cut is lin. acc&dec
Bit0 1: Whether detect the spindle speed is too slow in thread cutting don't detect\
0: Whether detect the spindle speed is too slow in thread cutting do detect

1	7	5

SPFD	SAR	***	VAL5	VAL4	VALY	VALZ	VALX

[Data type]: Bit channel type
Bit7 1: In cutting feed, do not permit the spindle stopping rotation; the spindle stops, the feed also stops when the system appears No.404 alarm cutting
0: In cutting feed, permit the spindle stops rotation; the spindle stops rotation, but the feed does not stop
Bit6 1: Detect spindle SAR signal prior to cutting
0: Not detect spindle SAR signal prior to cutting
Bit4 1: 5th movement key  is positive,  is negative
0: 5th movement key  is positive,  is negative
Bit3 1: 4th movement key  is positive,  is negative
0: 4th movement key  is positive,  is negative
Bit2 1: Y movement key  is positive,  is negative
0: Y movement key  is positive,  is negative

Bit1 1: Z movement key  is positive,  is negative

0: Z movement key  is positive,  is negative

Bit0 1: X movement key  is positive,  is negative

0: X movement key  is positive,  is negative

1	8	0

NAT	***	***	***	DISPC	CUSTOM	***	SPOS

Bit7 1: Function ATAN, ASIN range is 90.0～270.0;

0: Function ATAN, ASIN range is -90.0～90.0

Bit3 1: Boot dispaly if custom page setted to display custom page

0: Boot dispaly if custom page setted to display fault page

Bit2 1: Display custom page display

0: Display custom page hide

Bit0 1: DIS TO GO displayed in POS&PRG page

0: RELATIVE POS displayed in POS&PRG page

1	8	1

RLC	MRC	ZCL	CLMDI	MCL	M30&MDI	DDP	LCNMD

Bit7 1: G52 local coordinate system is cancelled after reset

0: G52 local coordinate system is not cancelled after reset

Bit6 1: G52 local coordinate system is cancelled after the execution of M30, M02

0: G52 local coordinate system is not cancelled after the execution of M30, M02

Bit5 1: G52 local coordinate system is cancelled after reference point return

0: G52 local coordinate system is not cancelled after reference point return

Bit4 1: When system reset,the program block of MDI are deleted

0: When system reset,the program block of MDI are not deleted

Bit3 1: Blocks edited in MDI mode are deleted after reset

0: Block edited in MDI mode are not deleted after reset

Bit2 1: After the execution of M02, M30,the program block of MDI are not deleted

0: After the execution of M02, M30,the program block of MDI are deleted

Bit1 1: The angle in CHF/CNR measurement-programe-inputing function was specified by Supplementary one

0: The angle in CHF/CNR measurement-programe-inputing function was specified by Nornal format

Bit0 1: In linear chamfering, L means the length between chamfering point and cross point;

0: In linear chamfering, L means the length of chamfering

1	8	3

***	***	***	MZR5	MZR4	MZRY	MZRZ	MARX

Bit4 1: for 5th, press  key to execute the machine zero return

0: for 5th, press  key to execute the machine zero return

Bit3 1: for 4th, press  to execute the machine zero return

0: for 4th, press  to execute the machine zero return

Bit2 1: for Y, press  to execute the machine zero return

0: for Y, press  to execute the machine zero return

Bit1 1: for Z, press  to execute the machine zero return

0: for Z, press  to execute the machine zero return

Bit0 1: for X, press  to execute the machine zero return

0: for X, press  to execute the machine zero return

1	8	4	***	***	***	***	***	L2	L1	L0

Bit0, Bit1, Bit2: Language selection:

Language	Bit2	Bit1	Bit0
Chinese	0	0	0
English	0	0	1
Portuguese	0	1	0
Spanish	0	1	1
Russian	1	1	0

1	8	5	SK0	SKF	AEO	***	***	***	PRPD	PLA

Bit7 1: Skip input active as G31 signal is 0

0: Skip input active as G31 signal is 1

Bit6 1: Feedrate override per minute or dry run active for G31

0: Feedrate override per minute or dry run inactive for G31

Bit5 1: Input active if G36, G37 (XAE, ZAE) signal is 0

0: Input active if G36, G37 (XAE, ZAE) signal is 1

Bit1 1: Axis rapid traverse rate of PLC by input value

0: Axis rapid traverse rate of PLC by parameter value (X axis: No.022；Z axis:No.023；Y axis:No.134)

Bit0 1: PLC axis control active, active after repowering

0: PLC axis control inactive, active after repowering

1	8	6	RTORI	SRS	***	***	***	RTCRG	***	***

Bit7 1: In executing M29, the spindle executes the machine zero return

0: In executing M29, the spindle does not execute the machine zero return

Bit6 1: The spindle selection signal is RGTSPn in the multiple spindle rigid tapping

0: The spindle selection signal is SWSn in the multiple spindle rigid tapping

Bit2 1: In rigid tapping cancel, do not wait for G61.0 to be 0 in executing the next block

0: In rigid tapping cancel, wait for G61.0 to be 0 in executing the next block

1	8	7	IS1	IS0	***	***	***	***	ROS	ROT

[Data type]: Bit axis type

BIT7, BIT6: set increment system of each axis.00:accordance with increment system of the system.

01:IS-A，10:IS-B；11:IS-C

Bit1, Bit0: 00: set the additional axis to the linear axis; 01: set the additional axis to the rotary axis （A type）;11: set the additional axis to the rotary axis (B type); 10: set the additional axis to be invalid

Note: The parameter is invalid to setting of X, Z axis

1	8	8	RFCK	RRT	***	***	***	RRL	RAB	ROA

[Data type]: bit axis

Bit7 1: G28 running every time to check the PC signal
0: G28 running only first to check the PC signal

Bit6 1: Zero return D mode is used when the additional axis is a rotary axis
0: Zero return A, B, C mode is used when the additional axis is a rotary axis (power-on again)

Bit2 1: The relative coordinate cycle function is valid when the additional axis is rotary
0: The relative coordinate cycle function is invalid when the additional axis is rotary(power-on again)

Bit1 1: The rotation direction is based on that of sign when the additional axis is rotary
0: The rotation direction is based on the nearby principle when the additional axis is rotary.

Bit0 1: The absolute coordinate cycle function is valid when the additional axis is rotary
0: The absolute coordinate cycle function is invalid when the additional axis is rotary (power-on again)

Note: The parameter is invalid to setting of X, Z axis

1	8	9	CTS	***	APC	APZ	***	***	***	***

[Data type]: Bit axis

Bit7 1: Each servo axis didn't connect servo drive , it is necessary to restart
0: Each servo axis connected servo drive , it is necessary to restart

Bit5 1: The used encoder type of each axis is absolute encoder
0: The used encoder type of each axis is incremental encoder

Bit4 1: When each axis matches with an absolute encoder, the mechanical position and absolute one are shown consistent
0: When each axis matches with an absolute encoder, the mechanical position and absolute one are shown inconsistent

1	9	5	SSPD	PLCS	***	CSD	RPOS	SRV	CSREF	RCS

Bit7 1: series spindle controllable function enabled (series spindle The slave ID number of the spindle 1~3 is 11~13) , it is necessary to restart
0: Series spindle controllable function disabled (analogy spindle), it is necessary to restart.

Bit6 1: PLC spindle rotation speed function of each axis enabled
0: PLC spindle rotation speed function of each axis disabled

Bit4 1: Cs axis in the velocity controllable mode, spindle coordinate hold
0: Cs axis in the velocity controllable mode, spindle coordinate changes with actual position

Bit3 1: Spindle position function is valid (When it is enabled, the spindle will perform mechanical releasing/clamping interlocking operation before/after positioning)
0: Spindle position function is invalid

Bit2 1: Servo motor spindle function is valid ,power on
0: Servo motor spindle function is invalid ,power on

Bit1 1: Change to position mode,the axis do return REF

0: Change to position mode, the axis doesn't return REF

Bit0 1: Cs function of spindle valid ,power on

0: Cs function of spindle is: invalid, power on

1 9 6	***	***	***	MSEN	***	***	***	***

Bit4 1: The multiple spindle control is enabled

0: The multiple spindle control is disabled

2 0 2	CTHDNOENC	***	***	***	***	***	***	EXTRAIO

Bit7 1: Circular screw interpolation adopts C axis interpolation mode

0: Circular screw interpolation adopts follow spindle mode

Bit0 1: Extended I/O unit linked is valid

0: Extended I/O unit linked is invalid

2 0 4	***	***	***	***	***	***	***	NRS

[Data type]: Bit channel type

Bit0 1: During resetting, the synchronous/mixed/overlap control is NOT released

0: During resetting, the synchronous/mixed/overlap control is released

2 0 5	SPV	SCM	SCD	SPS	SPM	SMR	SYW	PKU

[Data type]: Bit axis type

Bit7 1: When the synchronous control ends, the workpiece coordinate system of the slave axis is automatically set

0: When the synchronous control ends, the workpiece coordinate system of the slave axis is NOT automatically set

Bit6 1: The coordinate system directions of the slave axis and the master axis which are controlled synchronously are opposite

0: The coordinate system directions of the slave axis and the master axis which are controlled synchronously are same

Bit5 1: The workpiece coordinate values of the master axis controlled synchronously are calculated by the machine coordinate values of the master axis and the slave axis

0: The workpiece coordinate values of the master axis controlled synchronously are calculated by the machine coordinate values of the master axis

Bit4 1: After the synchronous control is released, the workpiece coordinate system of the master axis is automatically set

0: After the synchronous control is released, the workpiece coordinate system of the master axis is NOT automatically set

Bit3 1: When the synchronous control is started, the workpiece coordinate system of the master axis is automatically set

0: When the synchronous control is started, the workpiece coordinate system of the master axis is NOT automatically set

Bit2 1: Movement directions of the slave axis and the master axis are opposite

0: Movement directions of the slave axis and the master axis are same

Bit1 1: It can be used as the synchronous master axis and the synchronous slave axis meanwhile

0: It can NOT be used as the synchronous master axis and the synchronous slave axis

meanwhile

Bit0 1: When the synchronous control axis is resident, the relative and the absolute coordinates are refreshed

0: When the synchronous control axis is resident, the coordinates are NOT refreshed

Note: In the synchronous control, the parameters № 0205#2～№ 0205#7 are set in the slave axis side.

2	0	6	MCE	MCS	MCD	MPS	MPM	MRF	***	MDM

[Data type]: Bit axis type

Bit7 1: During the mixed control releasing, the workpiece coordinate system is automatically set with the absolute coordinate system of the mixed interchange control axes

0: During the mixed control releasing, the workpiece coordinate system is automatically set with the parameters and the absolute coordinate system of the mixed interchange control axes

Bit6 1: When the mixed control is started, the workpiece coordinate system is automatically set with the absolute coordinate system of the mixed interchange control axes

0: When the mixed control is started, the workpiece coordinate system is automatically set with the parameters and the absolute coordinate system of the mixed interchange control axes

Bit5 1: During the mixed control, the coordinate system directions of the interchange axes are opposite

0: During the mixed control, the coordinate system directions of the interchange axes are same

Bit4 1: During the mixed control, the workpiece coordinate system is automatic set

0: During the mixed control, the workpiece coordinate system is NOT automatic set.

Bit3 1: When the mixed control is started, the workpiece coordinate system is automatically set;

0: When the mixed control is started, the workpiece coordinate system is NOT automatically set

Bit2 1: During the mixed control, the rapid traverse rate is the one of the movement axis

0: During the mixed control, the rapid traverse rate is the one of the commanded axis

Bit0 1: The machine coordinate in the mixed control is displayed the coordinate value of the mixed target axes

0:The machine coordinate in the mixed control is displayed the coordinate value of the local channel axes

2	0	7	***	***	***	OPS	***	OMR	***	***

[Data type]: Bit axis type

Bit4 1: When the overlap control is released, the movement amount of the master axis is added into the workpiece coordinates of the slave axis

0: When the overlap control is released, the movement amount of the master axis is NOT added into the workpiece coordinates of the slave axis

Bit2 1: The movement directions of the overlapped slave axis and the master one are opposite

0: The movement directions of the overlapped slave axis and the master one are same

Note: During the overlap control, the parameter № 0207 is set in the slave axis side.

2	3	1	***	***	ZDCL	IFE	IFM	***	TY1	TY0

Bit5 1: During the interference check in the channels, check the interference in Z axis direction

0: During the interference check in the channels, NOT check the interference in Z axis

direction

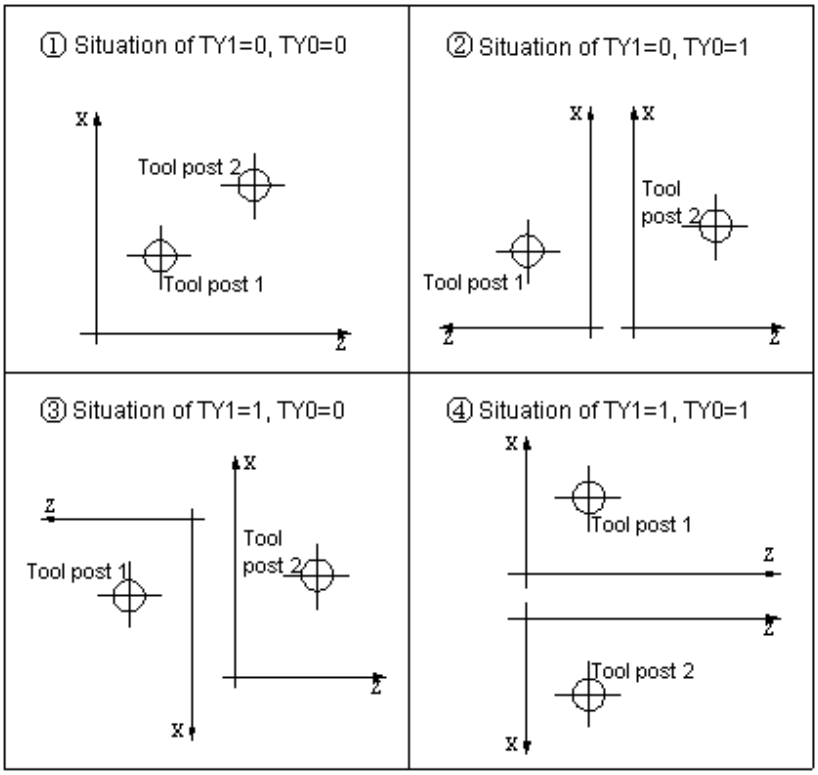
Bit4 1: The interference check is executed in the channels

0: The interference check is NOT executed in the channels

Bit3 1: In manual mode, the interference check is executed in the channels

0: In manual mode, the interference check is NOT executed in the channels

Bit1 Bit0: The relation of the coordinate systems of two tool posts is set by the parameters based on the tool post of the channel 1



