This user manual describes all items concerning the operation of the system in detail as much as possible. However, it is impractical to give particular descriptions of all unnecessary and/or unavailable operations of the system due to the manual content limit, product specific operations and other causes. Therefore, the operations not specified herein should be considered impossible or unallowable.

The User Manual is applied to the following CNC systems:

Type	structure	LCD size	Remark
GSK980TA3	horizontal	7.0	LCD size is defaulted to be 7.0 inch

GSK980TA3 is upgrading products of GSK980TA2, which can control three feed shaft, a spindle , and add the function of automatic feeding, groove cycle processing(G71) and so on, which can significantly improve the efficiency of machining parts, precision and surface roughness.

As GSK980TA2 upgrades, GSK980TA3 is the best choice of the economical cnc lathe technology upgrading.

FOREWORD

Dear user,

We are really grateful for your patronage and purchase of this turning CNC system of GSK980TA3 series made by GSK CNC Equipment Co., Ltd.

The user manual describes the programming, operation, installation and connection. Please read it carefully before operation in order to get the safe and effective working.

WARNING

This system can only be operated by authorized and qualified personnel as improper operations may cause accidents.

Please carefully read this user manual before use!

Note: The power supply installed on (in) the cabinet is exclusive to GSK'S CNC systems.

The power supply form is forbidden to be used for other purposes. Otherwise, there may be extreme danger!

SAFETY PRECAUTION

Warning, caution and note

This manual includes safety precaution for protecting the user and preventing damage to the machine. Precautions are clssified into Warning and Caution according totheir bearing on safety. Read the Warning, Caution and Note thoroughly before attempting to use the machine.

Warning

There is a danger of the user being injured or the equipment being damaged if the approved procedure is not observed.

Caution

There is a danger of the equipment being damaged if the approved procedure is not observed.

Note

It is used to indicate supplementary information other than Warning and Caution.

■ Delivery and storage

- Packing box over 6 layers in pile is unallowed.
- Never climb the packing box, neither stand on it, nor place heavy objects on it.
- Do not move or drag the product by the cables connected with it.
- Forbid collision or scratch to the panel and displayer.
- Packing box should be protected from damping, insolation and raining.

■ Open packing box to check

- Ensure things in packing box are the required ones.
- Ensure the product is not damaged in delivery.
- Ensure the parts in packing box are in accordance to the order.
- Contact us in time if the product type is inconsistent with the order, there is short of accessories, or product damage in delivery.

■ Connection

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

■ Troubleshooting

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

SAFETY RESPONSIBILITY

Manufacturer's safety responsibility

——The manufacturer should be responsible for the cleared or the controlled safety in the design and the structure of the CNC system and the accessories.

——The manufacturer should be responsible for the CNC system and the accessories.

——The manufacturer should be responsible for the message and the suggestion for the user.

User's safety responsibility

——The user should study and train the system safety operation, master the safety operation content.

——The user should be responsible for the danger caused by increasing, changing or modifying the CNC system, the accessories by itself.

——The user should be responsible for the danger because of the mistaken operation, regulation, maintenance, installation and storage.

This user manual shall be kept by final user.

All specification and designs are subject to change without further notice.

Contents

PART ONE　SUMMARY

CHAPTER ONE SUMMARY

1.1 Summary

Part One Summary Introduce the system, composition of chapters, he system model, relavant use explanations, and precautions before reading the user manual.

Part Two Programming Describe composition of part programs, fundamentals of programming, each code's function, command format, characteristics and restricts in NC programming, and so on.

Part Three Operation Narrate each window and setting of the system, each operation of machine, program input/output, edit, and the system's communication, and so on.

Part Four Connection Mention the system's structure, installation dimension, connection among devices, I/O interface definition, machine debugging, the system's specification, optional memory pitch error compensation function.

Appendix List parameter tables (including parameter default value and parameter setting range), alarm tables and diagnosis table.

The user manual is applied to **GSK980TA3** CNC systems.

1.2 Product Brief

GSK980TA3 is upgrading products of GSK980TA2, which can control three feed shaft, a spindle , and add the function of automatic feeding, groove cycle processing(G71) and so on, which can significantly improve the efficiency of machining parts, precision and surface roughness. As GSK980TA2 upgrades, GSK980TA3 is the best choice of the economical cnc lathe technology upgrading

As upgraded productes, the systems have the following features:

- 32-bit CUP, CPLD hardward interpolation technology to realize high-speed μm level control;
- 4-layer circuit board with high integrated level, proper system technology structure and high reliability;
- Chinese/English LCD，friendly interface and easy operation;
- Adjustable acceleration/deceleration to be matched with stepper drive unit or servo drive unit
- Changeable electronic gear ratio;

- Prepositioning USB interfaces and RS232 interfaces to be convenient to the user managing programs.

1.3 Model and Meaning

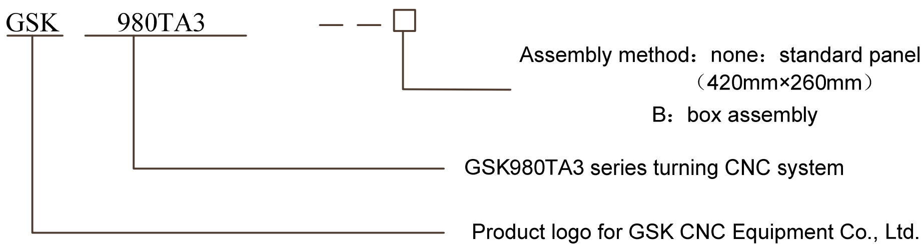


Fig. 1-3

1.4 Order

GSK980TA3 can select Y as the additional axis and the user must remark it. Refer to the supplementary about Y axis explanation.

Table 1-4

Model	Explanation
GSK980TA3	420mm×260mm aluminium operation panel
GSK980TA3-B	GSK980TA3 Be matched with AP01 to operation box（445mm×345mm×182mm）

PART TWO PROGRAMMING

CHAPTER ONE　PROGRAMMING FUNDATION

1.1　Coordinate axis definition

It is important to stipulate the coordinate axis name and movement direction of CNC machine. Designers, operators and maintenance personnel of CNC machine should correctly understand it, otherwise, it causes the mistaken programming and data communication, operation accidence, abnormal maintenance, and so on.