2	5	0

GWEN	***	***	***	***	***	VERI	GSKLINK

Bit7 1: Gateway connection is valid

0: Gateway connection is invalid

Bit1 1:System GSKLINK bus communication CRC verification enabled

0:System GSKLINK bus communication CRC verification disabled

Bit0 1: Whether the system GSKLINK bus connection is enabled

0: Whether the system GSKLINK bus connection is disabled

3.2 Data parameter

№	Data meaning	Data range	Explanation
015	Pulse frequency multiplication coefficient	1～99999999	Formula of electric gear ratio: $\frac{CMR}{CMD}=\frac{C*\delta}{L}*\frac{Z_M}{Z_D}$ Δ: Least command increment Z_M: Teeth number of pulley on leading screw side C: Motor encoder line number Z_D: Teeth number of pulley on motor side L: Screw lead
016	Pulse frequency division coefficient	1～99999999	
019	Run-out length in threading	0～225	Thread run-out width= THDCH×0.1×screw lead
021	Voltage offset value when spindle max. speed analog voltage output is 10V(Unit: mV)	-2000～2000	
022	Max.speed of rapid traverse	10～99999999	
023	Acc&dec T.const.in rapid traverse & JOG(ms)	0～4000	
024	Acc.&dec.T in Handwheel(ms)	0～4000	
026	Acc&dec T.const.in tailing(short axis)(ms)	0～4000	
027	Upper limit speed in feeding of each axis	Metric 10～30000 Inch 5～11600	
028	Initial speed in threading	Metric 6～8000 Inch 3～3000	
029	Acc.&dec.T in cutting(ms)	0～4000	
030	Initial feedrate of acc.&dec.	Metric 0～8000 Inch 0～3000	
031	JOG speed when feedrate 100%	10～30000	
032	Rapid traverse speed as override is Fo	Metric 0～4000 Inch 0～1500	
033	Low speed of each ax in ref.point return	Metric 6～4000 Inch 3～1500	
034	Backlash compensation of each axis	IS-B: 0～2000 IS-C: 0～20000	X is the diameter value
036	Voltage compensation value when SVC is 0V(mv)	-1000～1000	
037	Max. speed of spindle in gear 1(r/min)	10～9999	
038	Max. speed of spindle in gear 2(r/min)	10～9999	
039	Max. speed of spindle in gear 3(r/min)	10～9999	
040	Max. speed of spindle in gear 4(r/min)	10～9999	
041	Acc&dec initial speed in JOG or Handwheel	0～8000	
042	Increment of block sequence number	1～100	
043	Min. spindle speed in G96(r/min)	0～9999	
044	Serial communication rate	1200、2400、4800、9600、19200、38400、57600、115200	

№	Data meaning	Data range	Explanation
045	Max. machine coordinate in overtravel	-99999999～99999999	If the BIT2 of the parameter No.001 is set for diameter, the X axis value is specified by diameter; if for radius, the X axis value is specified by radius.
046	Min. machine coordinate in overtravel	-99999999～99999999	
047	Abs.coordinate setting value of each axis mach.zero	-99999999～99999999	
051	Each advance value in rough cut(G71,G72)	IS-B: 1～99999 IS-C: 1～999999	The setting value is used when the program code is not specified
052	Each retract value in rough cut(G71,G72)	IS-B: 0～99999 IS-C:0～999999	The setting value is used when the program code is not specified
053	Retract value in rough cut of X in G73	-99999999～99999999	The setting value is used when the program code is not specified
054	Retract value in rough cut of Z in G73	IS-B:-9999999～9999999 IS-C:-99999999～99999999	The setting value is used when the program code is not specified
055	Cutting times of G73	1～9999	The setting value is used when the program code is not specified
056	Retract value of Z in G74 or X in G75	IS-B: 0～99999 IS-C: 0～999999	The setting value is used when the program code is not specified
057	Finish machining times in G76	1～99	The setting value is used when the program code is not specified
058	Tool angle in G76	0～99	The setting value is used when the program code is not specified
059	Least cutting depth in G76	IS-B: 0～99999 IS-C: 0～999999	The setting value is used when the program code is not specified
060	Finish machining cut depth in G76	IS-B: 0～99999 IS-C: 0～999999	The setting value is used when the program code is not specified
061	Selecting cut depth in G78	0~1	0: Equidistant in-feed 1: Degressive in-feed.
062	Cutting mode in G78	0~3	0: Cutting edge is cut in along with the neutral cable of the thread teeth; 1: Cutting edge is cut in along the left of the thread teeth; 2: Cutting edge is cut in along the right of the thread teeth; 3: Cutting edge is cut in alternatively with left and right of the thread teeth.
067	Spindle voltage as shifting(mv)	0～10000	
068	Spindle revolve speed in JOG/HNDL mode(r/min)	0～3000	When set to 0, the spindle speed is specified by the S code
069	The corresponding spindle encoder number of each spindle	0～3	0: Without installing the encoder 1: Comes from the encoder interface 1 2: Comes from the encoder interface 2 3: Comes from the GSKLINK bus
070	Pulse of spindle coder per rotation	100～10000	
071	Output time for reset signal(ms)	16～4080	
072	Delay time of detect SAR signal(ms)	0～4080	
073	Maximum rev suppress of spindle motor	0～65535	Setting value =(max. clamp speed of spindle/max. speed of spindle

№	Data meaning	Data range	Explanation
			motor)×65535
074	Minimum rev suppress of spindle motor	0～65535	Setting value =(min. clamping speed of spindle motor/max. speed of spindle motor)×65535
075	Maximum spindle rev speed(r/min)	0～6000	0:unlimited
076	The spindle speed limit when speed limit signal is active(r/min)	0～9999	0:unlimited
077	The feedrate limit when speed limit signal is active	0～15000	0:unlimited
084	Total tool number	1～32	
086	User macro program G code of calling program No. 9010	0～9999	
087	User macro program G code of calling program No. 9011	0～9999	
088	User macro program G code of calling program No. 9012	0～9999	
089	User macro program G code of calling program No. 9013	0～9999	
090	User macro program G code of calling program No. 9014	0～9999	
091	User macro program G code of calling program No. 9015	0～9999	
092	User macro program G code of calling program No. 9016	0～9999	
093	User macro program G code of calling program No. 9017	0～9999	
094	User macro program G code of calling program No. 9018	0～9999	
095	User macro program G code of calling program No. 9019	0～9999	
096	Minimum pitch error compensation position number	0～1199	
097	Maximum pitch error compensation position number	0～1199	
098	Pitch error compensation position numbers corresponding to machine zero points	0～1199	
099	Pitch error compensation intervals	Note2	
106	Spindle fluctuation alarm limit in threading	0～100	not detect spindle fluctuation alarm if set to 0 Formula=|current speed － previous period speed|×200/(current speed＋previous period speed)
107	Short axis speed in threading run-out	0～8000	run-out by threading feedrate if set to 0
109	Spindle jogging speed(r/min)	1～8000	
110	Spindle gear teeth number in drive ratio	1～255	
111	Encoder gear teeth number in drive ratio	1～255	
113	High speed of each axis in ref. point return	10～99999999	

№	Data meaning	Data range	Explanation
114	Offset in each axismachine ref. point	IS-B:-99999～99999 IS-C:-999999～999999	
119	No.of valid keys pressed simultaneously	2～5	
120	REF1_COORDn 1nd reference point machine coordinates	-99999999～99999999	
121	REF2_COORDn 2nd reference point machine coordinates	-99999999～99999999	
122	REF3_COORDn 3rd reference point machine coordinates	-99999999～99999999	
123	REF4_COORDn 4th reference point machine coordinates	-99999999～99999999	
130	Input B of automatic tool offset,distance from benchmark position to X+ meseare plane	-99999999～99999999	
131	Input B of automatic tool offset,distance from benchmark position to X- meseare plane	-99999999～99999999	
132	Input B of automatic tool offset,distance from benchmark position to Z+ meseare plane	-99999999～99999999	
133	Input B of automatic tool offset,distance from benchmark position to Z- meseare plane	-99999999～99999999	
134	Reference frame offset consult tool offset num	1~32	not automatic tool offset
135	Least cyc num of confirm direction	0~8	
136	tool offset testing input in method B, X plus direction test signal add	0、20~27、30~37	0:invalid;20~27,30~37:add X2.0~X3.7
137	tool offset testing input in method B, X minus direction test signal add	0、20~27、30~37	0:invalid;20~27,30~37:add X2.0~X3.7
138	tool offset testing input in method B, Z plus direction test signal add	0、20~27、30~37	0:invalid;20~27,30~37:add X2.0~X3.7
139	tool offset testing input in method B, Z minus direction test signal add	0、20~27、30~37	0:invalid;20~27,30~37:add X2.0~X3.7
140	Limit-value of one time for tool wear input	IS-B: 1～99999 IS-C: 1～999999	
141	Rapid feedrate of automatic tool offset	IS-B: 15～1000 IS-C: 7～375	
142	X(AUTO_OFFSET_Y_X) γ values in automatic tool offset	1～9999999	X setting value is the radius in diameter or radius programming
143	Z(AUTO_OFFSET_Y_Z) γ values in automatic tool offset	1～9999999	
144	AUTO_OFFSET_E_X(X) ε values in automatic tool offset	1～9999999	X setting value is the radius in diameter or radius programming
145	AUTO_OFFSET_E_Z(Z) ε values in automatic tool offset	1～9999999	
146	automatic tool compensation X axis direction testing signal add	0、20~27、30~37	0:invalid; 20~27,30~37:add X2.0~X3.7

№	Data meaning	Data range	Explanation
147	automatic tool compensation Z axis direction testing signal add	0、20~27、30~37	0:invalid; 20~27,30~37:add X2.0~X3.7
148	Multilevel skip G31、G31 P1 or G04 Q1 signal detect address setting	0、20~27、30~37	0:invalid; 20~27,30~37:add X2.0~X3.7
149	Multilevel skip G31 P2 or G04 Q2 signal detect address setting	0、20~27、30~37	0:invalid; 20~27,30~37:add X2.0~X3.7
150	Multilevel skip G31 P3 or G04 Q3 signal detect address setting	0、20~27、30~37	0:invalid; 20~27,30~37:add X2.0~X3.7
151	Multilevel skip G31 P4 or G04 Q4 signal detect address setting	0、20~27、30~37	0:invalid; 20~27,30~37:add X2.0~X3.7
154	Maximum difference of radius at start and end points	Note1	
162	Initial speed of CS axis(deg/min)	0～4000	
163	Acc&decelerate time constant of CS axis(ms)	0～10000	
164	Set an M code to change the spindle rotating mode to the spindle positioning mode.	0～96	
166	Initial speed of rigid Tapping	0～4000	
167	Acc&decelerate time constant of rigid Tapping(ms)	0～10000	
169	Override rate of rigid Tapping return	0～200	100% if setting 0
170	M code of clampping spindle while rigid tapping	3～97	
173	Take count of machining piece M code	0～97	0:defined M99 & M30
209	axis recognize no.setting	0～999	
210	marst axis setting of slave axis in synchronous control	0、101～105、201～205	set slave axis aside master axis path no *100+axis no.in path
211	coordinate value of slave axis reference in master coordinate system	-99999999~99999999	
212	exchange axis setting in mix control	0、101～105、201～205	set slave axis aside mix path no.*100+axis no. in path
213	coordinate value in mix control coordinate of mix exchange axis reference	-99999999~99999999	Example:The coordinate value of the reference position of the channel 2 in the channel 1 workpiece coordinate system is （ΔX1m，ΔZ1m）, Z1mΔX1m is set by the parameter №0213 $_{X1,}$ and ΔX2m is set by the parameter №0213 $_{X2.}$
214	imaginary axis of each axis setting	0、101～105、201～205	imaginary path no.*100+axis no. in path
215	master axis setting of slave axis in overlap control	0、101～105、201～205	set slave axis aside master axis path no.*100+axis no. in path
219	Selection of the DI/DO group for any axis controlled by the PLC	0～4	when it is 0, it means it can't be used as PLC axis control
220	The corresponding interface number of each spindle output	0～14	0: Disabled analog spindle output; 1, 2: analog spindle interface 1 and 2; 11~14: extension IO unit analog output interface 1~4.
221	Feed axis number relevant to the spindle.	0～6	0: without 1~6:the 1st axis to the 6th axis

№	Data meaning	Data range	Explanation
223	set system channel no	1～2	
225	Set name of every axis	65、66、67、88、89、90	The axis name of X or Z axis is fixed, which can not be modified
227	each axis belong path setting	1～2	
228	each spindle belong path setting	1～2	
229	distribute axis no. of each path	1～5	
230	Attribute of axis	0～7	0: It is neither the three basic axes and an axis parallel to them 1: X axis of the three basic axes 2: Y axis of the three basic axes 3: Z axis of the three basic axes 5: An axis parallel to X 6: An axis parallel to Y 7: An axis parallel to Z
232	basic point x coordinate value of turret 2, in the coordinate system of z-x surface origined origined from turret 1 basic point	-99999999～99999999	Setting the metric/inch state and the system increment of the channel 2 and the parameters can become valid after power-on, again.
233	basic point z coordinate value of turret 2, in the coordinate system of z-x surface origined origined from turret 1 basic point	-99999999～99999999	Setting the metric/inch state and the system increment of the channel 2 and the parameters can become valid after power-on, again.
235	Linear axis setting of the polar coordinate interpolation	0～4	0: X axis 1: Z axis 2: Y axis 3: 4^{th} axis 4: 5^{th} axis 5: 6^{th} axis
236	Rotary axis setting of the polar coordinate interpolation	2～4	2: Y axis 3: 4^{th} axis 4: 5^{th} axis 5: 6^{th} axis
237	Acceptance two line angle error of 0 or 180 degree in C offset	ISB: 0～100 IS-C: 0～10000	
240	Clamp rapid traverse rate of manual handle retrace(%)(0-100)	0～100	
241	Go distance per pulse of handwheel for manual handle retrace(%)(0-100)	0～100	
243	Set the quantity of MPG(1~2)	1～2	
244	MPG interface of MPG 1	0～1	0: Rear cover MPG interface 1 (CN31) 1: Rear cover MPG interface 2(CN32)
245	MPG interface of MPG 2	0～1	
249	When an absolute encoder is equipped with each axis, the Max. position error value allows when the coordinate system is set up with power-on.	0～999999	

Note1	Metric	IS-B	IS-C	Inch	IS-B	IS-C
		0～1000	0～10000		0～50	0～500

Note 2	Metric	IS-B	IS-C	Inch	IS-B	IS-C
		1000～999999	10000～9999999		400～999999	4000～9999999

3.3 PLC parameters（standard PLC definition）

3.3.1 K parameters

K	1	0	LMIT	LMIS	EXESP	JSPD	OVRI	***	RSJG	RRW

Bit7 1: PATH 1 overtravel of each axis is enabled
0: PATH 1 overtravel of each axis is disabled
Bit6 1: PATH 1 alarm for overtravel signal low of each axis
0: PATH 1 alarm for overtravel signal high of each axis
Bit5 1: External ESP signal is enabled
0: External ESP signal is disabled
Bit4 1: PATH 1 spindle JOG of is enabled in any modes
0: PATH 1 spindle JOG of is enabled in Manual, MPG, Zero Return mode
Bit3 1: PATH 1 feedrate override is fixed to 100%
0: PATH 1 feedrate override can be adjusted.
Bit1 1: PATH 1 the spindle lubricating, cooling output remains when resets
0: PATH 1 the spindle lubricating, cooling output closes when resets
Bit0 1: PATH 1 the cursor returning to program home is enabled in any modes when resets
0: PATH 1 the cursor returning to program home is enabled in Edit mode when resets

K	1	1	***	***	CHET	TCPS	CTCP	TSGN	CHTB	CHTA

Bit5 1: check tool signal when channel 1 completes tool change
0: do not check tool signal when channel 1 completes tool change
Bit4 1: tool post lock signal of channel 1 is HIGH (connected with +24V) is enabled
0: tool post lock signal of channel 1 is LOW (disconnected with +24V) is enabled
Bit3 1: channel 1 detects tool post lock signal
0: channel 1 does not detect tool post lock signal
Bit2 1: channel 1's tool signal LOW (disconnected with +24V) is enabled
0: channel 1's tool signal HIGH (connected with +24V) is enabled
Bit1 Bit0 : channel 1's tool change mode selects bit 1 tool change mode selects bit 0

CHTB	CHTA	Tool post type
0	0	Standard tool change mode B
0	1	Standard tool change mode A

K	1	2	***	***	***	RCKC	***	NYQP	SLSP	SLQP

Bit4 1: external chuck control is enabled when channel 1's programs run
0: external chuck control is disabled when channel 1's programs run
Bit2 1: channel 1 uses the outer chuck mode
0: channel 1 uses the inner chuck mode
Bit1 1: do not check whether the chuck is clamped when channel 1's chuck function is enabled
0: check whether the chuck is clamped when channel 1's chuck function is enabled. Cannot start the spindle and an alarm occurs when the chuck is not clamped
Bit0 1: channel 1's chuck control function is enabled
0: channel 1's chuck control function is disabled

K	1	3	CHIPR	***	WKLP	***	***	RTSC	SPTW	SLTW

Bit7 1: The chip-removal controller is enabled
0: The chip-removal controller is disabled
Bit5 1: Indicator control is enabled
0: Indicator control is disabled
Bit2 1: External tailstock control is enabled during program running
0: External tailstock control is disabled during program running
Bit1 1: the spindle rotation and the tailstock forward/backward does not interlock, the tailstock executes the forward no matter what the spindle is in any states; the spindle rotates no matter what the tailstock is in any states
0: the spindle rotation and the tailstock forward/backward interlock, the tailstock cannot execute the forward when the spindle rotates; the spindle cannot be started when the tailstock does not execute the forward
Bit0 1: the tailstock control function is enabled
0: the tailstock control function is disabled

K	1	4	HPST	***	***	SGSP	SPB4	PB4	SPB3	PB3

Bit7 1: the hydraulic control function is enabled
0: the hydraulic control function is disabled
Bit4 1: Spindle is enabled when the protection door is open
0: Spindle is disabled when the protection door is open
Bit3 1: when SAGT is connected with +24V, the safety door is disabled
0: when SAGT is not connected with +24V, the safety door is disabled
Bit2 1: the safety door check function is enabled
0: the safety door check function is disabled
Bit1 1: when PRES is not connected with +24V，the system alarms for pressure LOW
0: when PRES is connected with +24V，the system alarms for pressure LOW
Bit0 1: the pressure LOW check function is enabled
0: the pressure LOW check function is disabled

K	1	5	RCS	RSCS	SPOR	AGOT	SHT	***	***	AGER

Bit7 1: channel 1's Cs function is enabled
0: channel 1's Cs function is disabled
Bit6 1: the spindle contour control is closed when channel 1's emergency stop resets
0: the spindle contour control is not closed when channel 1's emergency stop resets
Bit5 1: the spindle eight-point orientation function of channel 1 is enabled
0: the spindle eight-point orientation function of channel 1 is disabled
Bit4 1: PATH 1 spindle automatic gearing finished,solenoid valve output is closed
0: PATH 1 spindle automatic gearing finished,solenoid valve output is unchanged
Bit3 1: PATH 1 the spindle gear power-down memory
0: PATH 1 the spindle gear power-down does not execute the memory
Bit0 1: PATH 1 the spindle automatic gear change function is enabled
0: PATH 1 the spindle automatic gear change function is disabled

K	1	6	SINC	***	SCWM	LUBS	LUBCK	M32A	***	SOVI

Bit7 1: PATH 1 gear "×1000" in STEP (MPG) mode is disabled
0: PATH 1 gear "×1000" in STEP (MPG) mode is enabled

Bit5 1: PATH 1 when spindle stop command is in the same block with movement command, these commands are executed at the same time
0: PATH 1 when spindle stop command is in the same block with movement command, movement command comes first to spindle stop command
Bit4 1: Alarm for lubrication oil at low level
0: Alarm for lubrication oil at high level
Bit3 1: Lubrication oil detection valid
0: Lubrication oil detection invalid
Bit2 1: the lubricating outputs in power-on when the automatic lubricating is enabled
0: the lubricating does not output in power-on when the automatic lubricating is enabled.
Bit0 1: the external feedrate override switch is enabled
0: the external feedrate override switch is disabled

K 1 7	***	SCLP	CKSLM	SALM	SSTP	***	***	SPDIR

Bit6 1: PATH 1 Spindle clamp function is valid
0: PATH 1 Spindle clamp function is invalid
Bit5 1: PATH 1 Alarm for spindle detect
0: PATH 1 Alarm for spindle don't detect
Bit4 1: PATH 1 Alarm for spindle at low level
0: PATH 1 Alarm for spindle at high level
Bit3 1: PATH 1 When spindle stop,the output voltage closed
0: PATH 1 When spindle stop,the output voltage remain
Bit0 1: PATH 1 Spindle key in panel reverse
0: PATH 1 Spindle key in panel unreverse

K 1 8	***	***	***	***	MDOK	MD4	MD2	MD1

Bit3 1: the operation mode is specified by MD4, MD2, MD1 when channel 1's power-on;
0: the operation mode is the power-on state when channel 1's power-on

MD4	MD2	MD1	Operation mode in power-on
0	0	0	MDI mode
0	0	1	Auto mode
0	1	0	Program zero return
0	1	1	Edit mode
1	0	0	MPG mode
1	0	1	Manual mode
1	1	0	Machine zero return

K 1 9	DBPH	MACH	HDFD	HDIT	***	***	STPH	PATH

Bit7 1: dual channel function is enabled
0: dual channel function is disabled
Bit6 1: The machine tool panel is the vertical one
0: The machine tool panel is the horizontal one
Bit5 1: MPG interruption is enabled in operation
0: MPG interruption is disabled in operation
Bit4 1: Handle interruption is valid
0: Handle interruption is invalid
Bit1 1: all channels are enabled when the cycle start key on panel is pressed

0: the current channel is enabled when the cycle start key on panel is pressed

Bit0 1: external channel selection signal is enabled

0: external channel selection signal is disabled

K	2	0	LMIT	LMIS	***	JSPD	OVRI	***	RSJG	RRW

Bit7 1: PATH 2 Overtravel is valid

0: PATH 2 Overtravel is invalid

Bit6 1: PATH 2 Alarm for overtravel signal at low level

0: PATH 2 Alarm for overtravel signal at high level

Bit4 1: spindle jog in any modes of channel 2 is enabled

0: spindle jog in Manual, MPG, Zero return mode of channel 2 is enabled

Bit3 1: feedrate override of channel 2 is fixed to 100%

0: feedrate override of channel 2 can be adjusted

Bit1 1: the spindle lubricating, cooling output remains when channel 2 resets

0: the spindle lubricating, cooling output closes when channel 2 resets

Bit0 1: the cursor returning to program home is enabled in any modes when channel 2 resets

0: the cursor returning to program home is enabled in Edit mode when channel 2 resets

K	2	1	***	***	CHET	TCPS	CTCP	TSGN	CHTB	CHTA

Bit5 1: check tool signal when channel 2 completes tool change

0: do not check tool signal when channel 2 completes tool change

Bit4 1: tool post lock signal of channel 2 is HIGH (connected with +24V) is enabled

0: tool post lock signal of channel 2 is LOW (disconnected with +24V) is enabled

Bit3 1: channel 2 detects tool post lock signal

0: channel 2 does not detect tool post lock signal

Bit2 1: channel 2's tool signal LOW (disconnected with +24V) is enabled

0: channel 2's tool signal HIGH (connected with +24V) is enabled

Bit1 Bit0 : channel 2's tool change mode selects bit 1 tool change mode selects bit 0

CHTB	CHTA	Tool post type
0	0	Standard tool change mode B
0	1	Standard tool change mode A

K	2	2	***	***	***	RCKC	***	NYQP	SLSP	SLQP

Bit4 1: external chuck control is enabled when channel 2's programs run

0: external chuck control is disabled when channel 2's programs run

Bit2 1: channel 2 uses the outer chuck mode

0: channel 2 uses the inner chuck mode

Bit1 1: do not check whether the chuck is clamped when channel 2's chuck function is enabled

0: check whether the chuck is clamped when channel 2's chuck function is enabled. Cannot start the spindle and an alarm occurs when the chuck is not clamped

Bit0 1: channel 2's chuck control function is enabled

0: channel 2's chuck control function is disabled

K	2	5	RCS	RSCS	SPOR	AGOT	SHT	***	***	AGER

Bit7 1: PATH 2 Cs function is valid

0: PATH 2 Cs function is invalid

Bit6 1: PATH 2 when the system reset,the spindle contour control cancel
0: PATH 2 when the system reset,the spindle contour control unchanged
Bit5 1: PATH 2 Spindle orientation is valid
0: PATH 2 Spindle orientation is invalid
Bit4 1: PATH 2 Spindle automatic gearing finished,solenoid valve output is closed
0: PATH 2 Spindle automatic gearing finished,solenoid valve output is unchanged
Bit3 1: PATH 2 The spindle gear power-down memorizes
0: PATH 2 The spindle gear power-down don't memorize
Bit0 1: PATH 2 Spindle automatic gearing is valid
0: PATH 2 Spindle automatic gearing is invalid

K 2 6	SINC	***	SCWM	***	***	***	***	***

Bit7 1: when channel 2's system increment is 0.1u and the system is in single step(MPG) mode, ×1000 is disabled
0: when channel 2's system increment is 0.1u and the system is in single step (MPG) mode, ×1000 is enabled
Bit5 1: they are simultaneously executed when channel 2's spindle stop command and movement command are in the same block
0: the spindle does not stop till movement command execution ends when channel 2's spindle stop command and movement command are in the same block

K 2 7	***	***	CKSLM	SALM	SSTP	***	***	SDIR

Bit5 1: PATH 2 Alarm for spindle detect
0: PATH 2 Alarm for spindle don't detect
Bit4 1: an alarm occurs because channel 2's spindle is LOW
0: an alarm occurs because channel 2's spindle is HIGH
Bit3 1: close analog voltage when channel 2's spindle stops
0: does not close analog voltage when channel 2's spindle stops
Bit0 1: PATH 2 Spindle key in panel reverse
0: PATH 2 Spindle key in panel unreverse

K 2 8	***	***	***	***	MDOK	MD4	MD2	MD1

Bit3 1: the operation mode is specified by MD4, MD2, MD1 when channel 2's power-on
0: the operation mode is the power-on state when channel 2's power-on

MD4	MD2	MD1	Operation mode in power-on
0	0	0	MDI mode
0	0	1	Auto mode
0	1	0	Program zero return
0	1	1	Edit mode
1	0	0	MPG mode
1	0	1	Manual mode
1	1	0	Machine zero return

K	3	0

***	***	***	***	SLPCB	SLPCA	***	***

Bit3 1: The spindle encode single for path 2 using from path 1
0: The spindle encode single for path 2 using from path 2
Bit2 1: The spindle encode single for path 1 using from path 2
0: The spindle encode single for path 1 using from path 1

3.3.2 DT parameters

Address	Meaning
DT000	Time for previous gear disabling during spindle automatic gear change (ms) of channel 1
DT001	Delay time after the changed spindle gear is in position (ms) of channel 1
DT002	Detection time for low pressure alarm (ms)
DT004	Permit time for tool change (ms) of channel 1
DT005	M command execution time (ms) of channel 1
DT006	S command execution time (ms) of channel 1
DT007	Delay time for tool post from CW to CCW output (ms) of channel 1
DT008	Detection time (ms) for tool post lock signal of channel 1
DT009	Tool post CCW rotation lock time (ms) of channel 1
DT010	Spindle braking delay output time (ms) of channel 1
DT011	Spindle braking output time (ms) of channel 1
DT012	Spindle JOG time (ms) of channel 1
DT013	Manual lubricating ON time (0: time is not limited) (ms)
DT014	Chuck clamping command execution time (ms) of channel 1
DT015	Chuck releasing command execution time (ms) of channel 1
DT016	Interval time of automatic lubricating (ms) of channel 1
DT017	Time for automatic lubrication output (ms) of channel 1
DT018	Chuck pulse output width (ms) of channel 1
DT020	Spindle clamping/releasing time (ms) of channel 1
DT021	Spindle stop, chuck enabling delay time (ms) of channel 1
DT023	Delay time to reduce spindle servo impel time (ms) after spindle starts clamping of channel 1
DT025	PATH 1 Cycle time of spindle asway as gear shift(ms)
DT030	Time for previous gear disabling during spindle automatic gear change (ms) of channel 2
DT031	Delay time after the changed spindle gear is in position (ms) of channel 2
DT034	Permit time for tool change (ms) of channel 2
DT035	M command execution time (ms) of channel 2
DT036	S command execution time (ms) of channel 2
DT037	Delay time for tool post from CW to CCW output (ms) of channel 2
DT038	Detection time (ms) for tool post lock signal of channel 2
DT039	Tool post CCW rotation lock time (ms) of channel 2
DT040	Spindle braking delay output time (ms) of channel 2
DT041	Spindle braking output time (ms) of channel 2
DT042	Spindle JOG time (ms) of channel 2
DT044	Chuck clamping command execution time (ms) of channel 2
DT045	Chuck releasing command execution time (ms) of channel 2
DT048	Chuck pulse output width (ms) of channel 2

Address	Meaning
DT051	Spindle clamping/releasing time (ms) of channel 2
DT055	PATH 2 Cycle time of spindle asway as gear shift(ms)