Fig.1-1-1 is a axis sketch map of CNC turning machine.

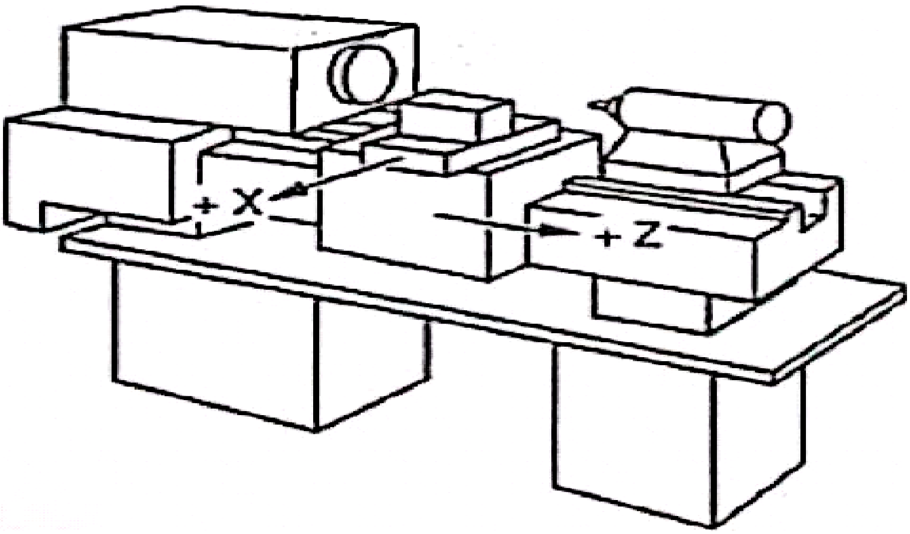


Fig.1-1-1

The system uses a rectangular coordinate system composed of X, Z axis to execute the positioning and the interpolation movement. X axis is in the direction of front and back in the plane, and Z axis is of left and right. The negative direction of them approach to the workpiece and positive one is away from it, which are shown in Fig.1-1-1.

The system supports a front tool post, a rear tool post function, and describes that the tool post before the workpiece is called as a front tool post and it behind the workpiece is called as a rear tool post. Fig. 1-1-2 is a coordinate system of the front tool post and Fig. 1—1-3 is a rear toolpost one. It shows exactly the opposite of X axis, but the same of Z axis from figures. In the manual, it will introduce programming application with the front tool post coordinate system in the following figures and examples.

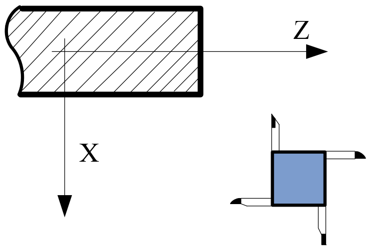


Fig.1-1-2 Front tool post coordinate system

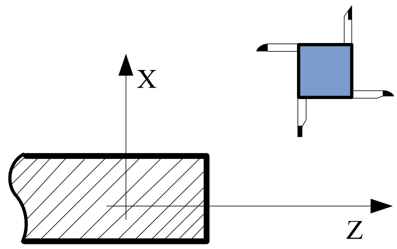


Fig.1-1-3 Rear tool post coordinate system

1.2 Machine coordinate system, Machine Zero

Machine tool coordinate system is a fixed one; its origin is taken as the machine zero installed on the max. travel in positive X, Z axis. The machine zero is defined after the machine is designed, manufactured and adjusted, and it is a fixed point. The machine zero is not defined when the CNC is turned on, and generally, the automatic or manual machine zero return is executed to create the machine coordinate system. CNC has created the machine coordinate system after the machine zero return is completed.

Note: Do not execute the machine zero function(such as G28) without the machine zero switch installed on the machine tool.

1.3 Workpiece Coordinate System & Reference Point (Program Zero)

A workpiece coordinate system (also called floating coordinate system) is used when programmers in programming, programmer selects a known point on the workpiece as reference point (also called program zero) to establish a new coordinate system, which is called a workpiece coordinate system. Once the workpiece coordinate system has been established, it is valid until it is replaced by a new one. When the system is turned off or power down, the program zero position is not saved. Using G50 for the system creates a workpiece coordinate system. When there is no G50 in programs, the current absolute coordinate value is taken the reference point to create a workpiece coordinate system. The reference point selection of the workpiece coordinate system should meet the simple programming, few dimension conversions and machining error. Generally, the reference point should be on the reference marked by the dimension or positioning reference. For turning machine programming, the reference point should be on the intersection point between the workpiece's axis and the end face of the chuck (Fig.1-3-1) or the workpiece's end face (Fig.1-3-2).