3.3.3 DC parameters

Address	Meaning
DC000	Spindle zero speed output range (r/min) of channel 1
DC001	Spindle zero speed output range (r/min) of channel 2

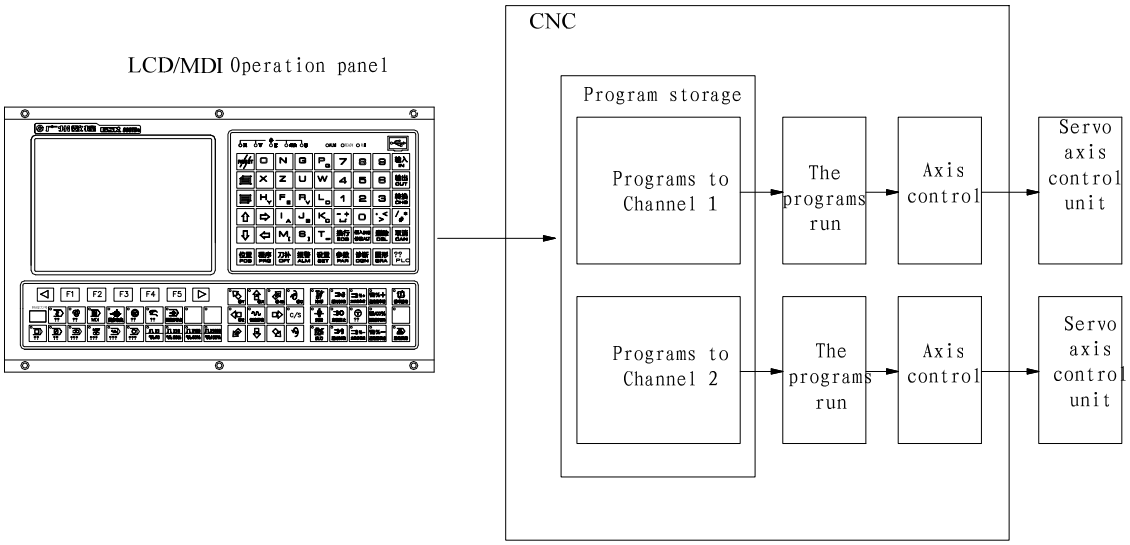
CHAPTER 4 DUAL-CHANNEL FUNCTION

4.1 Summary

GSK980TTi can realize the dual-channel control which control axes up to 5 (including Cs axis). The dual-channel control function is to simultaneously machine two workpieces alone. The user should firstly select programs of each channel used to machine, and then start the machine program of each channel to execute the machining.

For all channels, there is only one set of LCD, edit keyboard, machine panel. Switch the channel selection signal to execute the LCD, edit keyboard, machine panel's display and operation of corresponding channel.

Example: dual-channel system control is shown below:



4.1.1 System constitution

The system with dual-channel control has up to 2 channels, 6 control axes (including Cs axis) and 2 spindles, which consist of a two-channel system.

4.1.2 Channel setting

The system supports up to 2 channels, selects the max. control channel number of the current system by setting No.0223. When the max. control channels are 2, the system controls output with dual channels; when it is set to 1, the system controls output with a single channel.

Note: When it is set to a single channel, it is invalid to No.0227, No.0228, No.0229 relevant with the dual channel.

4.1.3 Control axis setting

The system's control axes are 6 (including Cs axis), and each channel controls up to 5 axes. No. 0227 and No.0229 distribute channels and axis number to each axis.

Example: for dual channel control, Channel 1 controls 3 axes, and its axis number is the 1st, 2nd, 5th axis, Channel 2 controls 3 axes, and its axis number is the 1st, 2nd ,5th axis.

№0227	№0229	Distribution
1	1	The 1st axis of Channel 1
1	2	The 2nd axis of Channel 1
2	1	The 1st axis of Channel 2
2	2	The 2nd axis of Channel 2
1	5	The 5th axis of Channel 1
2	5	The 5th axis of Channel 2

4.1.4 Spindle setting

The system's spindle has up to 3, each channel has up to control 2 spindles and each spindle's channel is allocated by No. 0228. When some channel's control spindles are 2, №196 Bit4 (multi-spindle function is enabled) is set to 1.

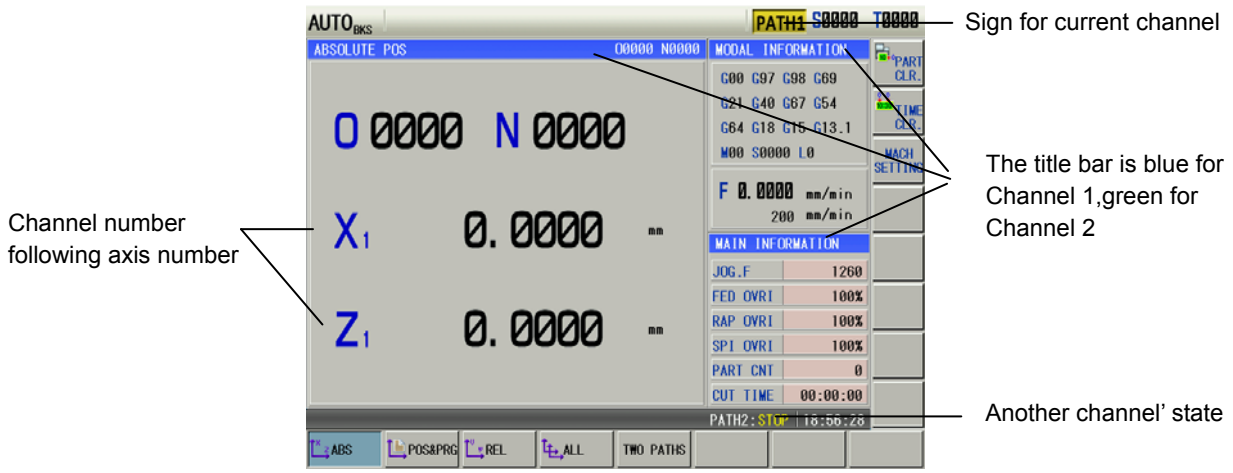
Example: For dual-channel control, the Channel 1 controls the 1st spindle and the Channel 2 controls the 2nd spindle, which is shown below:

№0228		Distribution
S1	1	The Channel 1's the 1st spindle
S2	2	The Channel 2's the 1st spindle

4.1.5 Operation display

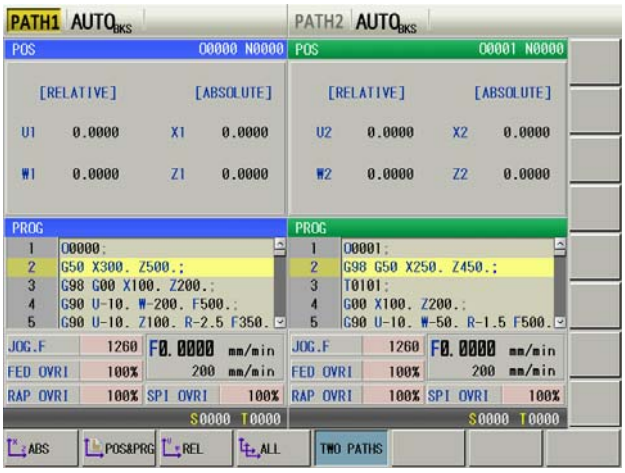
As a extended version of GSK980TTi, GSK980TTi's basic operation methods are the same those of GSK980TTi, but its some windows have been still changed a little.

Its display window's data is the current which is changed by switching channels, and some messages are added on windows used to identify the current channel and the channel's state, which is shown below:



Note: The displayed coordinate axis with the channel number on the window is taken as a sign, but the sign is not added when actual programming, and the program format is the same that of GSK980TTi.

Dual-channel display position window is used to simultaneously display workpiece coordinates, machine programs and some compound messages of the dual channels.



Setting, parameter, diagnosis and PLC ladder window display operations of the whole system, operations are executed in any one channel and are enabled to other channels. For example, a program switch is closed in Channel 1, and programs cannot be edit in other 2 channels. All parameters can be modified in Channel 1, including parameters relevant to Channel 2, so dual channels must be in stop state when parameters are modified.

4.1.6 Dual-channel special function

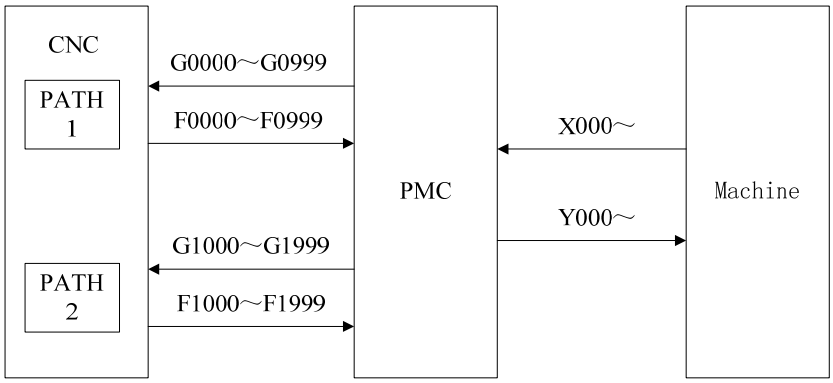
The dual-channel special function has the following concrete functions:

1) Channel selection and display
2) Interchannel common memory
3) Wait for M code function
4) Interchannel spindle control function
5) Interchannel interference check function
6) Balanced cutting
7) Synchronous control
8) Mix control
9) Overlap control
10) Imaginary axis control
11) Interchannel single block check function

Concrete characteristics of each function are referred to its explanations.

4.2 Function description

Each channel in GSK980TTi has separate F/G signal, the F/G singal of channel 1 is exactly the same as GSK980TTi, the F/G singal of channel 2 is incremented by 1000 at the address of channel. which is shown below:



For the PLC alarm, with the alarm number to distinguish:

1000～1999 : PATH 1's alarm;

2000～2999 : nothine to do with the channel;

3000～3999 : with both channels；

4000～4999 : PATH 1's alarm;

Note: channel-related alarms are only displayed on the channel's page.

4.2.1 Channel selection and display

For all channels, there is only one set of LCD, keyboard, machine panel. Display and setting of all kinds of data, program input in MDI mode, program edit in program memory of each channel can be switched by channel selection signals.

Channel selection signal HEAD<G063#0>	Selected channel
0	Channel 1
1	Channel 2

4.2.2 Interchannel common memory

In the dual-channel system, some data is taken as common data of all channels to store.

4.2.2.1 User macro program common variable

Common macro variable range of each channel is #100～#199， #500～#799, data of #100～#199 is cleared after power-off, data of #500～#799 is saved after power-off.

Macro variable of #100～#129，#500～#599 in common macro variable are used by the dual channels. Macro variable of #130～#199，#600～#799 is used by each channel alone, which is shown below:

Macro variable type	Common part	Separate part	All channels
Do not memory after power-off	#100～#129	#130～#199	Channel 1
		#130～#199	Channel 2
Memory after power-off	#500～#599	#600～#799	Channel 1
		#600～#799	Channel 2

4.2.3 Wait for M code function

M code controls wait in the course of processing of Channel 1 and Channel 2. M code used to wait in some channel is executed when automatic run, the system does not pause the next execution till the same M code is executed in other channel, and then the system starts to execute the next block.

Format：M__;

Code range：M100～M199

When only one channel is needed to run, the wait M code is ignored not to delete in programs.

- **Notes:**
  1. Command the wait M code in single blocks.
  2. The wait M code is different with other miscellaneous M codes, does not output signal to PLC and does not refresh modal message of M code on the new window.
  3. An alarm occurs when unmatched M codes in Channel 1 and Channel 2 are specified.
- **Signal explanation**

Wait ignorance signal NOWT<G0063.1>（channel's common signal interface）

NMWT<Gn063.7>（channel's alone signal interface）

［Classification］input signal

［Function］ specify whether to wait based on M code.

［Operation］ wait not be based on M code when the signal is "1" . ignore the commanded wait M code in the course of machining.

Execute the wait based on M code path when the signal is '0'. When the wait M code is commanded in some path, other paths are waited and are commanded by the same M code, and then the next block is executed.

Waiting signal WATO<Fn063.6>

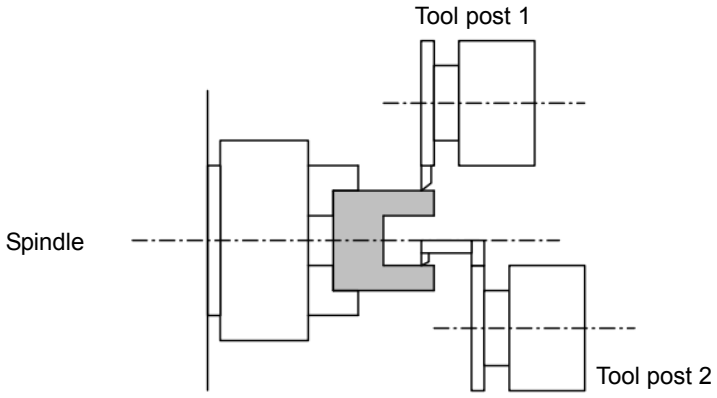
［Classification］output signal

［Function］ the signal informs that PLC's each channel is based on M code wait.

［Output］ when interchannel is waiting, i.e. after the wait M code is commanded, during the same M code being specified at another channel, the signal is "1". The signal is "0" when interchannel is not waited.

4.2.4 Interchannel spindle control function

Interchannel spindle control function can make the spindle command in some channel control a spindle in another channel, or make some channel use the spindle encoder feedback signal of another channel, which ensure two tool posts machine the workpiece mounted on one spindle.



Generally, the spindle control command in each channel control corresponding spindle of each channel, but using commands in interchannel can control the spindle which does not belong the corresponding channel by interchannel spindle command selections signal.

Besides, when feed per rev or thread cutting in each channel is executed by position encoder feedback pulse of spindle belonging to the same channel, so, is executed by position encoder feedback pulse of spindle belong to other channel.