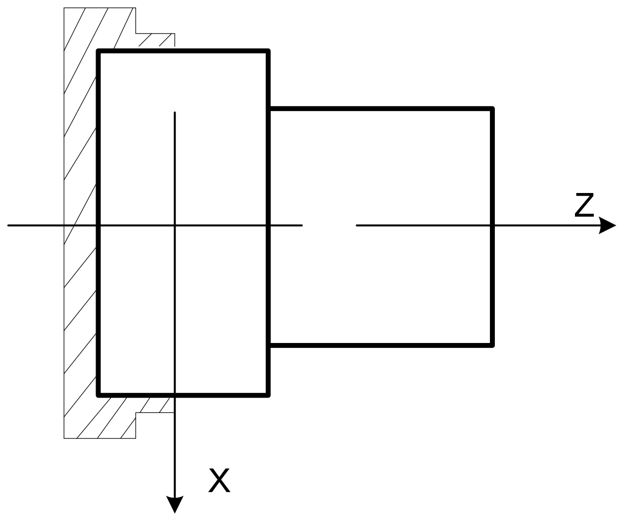


Fig.1-3-1　reference point on end face of chuck

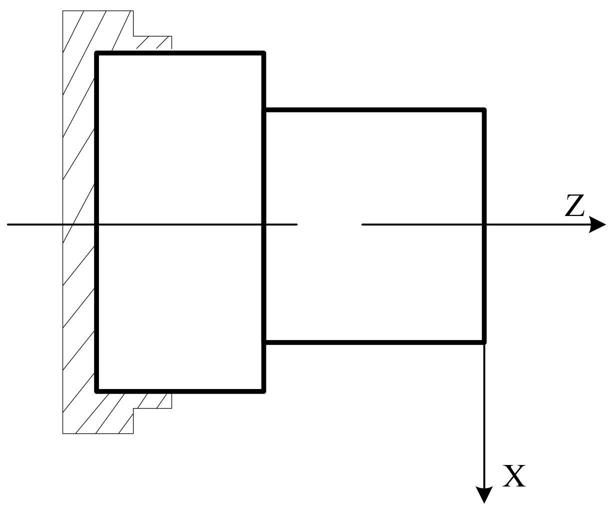


Fig. 1-3-2　reference point on workpiece's end face

1.4　Controlled axes

Table 1-4-1

GSK980TA3	Controlled axes	3 axes X，Z，Y）
	Simultaneously controlled axes	3 axes（X，Z，Y）

1.5 Input Increment

Table 1-5-1

Input/output	Least input increment	Least command increment
Metric input /metric output	X：0.001 mm（Diameter） Z：0.001 mm	X：0.0005 mm Z：0.001 mm
	X：0.001 mm （Radius） Z：0.001 mm	X：0.001 mm Z：0.001 mm

Diameter/radius designation is set by NO:1#2 and the parameter is valid only to X.

The input increment is referred to the machine manufacture's User Manual.

1.6 Maximum Stroke

Maximum stroke = least input increment×9999999

1.7 Absolute Programming & Incremental Programming

The movement of the command axis is divided into absolute command and incremental command. The absolute command is to use the end point of axis movement to execute programming, which is called absolute programming. The incremental command is to use the axis movement to directly execute programming, which is called incremental programming. For the system, the absolute programming uses X, Z and the incremental programming uses U, W.

Table 1-7-1

Absolute command	**Incremental command**	**Remark**
X	U	X movement command
Z	W	Z movement command

Example: Using an absolute coordinates, incremental coordinates and compound coordinates compile A→B program described in Fig.1-7-1.

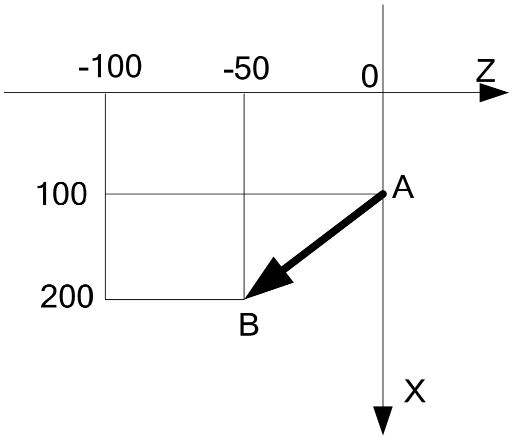


Fig. 1-7-1

Table 1-7-2

Programming mode	**Absolute programming**	**Incremental programming**	**Compound programming**
Programming	G1 X200 Z-50	G1 U100 W-50	G1 X200 W-50
Remark	Suppose that current coordinate point of the tool is on A, the linear interpolation is executed.		

Note: When there are command address X/ U or Z/ W at the same time, #132 alarm occurs: X, U or Z, W exist simultaneously.

Example: G50 X10 Z20;
G01 X20 W30 U20 Z30;

1.8 Diameter and Radius Programming

The appearance of the machined workpiece is a rotating body, among which X dimension can be specified by: the diameter and the radius, which is set by the bit of **NO:1#2**.

When **NO:1#2** is set to 1, the radius is specified to execute programming.

When **NO:1#2** is set to 0, the diameter is specified to execute programming.

Table 1-8-1 diameter, radius designation

Item	**Diameter designation**	**Radius designation**
Z command	Not related to diameter, radius designation	
X command	Diameter designation	Radius designation
Incremental command of	Diameter designation	Radius designation

Item	Diameter designation	Radius designation
address U		
Coordinate system setting（G50）	Diameter designation	Radius designation
X value of too offset	NO:2#5 specifies the diameter designation or radius designation	
Radius command of circular interpolation（R，I，K）	Radius designation	Radius designation
X feedrate	Radius change (mm/min, mm/r)	
X position display	Display diameter value	Display radius value

Note 1: The diameter designation is used except for special explanation in the User Manual.

Note 2: The tool offset using diameter/radius is defined that the outside diameter of workpiece uses diameter or radius when the tool offset is changed.

Example: when the diameter is specified, and the compensation value changes 10mm, the diameter value of the workpiece's outside diameter changes 10mm; when the radius is specified, and the compensation value changes 10mm, the diameter value of the workpiece's outside diameter changes 20mm.

1.9 Modal, Simple and Initial State

The modal is defined that after the function and state of the corresponding word are executed, they are valid till they are done again, namely, and the same functions and states are used in the following blocks are, the word need not be input again.

Example:

G0 X100 Z100；（rapid position to X100 Z100）

X120 Z30；（rapid position to X120 Z30，G0 is modal and can be omitted）

G1 X50 Z50 F300；（linear interpolation to X50 Z50，feedrate 300mm/min G0→G1，）

X100；（linear interpolation to X100 Z50，feedrate 300 mm/min, G1Z, 50, F300 are modal and can be omitted）

G0 X0 Z0；（rapid position to X0 Z0）

The simple is defined that after the function and state of the corresponding word are executed, they are valid one time, namely, and the same functions and states are used in the following blocks are, the word needs be input again.

The initial state is defined to the default function and state after the system is turned, namely, the system executes the initial function and state when the system is turned but does not define the corresponding function and state. The initial state of the system includes G00, G40, G97, G98, M05, M09, M33.