- **Interchannel spindle control signal：**

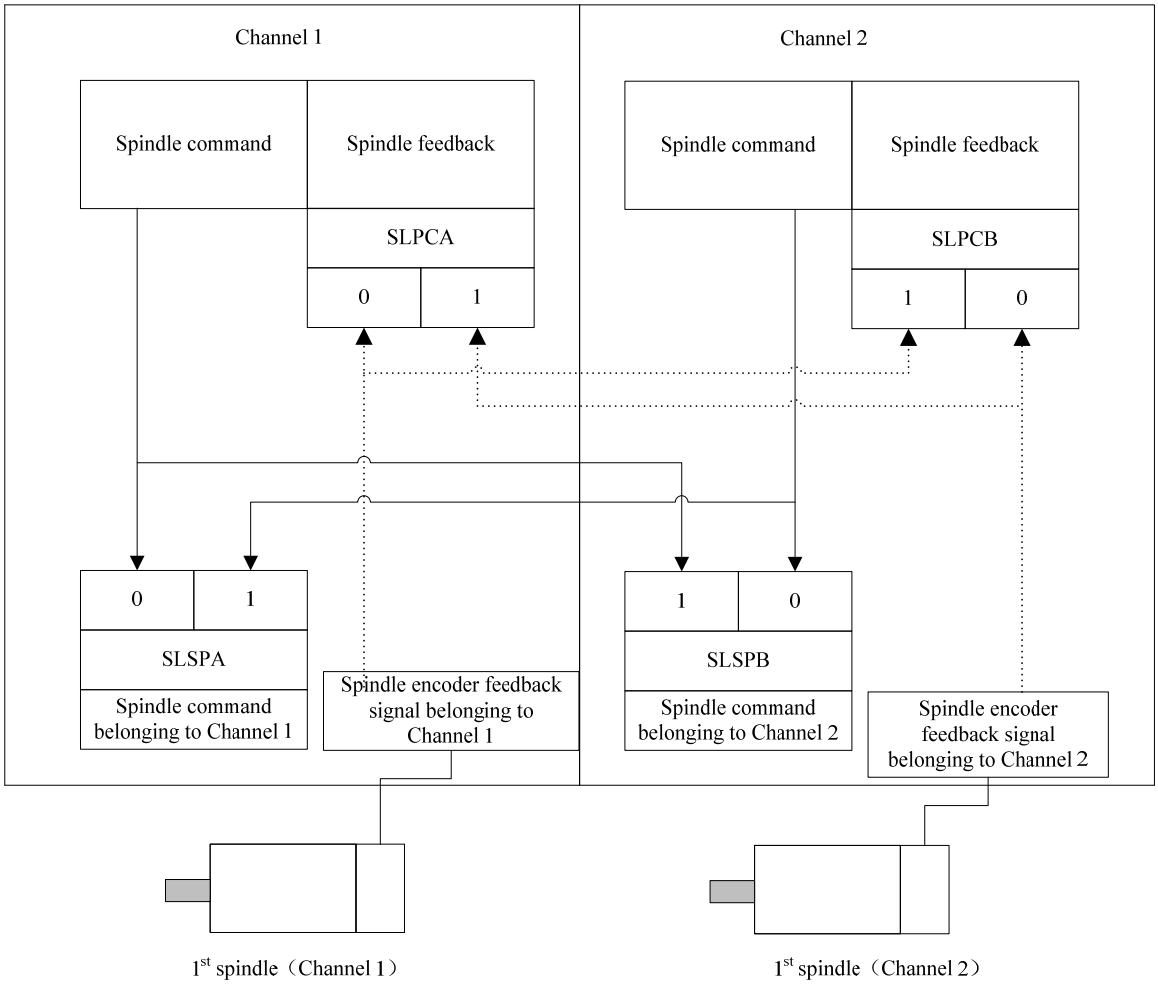
By interchannel spindle command selection signal SLSPn(input), selection commanded by spindle switches the spindle of corresponding channel to control the spindle command of corresponding channel.

By interchannel spindle feedback selection signal SLPCn (input), position encoder feedback signal switches position encoder feedback pulse of spindle belonging to corresponding channel, so, using the spindle belonging to different channels executes thread cutting and feed per rev.

Note 1: the spindle command is defined to S command , max. speed command (G50S), and constant speed command (G96, G97).

Note 2: G97 is modal during switching spindle command selection, S speed command is valid the next time; linear speed control is immediately valid when G96 is modal.

Control sketch map is shown below:



- **Notes**

①、Spindle commands include S command, max. speed command(G50S), constant speed control command (G96, G97).

②、When the spindle command selection is switched, PLC is defined whether to switch M3, M4, M5 to control channels. For example, to Channel 1 controlling channel 2's spindle, when pressing M3 S500 switches the spindle command selection, the NC executes S500 to switch channels, M3 switch is defined in the PLC.

③、When the spindle command selection is switched and G97 is modal, S speed command is valid the next specified; when G96 is modal, linear speed control is immediately valid.

④、When the spindle feedback selection is switched, the data in switch period is not consecutive because they come from different encoders, so, the spindle feedback selection switch should be avoid in G99 or thread interpolation.

- **Signal explanation**

Interchannel spindle command selection signal SLSPA<G0063.2>，SLSPB<G0063.3>

［Classification］input signal

［Function］ Specify the spindle's control channel of each channel.

［Operation］

Signal input SLSPA<G0063.2>	Actual control channel of spindle belonging to Channel 1
0	Channel 1
1	Channel 2

Signal input SLSPB<G0063.3>	Actual control channel of spindle belonging to Channel 2
0	Channel 2
1	Channel 1

Interchannel spindle feedback selection signal SLPCA<G0064.2>，SLPCB<G0064.3>

［Classification］input signal

［Function］ specify the channel of position encoder feedback pulse read by each channel.

［Operation］

Signal input SLPCA<G0064.2>	Channel belonged to reading position encoder pulse in Channel 1
0	Channel 1
1	Channel 2

Signal input SLPCB<G0064.3>	Channel belonged to reading position encoder pulse in Channel 2
0	Channel 2
1	Channel 1

Interchannel spindle command confirmation signal COSP<F0064.5>

［Classification］output signal

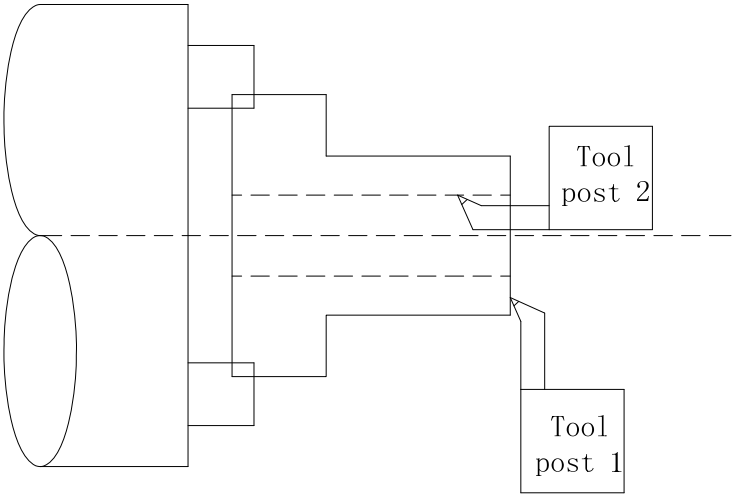
［Function］ the signal informs the channel of the last executed spindle command.

［Output］ the signal is “1” when the spindle command is executed at side of Channel 2. the signal is “0” when the spindle command is executed at side of Channel 1 or through none of channel.

Note: when the spindle belonging to some channel is controlled by the spindle command from Channel 1 or 2, the signal can confirm the last executed spindle command’s channel.

4.2.5 Interchannel interference check

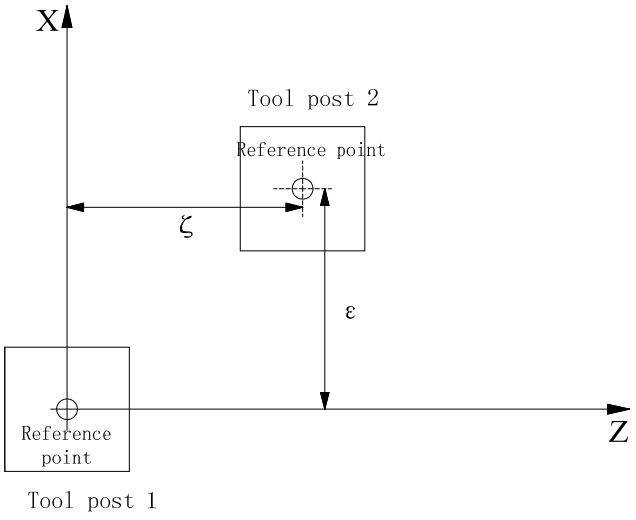
Some time, a machine with a dual channel control system needs the tool post controlled by each channel to machine the same workpiece, at the moment, the two tool posts approach very near, so, mistaken programs or other mistaken setting may cause the tool be damaged or other accidences because of tools touching each other. Interchannel tool path interference check function can make tools stop running before touching each, which can avoid the unnecessary losses.



For interchannel tool interference, relative relationship between two tool posts, tool post/tool shape and other data should be preset. Besides, set basic X, Z axis in each channel.

4.2.5.1 Relative position relationship setting of interchannel tool posts

To confirm relationship of two tool posts, X, Z of each channel must execute machine zero return. When the machine zero return is done, reference point of the tool post 2 is measured in ZX coordinate system which origin is the tool post 1's reference point, which is shown below:



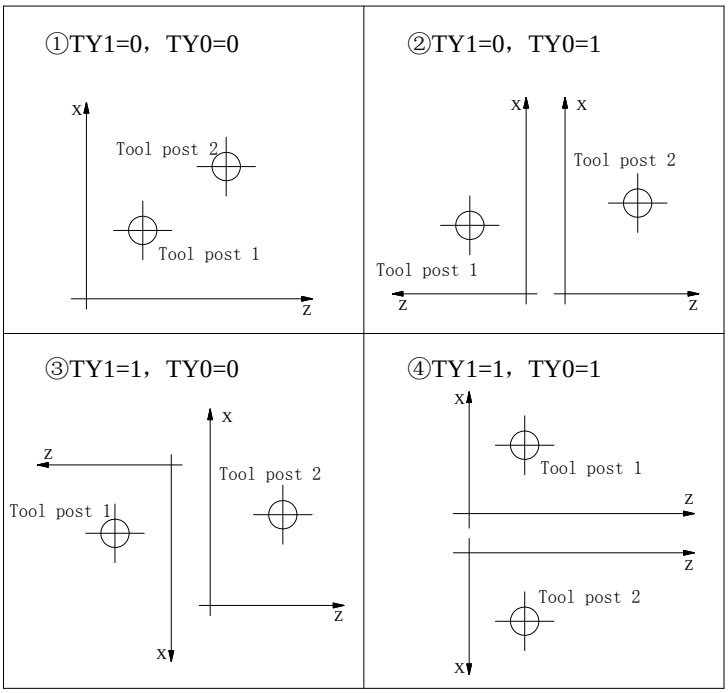
The X component value ε of tool post 2's reference point in ZX coordinate system which origin is the tool post 1's reference point is measured, and the measured value is set to No.232, The Z component value ζ of tool post 2's reference point in ZX coordinate system which origin is the tool post 1's reference point is measured, and the measured value is set to No.233. The axis with diameter programming is set with diameter value, and is set with metric/inch, system increment of Channel 2.

When relative position parameter of the two tool posts is set again, X, Z of the two channels must

machine zero return, otherwise, their relative position cannot be refreshed timely to the newly set parameter value.

4.2.5.2 Coordinate system setting relevant with interchannel interference check

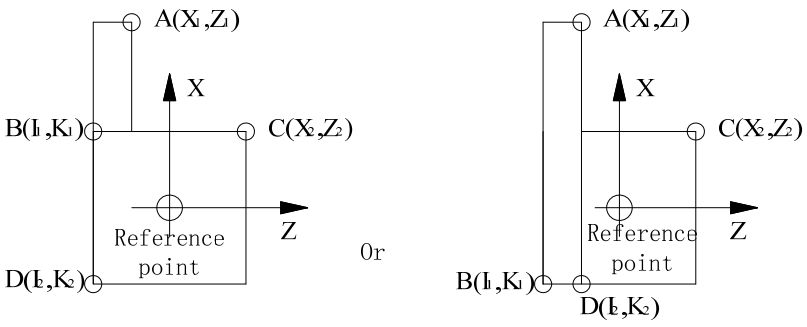
Although the relative position relationship between the two tool posts, their relative motion directions cannot be confirmed (i.e. the two tool posts approach or are far away from each other), so, setting No.231 TY0, TY1 can confirm machine coordinate system's relationship of the two tool posts. According to machine installation method, there are four forms as follows:



4.2.5.3 Tool shape data setting explanation

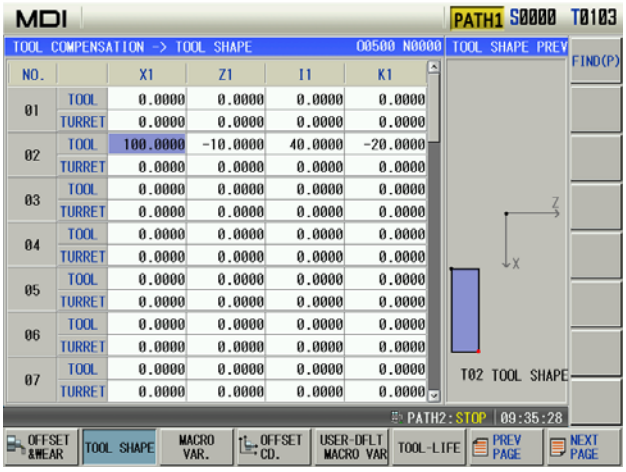
The CNC provides 32 tool offset numbers for each channel, so it does one group of tool shape data input value for each tool offset number.

Contact forbidden zone consisted of tool posts and set tools is divided into two rectangular region (region 1, 2). Input a coordinate value of 4 feature points $A(X_1,Z_1)$, $B(I_1,K_1)$, $C(X_2,Z_2)$, $D(I_2,K_2)$ common owned by the two rectangular regions in the tool shape data. which is shown below:

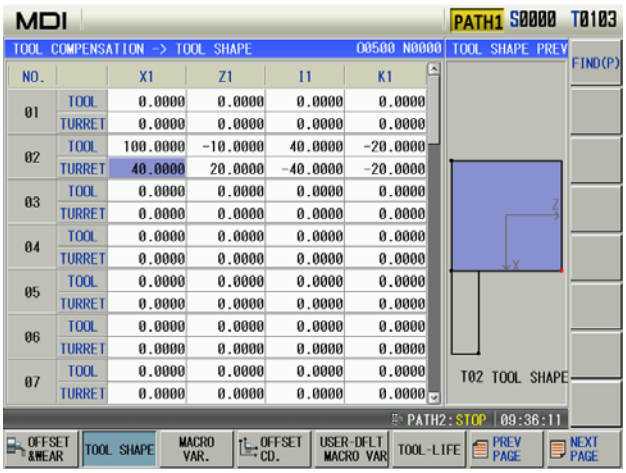


4.2.5.4 Tool shape data setting method

（1） Press  to enter the tool offset interface, press  key to enter the tool shape setting window, move the cursor to select the required set serial number, such as No.02, Region 1, press the address key to input numerical value as follows:



（2） As above-mentioned method, input point coordinates in Region 2, preview the tool shape at right window as follows:



Note: X input value is relevant with diameter/radius programming method. It is a diameter value when the diameter programming is executed, and is a radius value when radius programming is done.