Example:

O0001；

G0 X100 Z100；　　（rapid position to X100 Z100，G0 is the system's initial state）

G1 X0 Z0 F100；　　（linear interpolation X0 Z0，feed per minute，feedrate 100 mm/min，G98 is the initial state after power on）

CHAPTER TWO　STRUCTURE OF AN PART PROGRAM

A program is defined to a series collection of commands to control the CNC machine to complete workpiece machining. After the complied programs are input to the CNC system, the CNC system controls the tool movement along the linear and arc, the spindle starting/stopping, the cooling and the lubricating ON/OFF according to the commands. The commands in the program are compiled according to the actual movement sequence of the machine.

2.1　General Structure of a Program

A program consists of a sequence of block which is composed by words. Each block is separated by the block end command (ISO is LF, EIA is CR). “;” is used in the User Manual to mean the end of block.

General structure of a program is shown in Fig. 2-1-1.

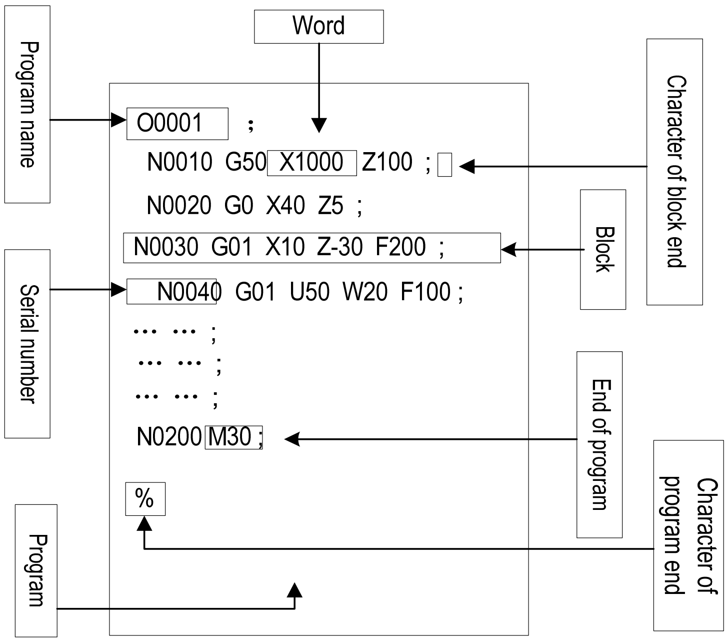


Fig. 2-1-1

2.1.1　Program Name

There are most 500 programs stored in GSK980TDa. To identify it, each program has only one program name (there is no the same program name) beginning with command address O and the following 4 digits.

O □□□□

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

→ Address O

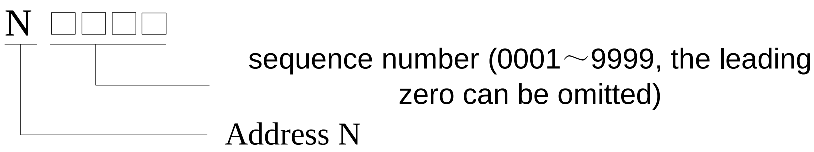
End of a program

A program starts with its program name and ends with “%”.

2.1.2 Sequence Number and Block

A program consists of many commands and one command unit is called a block (Fig.2-1-1). A block end command is used to separate blocks (Fig. 2-1-1), and “;” is the block end command in the User Manual.

The beginning of a block can use a sequence number composed by the address N and its following 4 digits.



The sequence of a sequence number is arbitrary (**NO:2 # 7** sets whether another sequence number is inserted) and its interval is not equal (the interval size is set by P50). All blocks can be with sequence numbers and some key blocks can be with them. Generally, sequence number is from the small to the big according to machining sequence.

2.1.3 Word

A word is an element of a block. A word is composed by an address and its following digit(some is with + or - before the digit), which is shown in Fig. 2-1-3:

Fig. 2-1-3-1

An address is one of English letters. The address describes the meaning of its following numerical. In the system, the useful addresses and meanings are shown in Fig. 2-1-3-1. Some address has different meanings according to different commands.

Table 2-1-3-1 Address list

Address	Value range	Function
O	0～9999	Program name
N	1～9999	Block number
G	00～99	Preparatory function
X	-9999.999～9999.999（mm）	X coordinate
	0～9999.999（s）	Pause time
Z	-9999.999～9999.999（mm）	Z coordinate
U	-9999.999～9999.999（mm）	X increment
	-9999.999～9999.999（mm）	X finishing allowance in G71,G72, G73
	0.001～9999.999 （mm）	Cutting depth in G71
	-9999.999～9999.999（mm）	Travel of X tool retraction in G73
W	-9999.999～9999.999（mm）	Z increment
	0.001～9999.999（mm）	Cutting depth in G72
	-9999.999～9999.999（mm）	Z finishing allowance in G71,G72, G73
	-9999.999～9999.999（mm）	Z tool retraction in G73
R	0～9999.999（mm）	Arc radius
	0.001～9999.999（mm）	Tool retraction in G71, G72
	1～9999999（times）	Roughing cycle times in G 73
	0～9999.999（mm）	Tool retraction in G74, G75
	0～9999.999（mm）	Tool retraction distance G74, G75
	0～9999.999（mm）	Finishing allowance in G76
	-9999.999～9999.999（mm）	Taper in G90, G92, G94, G96
I	-9999.999～9999.999（mm）	X vector between arc center and starting point
	0.06～25400（tooth/inch）	Metric thread tooth
K	-9999.999～9999.999（mm）	Z vector between arc center and starting point

Part Two Programming

Address	Value range	Function
F	1～8000（mm/min）	Feedrate per minute
	0.001～500(mm/r)	Feedrate per rev
	0.001～500（mm）	Metric thread lead
S	0～9999（r/min）	Specifying spindle speed
	0～9999（m/min）	Specifying spindle constant surface speed
	00～04	Multi-gear spindle output
	10～99	Subprogram call
T	0100～0800	Tool function, subprogram call
M	00～99	Miscellaneous function output, program executed flow, subprogram call
P	1～9999999（0.001s）	Pause time
	0～9999	Call subprogram number
	0～999	Call times of subprogram
	0.001～9999.999（mm）	X circular moving distance in G74, G75
	See Chapter 3.4, G76 Explanation	Thread cutting parameter in G76
	1～9999	Initial block number of finishing in the compound cycle command
Q	1～9999	End block number of finishing in the compound cycle
	0.001～9999.999（mm）	Z circular moving distance in G74, G75
	1～9999.999（mm）	First cutting depth in G76
H	01～99	Operand in G65
L	01～99	Thread heads in G92

The limited values described in Table 2-1-3-1 are for the CNC device, but the limited values for the machine are not described here. Please refer to the user manual, another user manual from the machine manufacturer when programming.

2.2 Relationship between Command Numerical Value and Decimal Point

In the system, some command cannot be with a decimal point, and No:11#0 sets whether the decimal point is used when programming, and the relationship between the command numerical value and the decimal point is shown in Fig.2-2-1:

Table 2-2-1

Address	Having a decimal point	NO:11#0=1	NO:11#0=0	Remark
X	Yes	Command：G1 X20 Positioning point：20	Command：G1 X20 Positioning point：0.02	
		Command：G4 X20 Delay：20s	Command：G4 X20 Delay：0.02s	
Z	Yes	Command：G1 Z20 Positioning point：20	Command：G1 Z20 Positioning point：0.02	
U	Yes	Command：G1 U20 Incremental value： 20mm	Command：G1 U20 Incremental value： 0.02mm	
W	Yes	Command：G1 W20 Incremental value： 20mm	Command：G1 W20 Incremental value： 0.02mm	
R	Yes			It is the same as the address X
I	Yes			
K	Yes			
P	No	Command：G4 P2 Delay：0.002s		It is not related to the decimal point
S	No			

2.3 Subprogram

2.3.1 Main Program and Subprogram

To simply the programming, when the same or similar machining path and control procedure is used many times, its program commands are edited to a sole program to call. The main program is defined to call others and the subprogram(end with M99) is to be called. They both take up the program capacity and storage space of system. The subprogram has own name, and can be called at will by the main program and also can run separately. The system returns to the main program to continue when the subprogram ends, which is shown below:

```
O0001;
G50 X100 Z100;
M3 S1 T0101;
G0 X0 Z0;
G1 U200 Z200 F200;
M98 P21006;
G0 X100 Z100;
M5 S0 T0100;
M30;
%
```

Call

Return

```
O1006;
G1 X50 Z50;
U100 W200;
G03 U30 W-15 R15 F250;
M99;
%
```

Main program Subprogram

Fig. 2-3-1-1

2.3.2 Subprogram Call（M98）

Command format:

M98 P○○○ □□□□

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

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

Command function: After other commands of current block are executed in M98, CNC calls subprograms specified by P instead of the next block, and subprograms are executed 9999 times at most.

Note: The system cannot call a subprogram in MDI mode.

2.3.3 Return from subprogram（M99）

Command format

M99 P○○○○

○○○○: Return to the block number （0001～9999）of the main program, the leading zero can be omitted.

Command function: After other commands of the current block (in the subprogram) are executed, the system returns to the main program and continues to execute next block specified by P, and calls a block following M98 of current subprogram when P is not input. The current program is executed repeatedly when M99 is defined to end of the main program.

Example: Execution path of calling subprogram (with P in M99) is shown in Fig. 2-3-3-1.

Execution path of calling subprogram (without P in M99) is shown in Fig. 2-3-3-2.

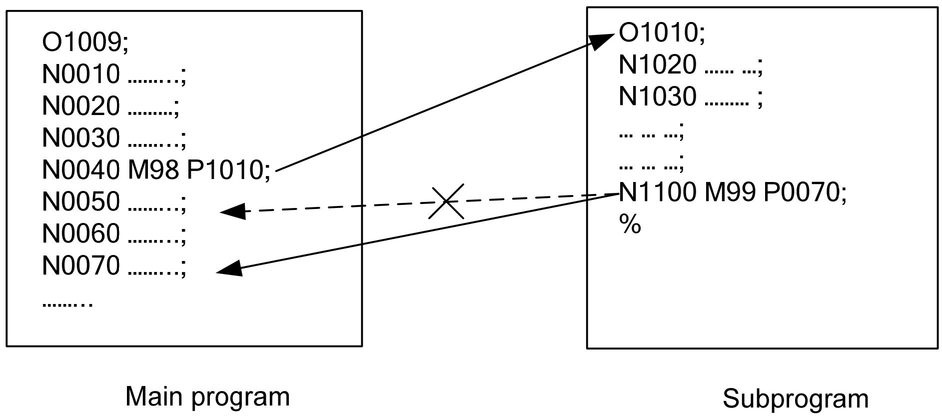


Fig. 2-3-3-1

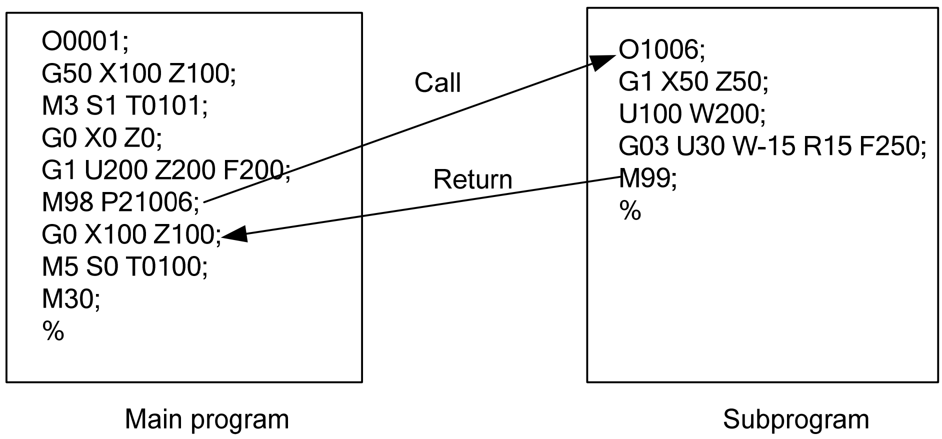


Fig. 2-3-3-2

The system can call fourfold-embedded subprograms, namely can call other subprograms in another subprogram (Fig.2-3-3-3 is an example of double-embedded).