4.2.5.5 Conditions for interchannel interference check

（1） Set the interchannel interference check function to be valid, and set IFE（NO231 bit4）to 1;;

（2） Each channel must set basic axis X, Z;

（3） Execute X, Z of each channel to return reference point after power on;

（4） Each path must go through non zero tool offset number specified by T code;

（5） Input a value to one of tool shape data's regions corresponding to commanded tool offset numbe;

（6） When IFM（NO231 bit3）is set to 1, interchannel interference check is executed in Manual mode;

4.2.5.6 Notes

（1） For the interchannel interference check, the input tool offset number of tool shape data must be consistent with the tool offset number specified by programs. It is difficult to execute correctly the interference check when manual run selects a tool or the tool selection command is not executed after power on;

（2） Setting parameters and tool shape data (contact forbidden zone) used to the interference check function must be in Manual mode (parameters for valid interference check must be preset in Manual mode), which make tool post interference of each channel, and then the normal interference region is confirmed correctly.

4.2.5.7 Signal explanation

Interchannel interference checking signal TICHK<F0064.6>

［Classification］output signal

［Function］the signal informs whether PLC is performing the interchannel interference check

［Output］ when all required conditions for the interchannel interference check on the tool posts in Channel 1 and in Channel 2 are provided, the signal becomes ‘1’, otherwise, it does ‘0’.

Interchannel interference alarm signal TIALM<F0064.7>

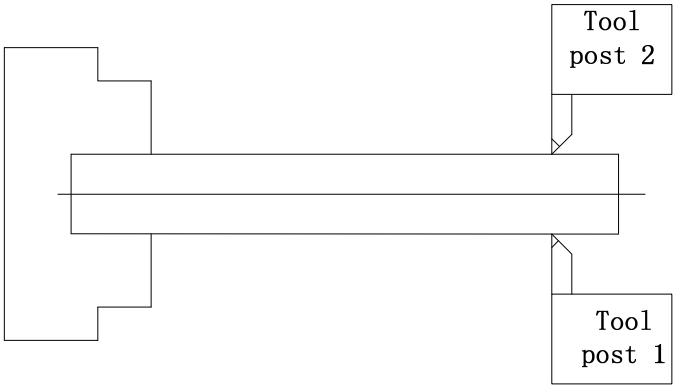
［Classification］output signal

［Function］the signal informs PLC interchannel interference check alarm

［Output］ when the interchannel interference check on the tool posts in Channel 1 and in Channel 2 can judge that the two tool posts interfere each other, the signal becomes ‘1’; when the interchannel interference check on the tool posts in Channel 1 and in Channel 2 can judge that the two tool posts do not execute interference or the interchannel interference check is not executed (i.e. when the interchannel interference signal TICHK is ‘0’), the signal becomes ‘0’.

4.2.6 Balanced cutting

When a tiny workpiece is machined, two turning tools align at the two sides of the workpiece to simultaneously machine, which can avoid a bent workpiece caused by single side machine, and get precise machining (see below) . Besides, when the two tools’ cutting movement is not exactly adjusted synchronously, the workpiece vibrates and is not correctly machined. The function can synchronously start the tool posts’ cutting movement.



- **Command format**

G68 start the balanced cutting mode
G69 cancel the balanced cutting mode

- **Function explanation**

The balanced cutting mode is started when G68 is commanded in programs belonging to Channel 1 and 2; the balanced cutting mode is cancelled when G69 is specified. Programs wait to be executed when G68/G69 is specified only one channel.

In the balanced cutting mode, the balanced cutting is executed when movement commands for cutting feed in two channels are specified and the movement amount is not 0. The balanced cutting is not executed when rapid traverse command is executed.

In the balanced cutting mode, for blocks specifying cutting feedrate traverse commands, programs are simultaneously started to execute.

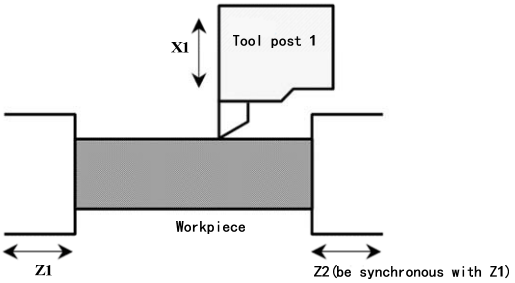
The balanced cutting is not executed when any one channel is in Dry run mode, machine lock state. But, programs wait when G68/G69 is executed.

- **Notes**

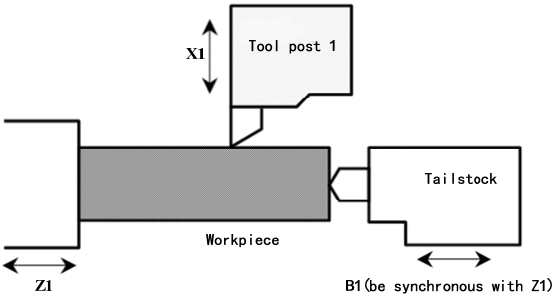
1、Specify G68 or G69 in single blocks.
2、Programs do not wait when G68 is executed in the balanced cutting start mode or G69 is executed in the balanced cutting cancel mode.
3、The balanced cutting function makes cutting feeds in the two channels are simultaneously executed. Feedrate commands, movement amount, feedrate, feedrate override in the two channels are set to the same one to ensure the two tool posts synchronously move.
4、The balanced cutting is not executed when it is started again and feed pause is performed in the two channels in the course of the balanced cutting, it is executed to wait in the next block specifying cutting movement commands.
5、The balanced cutting function is cancelled when emergency stop, reset is executed.

4.2.7 Synchronous control

The synchronous control function makes a axis belonging to other channel or in the same channel with some axis move synchronously, which is shown below:



Z2（slave control）axis be synchronous with Z1 (main control)axis



B1（slave control）axis be synchronous with Z1 (main control)axis

- **Synchronous control setting:**

Set the synchronous main control axis' axis number of each in No. 210. Its setting method: main control axis path number * 100+relative axis number in its path.

Example: when axis allocation is X (the 1st axis), Z (the 2nd axis):

Z2（slave control）axis is synchronous with Z1（main control）axis, №210_{Z2} = 102

X1 （slave control）axis is synchronous with X2（main control）axis, №210_{X1} = 201

- **Synchronous control signals:**

Synchronous control start/release is executed at side of PMC by setting the slave control axis's synchronous control selection signals SYNC1～SYNC5. When SYNC1～SYNC5 becomes '1' from '0', the synchronous control is started; the synchronous control is released when it becomes '0' from '1'

When some axis' movement is required to stop state in synchronous control, PK1～PK5 is set to '1' not to execute actual movement, but whether to refresh the workpiece coordinate system is set by parameters.

Note: the synchronous control start/release is to check edge signal of SYNCn signal, i.e. SYNCn rising edge is checked when the synchronous control is started, and SYNCn falling edge is checked when the synchronous control is released

- **Synchronous control commands:**

Main control axis' path	Slave control axis' path	
:	:	
M100;	M100;	wait
	M80;	start the synchronous control
M101;	M101;	wait
:	:	
Separate run	Separate run	
:	:	
M100;	M100;	wait
	M81;	release the synchronous control
M101;	M101;	wait
:	:	

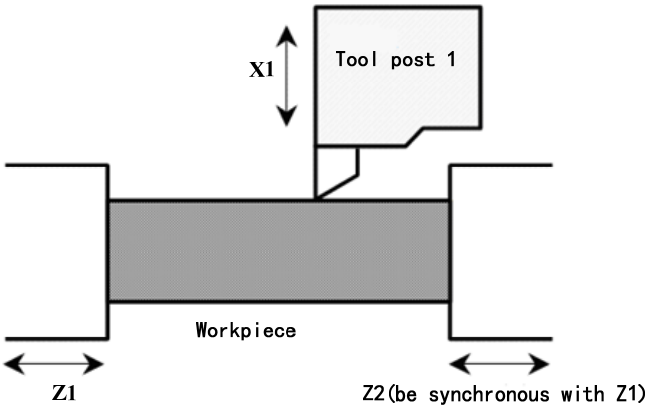
Note 1: The synchronous control start/release command M80/M81 is used to control change of synchronous control selection signal SYNCn. (Here, M80/M81 is a sample command and the actual function command is required to define in PLC according to machine allocation).

Note 2: The wait M code before starting/releasing the synchronous command is added to pause program read, which can ensure synchronous control axis data is normally processed after starting/releasing the synchronous control.

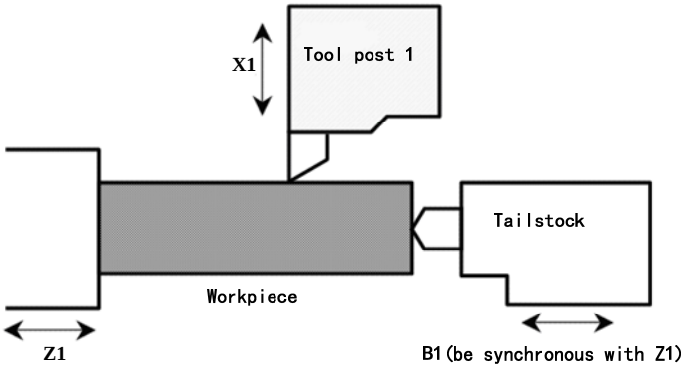
- **Example：**

1. Any one axis is synchronous with other axis to move（the main axis and the slave axis both move）

（Example 1）Z2（slave control）axis is synchronous with Z1（main control）axis

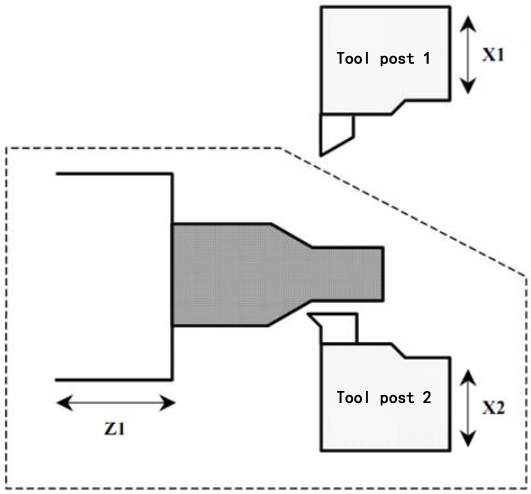


（Example 2）B1（slave control）axis is synchronous with Z1 (main control) axis



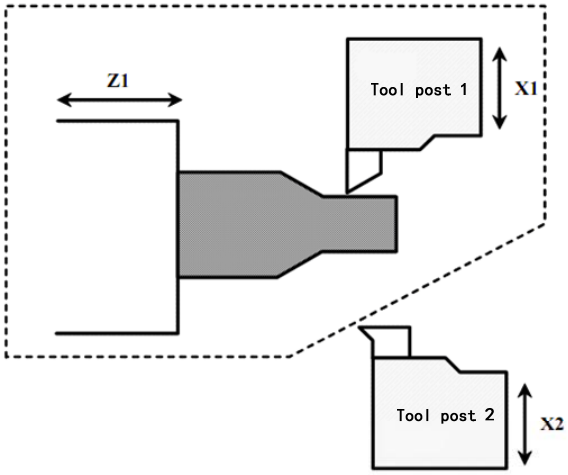
2. Some axis does not move by parking. Using parking signal sets movement of the synchronous main control axis to stop state, which is called main control parking. In this case, only slave axis moves. Oppositely, setting the synchronous slave control axis to stop state is called slave control parking. Under the circumstances, only main control axis moves.

（Example 3）Z1 axis and Z2 axis share one motor.（suppose the motor is connected with Z1 axis, the workpiece coordinate system is refreshed when parking is set）



Main control parking：

Make Z1 （ slave control ） axis be synchronous with Z2（main control）axis, preset Z2（main control）to parking state. Command X2 and Z2 axes to move in Channel 2. At the moment, Z1 be synchronous with Z2 axis moves and Z2 axis is parked, so, the actual is X2 and Z1 to move.



Slave control parking：

Make Z2 （ slave control ） axis be synchronous with Z1（main control）axis, preset Z2（main control）to parking state. Command X1 and Z1 axes to move in Channel 1. At the moment, Z2 be synchronous with Z1 axis moves and Z2 axis is parked, so, the actual is Z2 does not move and the coordinate system is refreshed.

In the above main control parking and the slave control parking, coordinates of Z1, Z2 are refreshed, no changing the coordinate system can execute movement command when the synchronous state is switched.

- **Reference point return in synchronous control:**

  When reference point return of synchronous main control axis in the synchronous control is

commanded, the main control axis executes the reference point return operation in general mode, the slave control axis is synchronous with movement of the main control axis and does not execute reference point return.

But, when the main control axis in parking commands G28 automatic reference point return, the main control axis' movement is calculated from the slave control axis to main control axis. After reference point return, the main control axis and slave control axis do not set a coordinate system. In this case, the slave control axis must establish a reference point. An alarm occurs when a reference point of slave control axis is not established.

A main control axis has many slave control axes, a slave control which axis number is the smallest moves to the reference point. Besides, relative to the same main control axis, interchannel synchronization and synchronization in channels are simultaneously executed, the axis with the smallest axis number of synchronous slave control axes moves to the reference point.

When the 2^{nd} (3^{rd}, 4^{th}) reference point return of G30 in synchronous control is executed, it also executes the same operations with those of G28. Namely, only main control axis moves to the 2^{nd} (3^{rd}, 4^{th}) reference point. When the main control axis is in parking state, the slave control axis with the smallest number moves to the 2^{nd} (3^{rd}, 4^{th}) of the slave control axis.

- **Setting of a workkpiece coordinate system:**

When the synchronous control is started, a workpiece coordinate system used in the synchronous control can be automatically set. Besides, when the synchronous control ends, the system automatically returns to the workpiece coordinate system used in non synchronous control.

The workpiece coordinate system used in the synchronous control is the following coordinate system. For example, when using the synchronous control makes the axis which is different from the previous command axis move, sometime, the main control axis is set to parking state, which makes the slave control axis move. At the moment, as the workpiece coordinate system of main control axis, it is convenient to use the coordinate system meaning the current position of the slave control axis. The workpiece coordinate system is different from the main control axis' previous workpiece coordinate system, so, programs should set the coordinate system when the synchronous control is started. But, the workpiece coordinate system's automatic setting function in the synchronous control can automatically set the coordinate system. Similarly, when the synchronous control ends, the system automatically returns to the previous workpiece coordinate system of main control axis. In addition, the synchronous control ends, whether the slave control axis' workpiece coordinate system is automatically set by parameters.

Besides general synchronous control setting, the following parameters should be set at side of the slave control axis when automatic setting is performed.

(1) automatically set a workpiece coordinate system of main control axis when the synchronous control is started:

Set to "1" in SPM (No. 205#3).

Set coordinate values in No.211 when the main control axis is on the reference point and the slave control axis' reference point is on the main control axis' coordinate system.