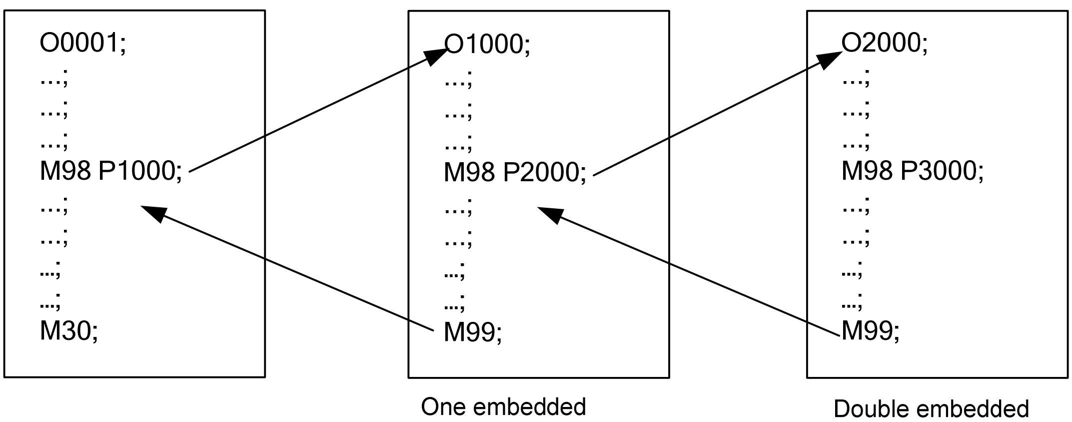
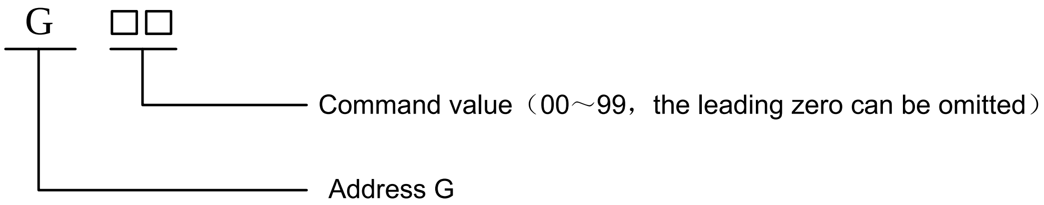


Fig. 2-3-3-3 double-embedded subprogram

CHAPTER THREE PREPARATORY FUNCTION G CODE

3.1 Category of Preparatory Function G Code

Preparatory function---A G Codeconsists of command address G and its following 2 digits numerical value, and is used for defining the motion path of the tool relative to the workpiece, setting the coordinates and so on. G Codes are shown in Fig. 3-1-1.



G words are divided into 6 groups (00, 01, 02, 03, 06, 07). That commands in the group 01 are simple and others are modal.

After G Codes are executed, their defined functions and states are valid until they are changed by others in the same group, the previous functions and states are cancelled.

The initial G Codeis the initial mode after the system is turned on, the commands in the initial mode include G00, G97, G98, G40, G21.

The defined functions and state are valid one time after the simple G Codeis executed, and it must be input again when it is executed every time

After the system is switched on, the valid modal G Codes which are not executed their functions or states are called initial mode G command. Take it as the initial mode G Codeto be executed when it is not be input after the system is switched on. The initial commands of GSK980TDa include G00, G40, G97, G98.

Several G codes (Group 00 and 01 must not be in the same block) which belong to different groups can be commanded in the same block. No.129 alarm occurs when more than two G codes which belong to the same group are commanded in the same block. When G codes with common word which belong to different group are in the same block, their functions are valid simultaneously and are unconcerned with their sequence. The system alarms when other G codes except for ones described in Table 3-1-1 or G codes which have no selection functions.

Table 3-1-1 G Codelist

Command	Group	Format	Explanation
*G00	01	G00 X(U)__Z(W) __	Positioning, rapid traverse, speed rate of each aixs set by parameters

G01		G01 X(U)__Z(W) __ __F	Linear interpolation
G02		G02 X(U)_Z(W)_R_(I_K_) F_	Circular interpolation CW
G03		G03 X(U)_Z(W)_R_(I_K_) F_	Circular interpolation CCW，
G04	00	G04 P_； or G04 X_；	Dwell
G10	00	G10 P__（parameter number）Q__（numerical value）	A program specifying a parameter function
G20	06	G20	Inch selection
*G21		G21	Metric selection
G28	00	G28 X(U)__Z(W) __	Return to reference point, and X, Z specifies the middle point
G31	00	G31 X（U）__ Z（W）__ F__	Skip function
G32	01	G32 X(U)__Z(W) __F(I)	Invariable pitch thread cutting
G33	01	G33 Z(W) __F(I)；G33 X(U) __F(I)	Tapping cycle
G34	01	G34 X(U)__Z(W) __F(I) K__	Variable pitch thread cutting
*G40	07	G40	Tool radius compensation cancel
G41		G41	Tool radius compensation (left)
G42		G42	Tool radius compensation (right)
G50	00	G50 X(U)__Z(W)__	Coordinate system setting
G51	00	G51 X（U）__ Z（W）__	Local coordinate system function
G65	00	G65 Hm P#I Q#J R#K	Macro code（see the following）
G70	00	G70 P(ns) Q(nf)	Finishing cycle
G71		G71 U(ΔD) R(E) G71 P(NS) Q(NF) U（ΔU）W(ΔW) F(F) S(S) T(T)	Outer roughing cycle

Part Two Programming

G72		G72 W (ΔD) R (E) G72 P(NS) Q(NF) U(ΔU) W(ΔW) F(F) S(S) T(T)	End roughing cycle
G73		G73 U (ΔI) W (ΔK) R (D) G73 P(NS) Q(NF) U(ΔU) W(ΔW) F(F) S(S) T(T)	Closed cutting cycle
G74		G74 R(e) G74 X(U) Z(W) P(Δi) Q(Δk) R(Δd) F(f)	End deep hole machining cycle
G75		G75 R(e) G75 X(U) Z(W) P(Δi) Q(Δk) R(Δd) F(f)	Outer/inner grooving cycle
G76		G76 P(m) (r) (a) Q(△dmin) R(d) G76 X(U) Z(W) R(i) P(k) Q(△d) F(L)	Compound thread cutting cycle
G90	01	G90 X(U)__ Z(W)__ R__ F__	Outer, inner turning cycle
G92		G92 X(U)__ Z(W)__ R__ F(I) __J __K__	Thread turning cycle
G94		G94 X(U)__ Z(W)__ R__ F__	End turning cycle
G96	02	G96 S	Constant surface control
*G97		G97 S	Constant surface control cancel
*G98	03	G98	Feed per minute
G99		G99	Feed per rev

Note 1: When the system is turned on, it is in the state of G Codewith *.

Note 2: G Codes in Group 00 are simple.

3.2 Simple G Code

3.2.1 Rapid Positioning（G00）

Command format：G00 X（U）_ Z（W）_ ；

Function: X, Z rapidly traverses at the respective traverse speed to the position specified by X(U), Z(W).

Explanation：X（U）：absolute (incremental) coordinate of X positioning end point;

Z（W）：absolute (incremental) coordinate of Z positioning end point;

1. X, Z rapidly traverses at the respective traverse speed and their combined path is really

not linear, so maybe they cannot reach the end points simultaneously, please pay attention to it when programming (Fig. 3-2-1-1).

2. X, Z rapidly traverses at the speed separately set by P21, P22 are adjusted by pressing 

on the operation panel, or are performed by selecting manual rapid override which has five grades including Fo, 25％, 50％, 75％, 100％. (Fo speed is set by P32)

3. The tool does not traverse when there is a positioning parameter after G00, the system only changes the current tool traverse mode into G00.

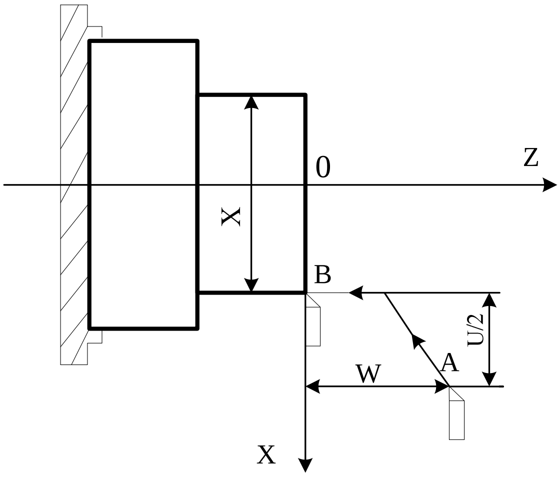


Fig. 3-2-1-1

Example：the tool rapidly traverses from A to B, which is shown in Fig. 3-2-1-2:

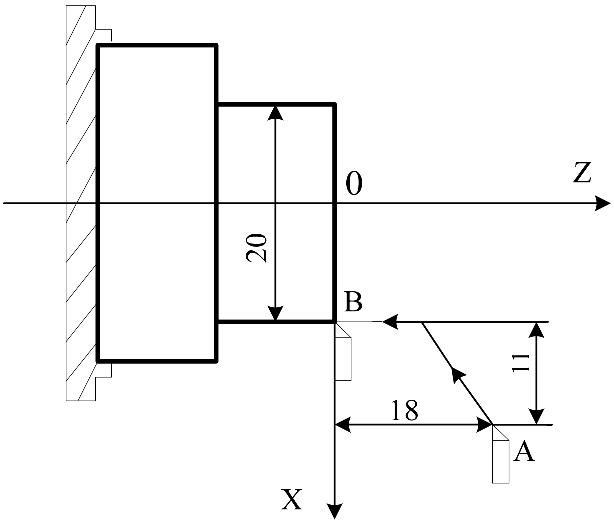


Fig. 3-2-1-2

Programming:

G0 X20 Z0；（absolute programming, diameter programming）

G0 U-22 W-18；（incremental programming, diameter programming）

G0 U-22 Z0；（compound programming, diameter programming）

3.2.2 Linear Interpolation（G01）

Command format：G01 X（U）_ Z（W）_ F_；

Function：the tool traverses to the specified position at the feedrate （mm/min）specified by F. The interpolation path is shown in Fig. 3-2-2-1.

Explanation：X（U）：absolute (incremental) coordinate of X interpolation's end point;

Z（W）：absolute (incremental) coordinate of Z interpolation's end point;

F：is combined federate of X, Z, is modal. It value is related to G98 OR G99, which is shown in 3-2-2-1：

Table **3-2-2-1**

	G98（mm/min）	G99（mm/r）
Value range	1~8000	0.001~500

Command path:

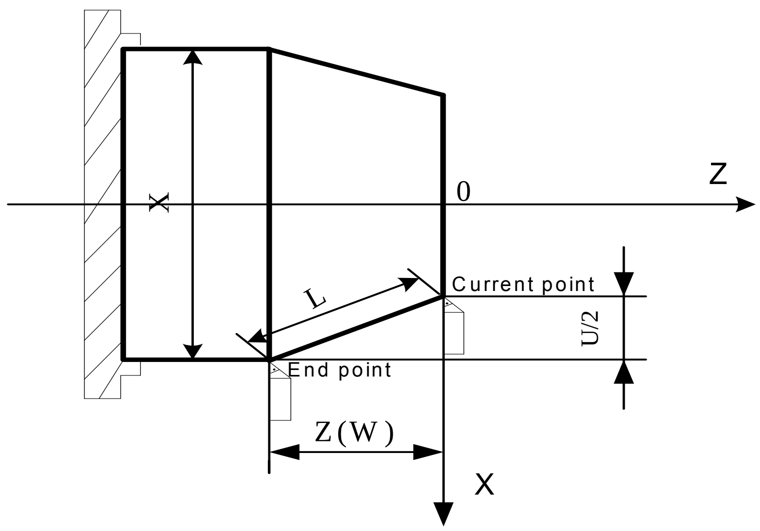


Fig. 3-2-2-1

1. Feedrate specified by F is valid until a new F value is specified. Feedrate specified by F is counted by linear interpolation. When F is not commanded in programs, feedrate uses F feedate of initial speed.（refer to P103 about setting）.