(2) automatically set the main control axis' workpiece coordinate system when the synchronous control ends:

Set to "1" in SPS (No. 205#4) .

Set the main control axis' workpiece coordinate values in No. 047 when the main control axis is on the reference point.

(3) automatically set the slave control axis' workpice coordinate system when the

synchronous control ends:

Set to “1” in SPS (No. 205#7).

Set the main control axis’ workpiece coordinate values in No. 047 when the main control axis is on the reference point.

The command methods is the same that of generally synchronous control. When the synchronous control is started, the system automatically sets the main control axis’ workpiece coordinate system. Similarly, when the synchronous control is released, the system automatically sets a workpiece coordinate system of the main control axis or slave control axis.

Calculation method of workpiece coordinates:

(1) main control axis’ workpiece coordinate system when the synchronous control is started

Workpiece coordinate values of main control axis = main control axis’ parameter(№211)

±（slave control axis’ machine coordinate values） …①

＋（main control axis’ machine coordinate values） …②

① main control axis’ parameter SCD (№205#5) ="0": the sign is ＋

="1": the sign is－

② main control axis’ parameter SCM (№205#6)="1": summation.

(2) Release the synchronous control’s workpiece coordinate system

Main control axis’ workpiece coordinate values=main control axis’ parameter (No.047) + main control axis’ machine coordinate values

Slave control axis’ workpiece coordinate values=slave control axis’ parameter (No.047) + slave control axis’ machine coordinate values

Note 1: when many slave control axes be relative to one main control axis are synchronous, setting the main control axis’ workpice coordinate system is set to a workpiece coordinate system corresponded to the slave control axis which first starts to be synchronous.

Note 2: the above coordinate calculation is defined that the system has no tool offset, coordinate system offset. Compensation value or offset value of each channel is added to the calculation result when the system has tool offset or coordinate system offset.

- **Notes**

①、The control axes must be in stop state when the synchronous control is started or released.

②、The system cannot send movement commands to the synchronous slave control axis in the synchronous control.

③、Least command unit, diameter/radius type, acceleration/deceleration time constant of main control axis should be consistent as possible with those of the slave control axis.

④、A wait M code is added before starting/releasing the synchronous command to pause the program read, which ensures the synchronous control axis data is normally processed after starting/releasing the synchronous control.

⑤、Confirm the corresponding control axis has completed the reference point return after power on when automatic setting function of the synchronous control coordinate system is used and the synchronous control is started.

⑥、The system automatically cancels the synchronous control when emergency stop; whether the synchronous control is cancelled is determined by parameter when reset；

⑦、When an alarm occurs in the course of synchronous control, the channel of synchronous control axis is in feed hold state, without the channel of synchronous control axis in feed hold being set by parameter (No. 020#1).

- **Signal explanation**

Synchronous control axis selection signal SYNC1~SYNC5 <Gn138.0~ Gn138.4>

［Classification］input signal

［Function］ perform the synchronous control.

［Operation］ when the signal becomes ‘1’ from ‘0’, the corresponding axis as a slave control axis starts the synchronous control. When it becomes ‘0’ from ‘1’, the synchronous control is released.

Parking signal PK1~PK5 <Gn122.0~Gn122.4>

［Classification］input signal

［Function］ place each axis in parking state and make each axis not to move.

［Operation］ when the signal is ‘1’ in synchronous control, the corresponding axis is set to parking state. The signal is invalid in non synchronous control.

Synchronous main control axis confirmation signal SYCM1~SYCM5 <Fn341.0~Fn341.4>

［Classification］output signal

［Function］ the signal informs each axis is the synchronous main control axis.

［Output］ it is ‘1’ when the corresponding axis is the synchronous main control axis, and it is ‘0’ when the corresponding axis is released by synchronous slave control.

Synchronous slave control axis confirmation signal SYCS1~SYCS5 <Fn342.0~Fn342.4>

［Classification］output signal

［Function］ the signal informs each axis is the synchronous slave axis.

［Output］ it is ‘1’ when the corresponding axis is the synchronous slave control axis, and it is ‘0’ when the corresponding axis is released by synchronous slave control.

Parking axis confirmation signal SMPK1~SMPK5 <Fn346.0~Fn346.4>

［Classification］output signal

［Function］ the signal informs each axis is the parking axis of synchronous control.

［Output］ it is ‘1’ when the corresponding axis is a parking axis of synchronous control, and it is ‘0’ when the corresponding axis is released by synchronous slave control or parking is released.

Synchronous/mix/overlap controlling signal SYN1O~SYN5O <Fn118.0~Fn118.4>

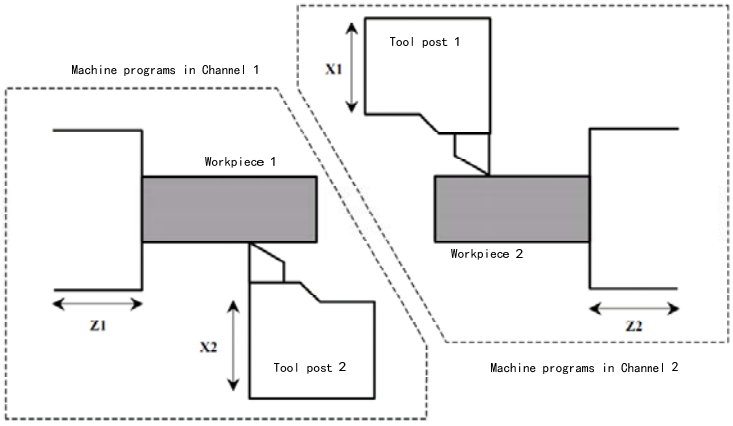
［Classification］output signal

［Function］ the signal informs each axis is in synchronous/mix/overlap control.

［Output］ it is ‘1’ when the corresponding axis is in synchronous/mix/overlap control and it is ‘0’ when it is not in synchronous/mix/overlap control.

4.2.8 Mix control

Mix control function can exchange interchannel any axes to realize interchannel axis' movement control, which is shown below:



Exchange commands of X1 and X2.

Programs through path 1 command X2 and Z1 to move.

Programs through path 2 command X1 and Z2 to move.

- **Mix control setting:**

  Set axis numbers of each exchange axis. Setting method is: path number of exchange axis*100+relative axis number in channel.

  Example: when the axis' allocation is X (1st axis), Z (2nd axis):

  When Z2 and Z1 interchange, №212_{Z2} = 102

  When X1 and X2 interchange, №212_{X1} = 201

  Note: mix control is performed only in channel.

- **Mix control signals:**

Mix control start/release is performed at side of PMC, which is realized by setting mix control axis selection signals MIX1～MIX5. the mix control is started when MIX1～MIX5 become '1' from '0', it is released when these signals become '0' from '1'.

Note 1: starting/releasing mix is to check edge signal of MIXn, i.e., check rising edge of MIXn when starting mix, and check falling edge of MIXn when releasing mix.

Note 2: mix control is to interchannel exchange of two axes. So, when mix control is started/mixed, MIXn signal can select one of two axes, but the reads the corresponding parameter setting value according to selected MIXn signal when reading exchange axis number.

- **Mix control commands:**

Command the waited M code before/after starting/releasing mix control M code.

Channel 1	Channel 2	
:	:	
M100;	M100;	Wait
	M82;	Start mix control
M101;	M101;	Wait
:	:	
Independent run	Independent run	
:	:	
M100;	M100;	Wait
	M83;	Mix control released
M101;	M101;	Wait
:	:	

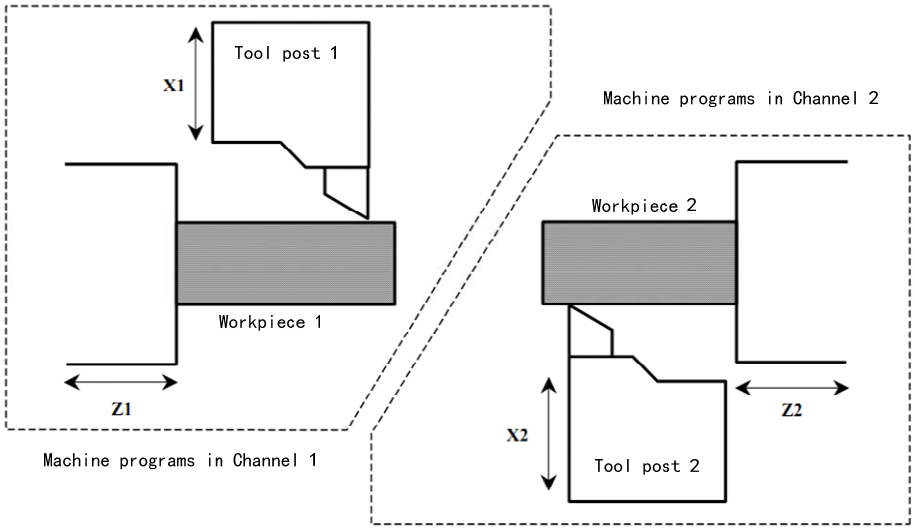
Note 1: mix control starting/releasing M82/M83 is used to control change of mix control axis selection signal MIXn. (here, M82/M83 is a sample command, and the actual function command is customized in PLC according to machine allocation).

Note 2: the wait M code is added before the command for starting/releasing mix control to pause program read, which can ensure the mix control axis data is normally processed after starting/releasing mix control.

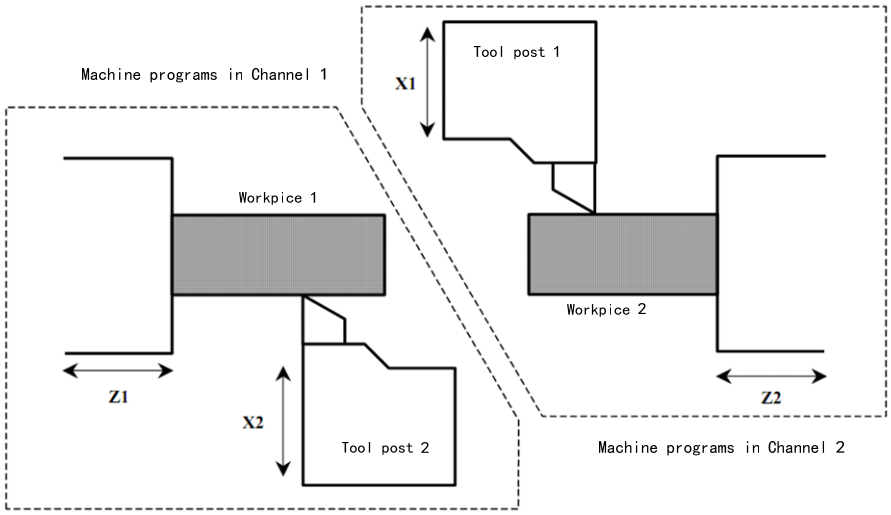
- **Example:**

1. Exchange X1 and X2 to machine on X1, Z1 in Channel 1 and on X2 and Z2 in Channel 2.

1)Independent control:



2)Mix control:



Programs in Channel 1 command X2 and Z1 to move when mix control. X's workpiece coordinate in Channel 1 means to position of tool post 2. Similarly, programs in Channel 2 command X1 and Z2 to move, and X's workpiece coordinate in Channel 2 means position of tool post 1.

- **Reference point return in mix control:**

When G28/G30 automatic reference point return is commanded in mix control, the movement amount is calculated by axes in other channels moving to reference point. In this case, axes in other channels must have established reference points, otherwise, an alarm occurs. Besides, manual reference point return cannot be executed.

- **Workpiece coordinate system's setting:**

A workpiece coordinate system used in mix control can be automatically set when the mix control is started. Besides, when it is ended, the system automatically returns to the workpiece coordinate system used in non mix control.

The workpiece coordinate system used in the mix control is the following coordinate system. For example, when using the mix control make the axis different from the previous command axis move, using the workpiece coordinate system being command axis means it is convenient to current position's coordinate system of movement axis. The workpiece coordinates are different from the previous workpiece coordinates of command axis, so, when the mix control is started, the coordinate system is set by program. When the mix control is ended, the system should return to previous workpiece coordinate system of command axis. Using the function, these coordinates can be automatically set.

Besides general mix control setting, the following parameters should be set when automatic setting is performed.

1. it is set to "0" in MCS(№206#6):

(1) the workpiece coordinate system is automatically set when the mix control is started:

it is set to "0" in MPM(№206#3).

Set coordinate values of the mix control exchange axis' reference point in mix control coordinate system in No. 213.

(2) the workpiece coordinate system is automatically set when the mix control is ended:

It is set "1" in MPS (№206#4).

Set coordinate values on the previous workpiece coordinate system in No.047 when the mix control axis is on the reference point.

The command method is the same that of general mix control. When the mix control is started, the mix axis' workpiece coordinate system is automatically set. Similarly, the system automatically sets the mix axis' coordinate values on the previous workpiece system when the mix control is released.

Calculation method of workpiece coordinates: (take example of exchanging X1 and X2)

(1) workpiece coordinate system when the mix control is started

X1 = (№0213 $_{X1}$setting value)±(X2 machine coordinate value)

X1's MCD $_{X1}$ (№206#5) = 0, it is +,=1: it is -

X2 = (№0213 $_{X2}$ setting value)±(X1's machine coordinate value)

X2's MCD $_{X2}$ (№206#5) = 0: it is +,=1: it is -

(2) workpiece coordinate system when the mix control is released

X1 = (№047 $_{X1}$ setting value)+(X1's machine coordinate value)

X2 = (№047 $_{X2}$ setting value)+(X2's machine coordinate value）

2. it is set to "1" in MCS(№206#6)：

X1 = X2's coordinate value

X2 = X1's coordinate value

Note: the above coordinate calculation is for the system without tool offset, coordinate system offset. Compensation value or offset value of each channel is added to the calculation result when the system is with offset or coordinate system offset.

Notes

①、Control axes must be in stop state when the mix control is started/released.

②、Least command unit, diameter/radius type, acceleration/deceleration time constant, axis type must be consistent.

③、A wait M code is added before starting/releasing the mix command to pause the program read, which ensures the mix control axis data is normally processed after starting/releasing the mix control.

④、Confirm the corresponding control axis has completed the reference point return after power on when automatic setting function of the mix control coordinate system is used and the mix control is started.

⑤、The system automatically cancels the mix control when emergency stop; whether the mix control is cancelled is determined by parameter when reset.

⑥、Axes in the mix control cannot use G31, G36, G37 and other commands with axis stop function.

- **Signal explanation**

Mix control axis selection signal MIX1~MIX5 <Gn128.0~ Gn128.4>

［Classification］input signal

［Function］ perform mix control.

［Operation］ the corresponding axis' mix control is started when the signal becomes '1' from '0', the corresponding axis' mix control is released when the signal becomes '0' from '1'.

Mix axis confirmation signal MIXO1~MIXO5 <Fn343.0~Fn343.4>

［Classification］output signal

［Function］ the signal informs each axis is a mix control axis.

［Output］ it is '1' when the corresponding axis is the mix control axis, it is '0' when the corresponding axis is released by the mix control .

Synchronous/mix/overlap controlling signal SYN1O~SYN5O <Fn118.0~Fn118.4>

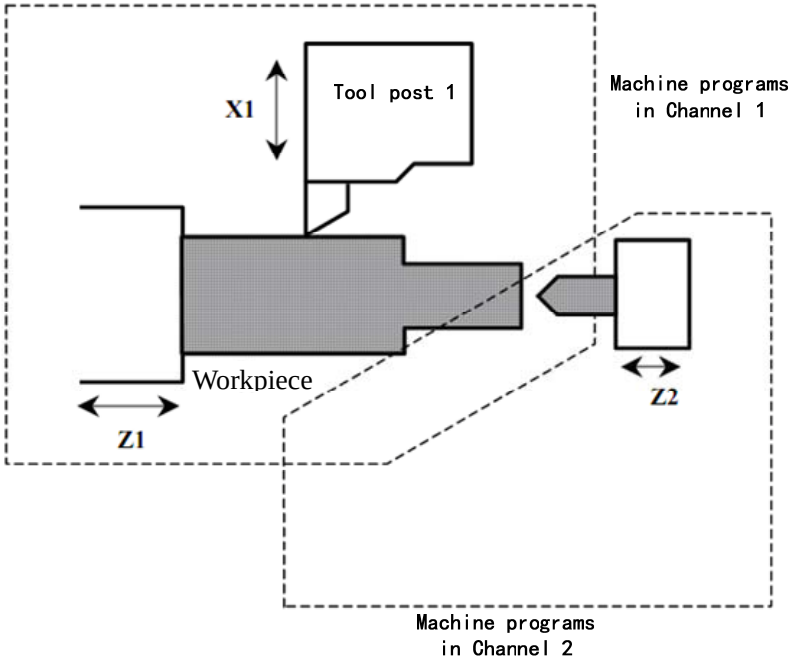
［Classification］output signal

［Function］ the signal informs each axis is in the synchronous/mix/overlap control.

［Output］ it is '1' when the axis is in the synchronous/mix/overlap control；
it is '0' when the axis is in not the synchronous/mix/overlap control.

4.2.9 Overlap control

The overlap control is that an axis (overlap slave control axis) executing movement command in movement amount adds other axis(overlap main control axis)' movement amount. Similar to other synchronous control, the overlap control can send movement command to the salve control axis. Motion of slave control axis consists of its own command motion and main control axis motion, which is shown below:



Movement amount of Z1（main control）axis overlap Z2 (slave control)axis

- **Overlap control setting:**

Set the main control axis' axis number of each axis in No. 215 in overlap control. Its setting method: main control axis path number * 100+relative axis number in its path.

Example: when axis allocation is X (the 1st axis), Z (the 2nd axis):

Z2（slave control）axis overlaps Z1（main control）axis, №215_{Z2} = 102

X1（slave control）axis is synchronous with X2（main control）axis, №215_{X1} = 201

- **Overlap control signals：**

Overlap control start/release is executed at side of PMC by setting the slave control axis's overlap control selection signals OVLS1～OVLS5. When OVLS1～OVLS5 becomes '1' from '0', the overlap control is started; the overlap control is released when it becomes '0' from '1'

Note: the overlap control start/release is to check edge signal of OVLSn signal, i.e. OVLSn rising edge is checked when the overlap control is started,and OVLSn falling edge is checked when the overlap control is released.

- **Overlap control command:**

Command the wait M code before/after starting/releasing the overlap control.

Main control axis' path Slave control axis' path

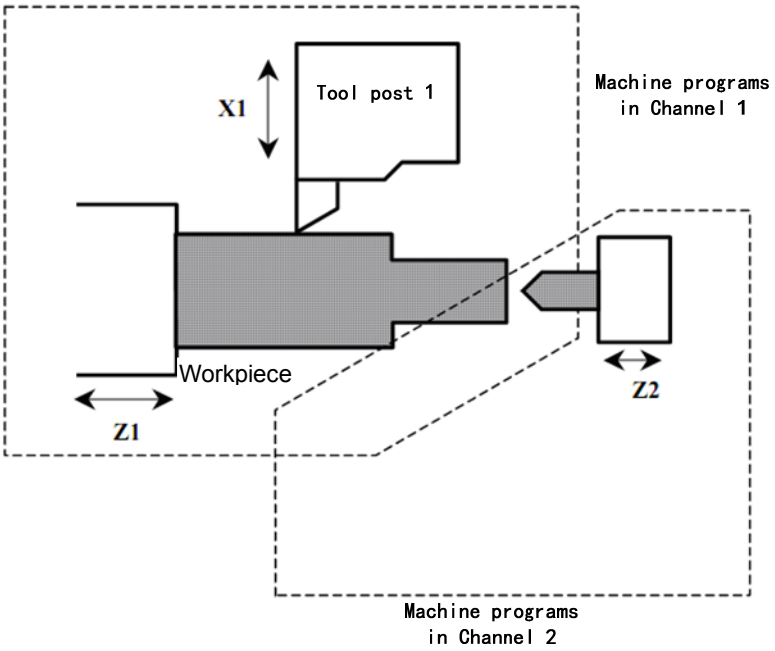
:	:	
M100;	M100;	wait
	M84;	start the overlap control
M101;	M101;	wait
:	:	
Separate run	Separate run	
:	:	
M100;	M100;	wait
	M85;	release the overlap control
M101;	M101;	wait
:	:	

Note 1: The overlap control start/release command M84/M85 is used to control change of overlap control selection signal OVLSn. (Here, M84/M85 is a sample command and the actual function command is required to define in PLC according to machine allocation).

Note 2: The wait M code before starting/releasing the overlap command is added to pause program read, which can ensure overlap control axis data is normally processed after starting/releasing the overlap control.

- **Example：**

◆ Make Z1 (main control)'s movement amount overlaps Z2(slave control)



- **Difference with general synchronous control**

①、Parking signal is invalid to axes in the overlap control.

②、The slave control axis in the overlap control cannot return to reference point.

- **Reference point return in overlap control：**

When reference point return of overlap main control axis in the overlap control is commanded, the main control axis executes the reference point return operation in general mode, the overlap slave control axis is synchronous with movement of the main control axis and does not execute the reference point return of overlap slave control.

- **Notes**

①、The control axes must be in stop state when the overlap control is started or released.

②、Least command unit, diameter/radius type, acceleration/deceleration time constant of main

control axis should be consistent as possible with those of the slave control axis.

③、Its own movement amount of the slave control axis in the overlap control addes the main control axis' movement amount, which causes danger because of actual movement speed of the slave control axis be more than the specified speed.

④、A wait M code is added before starting/releasing the overlap command to pause the program read, which ensures the overlap control axis data is normally processed after starting/releasing the overlap.

⑤、When the overlap control is executed, workpiece coordinates of slave control axis does not change along the main control axis, but the machine coordinates do.

⑥、Confirm the corresponding control axis has completed the reference point return after power on when the overlap control is started.

⑦、The system automatically cancels the overlap control when emergency stop; whether the overlap control is cancelled is determined by parameter when reset.

⑧、When an alarm occurs in the course of overlap control, the channel of overlap control axis is in feed hold state, without the channel of overlap control axis in feed hold being set by parameter (No. 020#1).

- **Signal explanation**

Overlap control axis selection signal OVLS1~OVLS5 <Gn190.0~ Gn190.4>

［Classification］input signal

［Function］ perform the overlap control.

［Operation］ when the signal becomes '1' from '0', the corresponding axis as a slave control axis starts the overlap control. When it becomes '0' from '1', the overlap control is released.

Overlap main control axis confirmation signal OVMO1~OVMO5 <Fn344.0~Fn344.4>

［Classification］output signal

［Function］ the signal informs each axis is the overlap main control axis.

［Output］ it is '1' when the corresponding axis is the overlap main control axis, and it is '0' when the corresponding axis is released by overlap slave control.

Overlap slave control axis confirmation signal OVSO1~OVSO5 <Fn345.0~Fn345.4>

［Classification］output signal

［Function］ the signal informs each axis is the overlap slave axis.

［Output］ it is '1' when the corresponding axis is the overlap slave control axis, and it is '0' when the corresponding axis is released by overlap slave control.

Synchronous/mix/overlap controlling signal SYN1O~SYN5O <Fn118.0~Fn118.4>

［Classification］output signal

［Function］ the signal informs each axis is in synchronous/mix/overlap control.

［Output］ it is '1' when the corresponding axis is in synchronous/mix/overlap control and it is '0' when it is not in synchronous/mix/overlap control.

4.2.10 Programs commanding synchronous/mix/overlap control

Starting/releasing the synchronous/mix/overlap is commanded by programs instead of control axis selection signals. Start/release process is the same that of using control axis selection signals.

- **Synchronous control:**

Command format:

G51.4 P_ Q_ L_ start the synchronous control（L can be omitted）

G50.4 Q_ release the synchronous control

Command explanation：

P：synchronous main control axis' identification number（range：0～999）

Q：synchronous slave control axis' identification number（range：0～999）

L：parking state（range：0～2）

0：do not parking. Parking is released

1：main control is parked and slave control axis' parking is released

2：slave control is parked and main control axis' parking is released

Note: L can be omitted, and its default is 0 when it is omitted.

- **Mix control：**

Command format:

G51.5 P_ Q_ start the mix control

G50.5 P_ Q_ release the mix control

Command explanation：

P：mix axis 1' identification number（range：0～999）

Q：mix axis 2' identification number（range：0～999）

- **Overlap control：**

Command format：

G51.6 P_ Q_ start the overlap control

G50.6 Q_ release the overlap control

Command explanation：

P：overlap main control axis' identification number（range：0～999）

Q：overlap slave control axis' identification number（range：0～999）

- **Example：**

Identification number of X, Z in Channel 1 is separately set to 11, 12; that of X, Z in Channel 2 is separately set to 21, 22 .

Programs in Channel 1	Programs in Channel 2	Note
……	……	
M100	M100	Interchannel waits
G51.4 P12 Q22 L0		Z in Channel 2 is synchronous with Z in Channel 1,but does not park
M101	M101	
……	……	
M102	M102	
G50.4 Q22		Release the synchronous control of Z being a slave control axis
M103	M103	

- **Explanations：**

1、 P, Q identification number in synchronous/mix/overlap control is set in No. 209

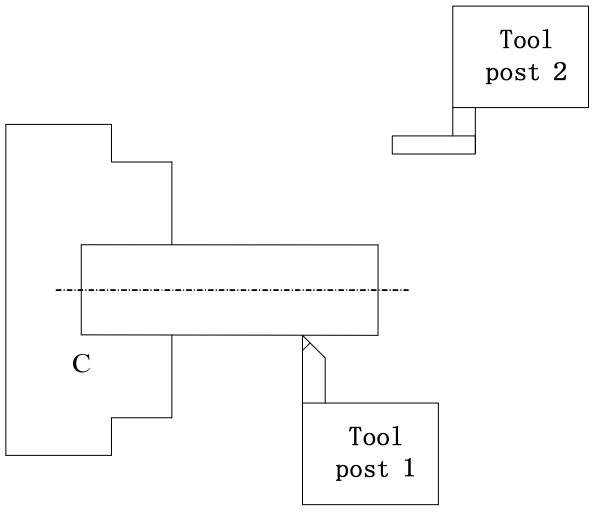
2、 G50.4/G51.4/G50.5/G51.5/G50.6/G51.6 is non modal command in Group 00.

3、 A control axis selection signal or a program command can start/release synchronous/mix/ overlap control. Namely, after a control axis selection signal starts the synchronous/mix/ overlap control, a program command can release the control; or after a program command start the control, falling edge of the control axis selection signal can also release the control.

4、 Add the wait M code before starting/releasing the synchronous/mix/overlap command to pause program read, which ensures control axis data can be normally executed after starting/releasing the synchronous/mix/overlap control.

4.2.11 Imaginary axis control

Using the function can set some axis' imaginary axis number belonging to a channel in another channel, and then using the mix control can exchange its control authority to another channel, which makes some special axis realize interpolation in two channel. For example, Channel 1 distributes X, Z, C axis, among which C is CS axis, and Channel 2 distributes X, Z axis. Generally, C axis' CS contour control is performed only in Channel 1, which makes C control authority be transferred to Channel 2, ensuring Channel 2 also performs C axis' CS contour control as follows:



- **Function setting**

Set imaginary axis' axis number of each axis in No. 214, and its setting method is : imaginary axis channel number *100+ relative axis number in channel. The axis' imaginary axis function is disabled when it is set to 0.

Example: when C axis (the 5th axis) in Channel 1 is set its imaginary axis to the 5th axis of Channel 2:

№214_{C1} = 205

Note 1: The function is mainly used to make the axis' control authority exchange in channels, so, the imaginary axis' channel cannot be in the same channel with the axis.

Note 2: All kinds of attributes of axes are not changed after their control authorities are exchanged, so, these axes' attributes cannot conflict with other axes in channel after exchange.

Note 3: The setting value is taken as the imaginary axis' identification number in command mix control, at the moment, other axes' identification setting cannot be repeated with it.

- **Control authority exchange**

After the imaginary axis is set, the mix control between the axis and the imaginary axis can realize the control authority exchange in channels. The mix control methods is referred to relevant

explanations about the mix control function.

- **Notes**

1. When the imaginary axis is CS, its speed and position control cannot be exchanged after the control authorities are exchanged.
2. The function is mainly used to make the axis' control authority exchange in channels, so, the imaginary axis' channel cannot be in the same channel with the axis.
3. All kinds of attributes of axes are not changed after their control authorities are exchanged, so, these axes' attributes cannot conflict with other axes in channel after exchange.

4.2.12 Interchannel single block check function

In dual channel control, each channel separately has the single block stop function, so, single block stop function makes each automatic running program stop. But, even if single block stop in each channel is enabled, their stop opportunities are different because of different program commands.

Example: two channels start single block stop functions.

Channel 1	Channel 2
O0001	O0002
N10 G01 U10 F100	N10 G01 U20 F100
N20 U-10	N20 U-20
……	……
M30	M30

Channel 1 and Channel 2 simultaneously start run, Channel 2 still runs after Channel 1 completely runs N10 single block. Their stop opportunities are different although the two channels start single block stop function.

No.020 Bit6 is set to 1. After interchannel single block check function is started and when the single block stops in some channel, another channel starts pause state, which makes the two channel almost stop, ensuring the two channels simultaneously single block stop.

Note 1: When the pause channel is executing thread cutting, tapping and so on, it does not immediately stop even if it is in pause state.

Note 2: Using the function makes the two channels' stop opportunities not be completely different.