   For interpolation mode of two axes simultaneously moving, F is specified to the composite feedrate of two axes.

   It shows from Fig. **3-2-2-1**：

X feedrate $Fx = \frac{U}{L} \times F$;

Z feedrate $Fz = \frac{W}{L} \times F$ （L＝$\sqrt{U^2/4+W^2}$ ，U is a diameter value）

2. **P27** can set the upper of cutting feedrate F. When the actual cutting feedrate (federate after using override) exceeds the upper, NO.11 alarm occurs, feedrate unit is mm/min.
3. When the positioninG Codefollowing G01 is not commanded, the tool does not traverse, the system only changes the current tool traversing mode to G01.
4. Pressing 

can tune the feedrate override, which is divided into 16 grades from 0%~150%.

Example: compile the linear interpolation program from current point to end point, which shown in Fig. **3-2-2-2**.

Program (diameter programming):

G01 X60.0 Z-25;（absolute programming）

G01 U20.0 W-25.0;（incremental programming）

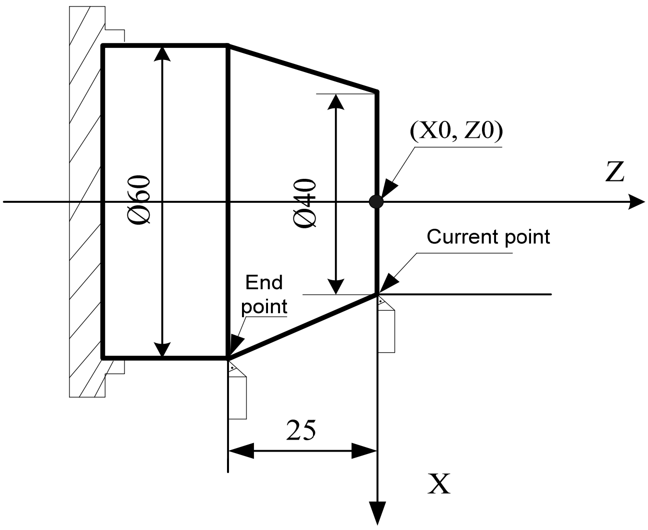


Fig. 3-2-2-2

3.2.3 Circular Interpolation（G02/G03）

Command format:

$$\left\{\begin{matrix} G02 \\ G03 \end{matrix}\right\} X(U)\_\ Z(W)\_ \left\{\begin{matrix} R\_ \\ I\_K\_ \end{matrix}\right\} F\_;$$

Function: X, Z from starting point (the position before the current block runs) moves along the radius specified by R or CW/CCW interpolates along the circle center defined by I, K value to end point specified by X(U), Z(W).

G02 motion path is an arc from starting point to end point along CW (the rear post tool coordinate system)/CCW (the front tool post coordinate system), which is shown in Fig.3-2-3-1.

G03 motion path is an arc from starting point to end point along CCW (the rear post tool coordinate system)/CW (the front tool post coordinate system), which is shown in Fig.3-2-3-2.

Explanation：X（U）：absolute (incremental) coordinate of X circular interpolation’s end point;

Z（W）：absolute (incremental) coordinate of Z circular interpolation’s end point;

R：arc radius;

I：X difference value of circle center relative to starting point of arc; (radius command)

K：Z difference value of circle center relative to starting point of arc;

X, U, Z, W, I, K range: -9999.999 mm～9999.999 mm, R range: 0mm～9999.999mm

F：cutting speed of arc.

Command path:

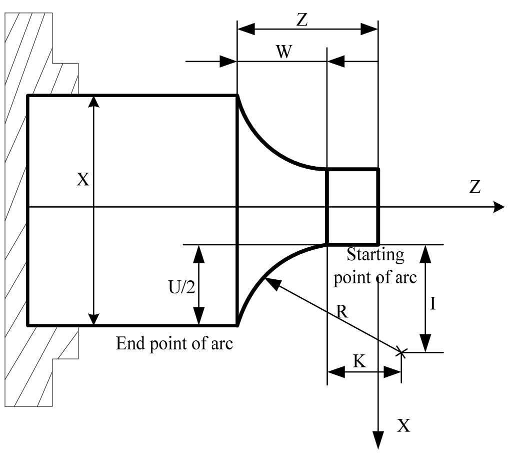


Fig.3-2-3-1 G02 path

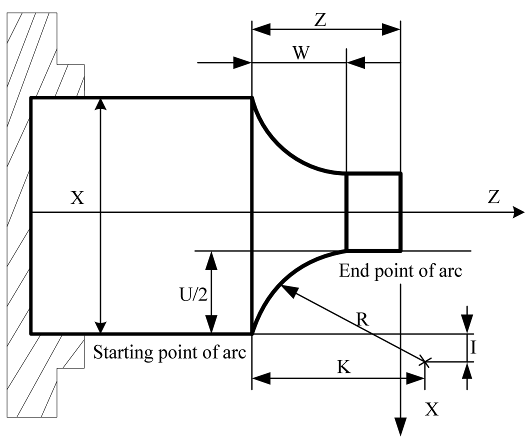


Fig.3-2-3-2 G03 path

CW or CCW interpolation is related that the system uses the front tool post coordinate system or the rear tool post coordinate system. The system uses the front tool post coordinate system, which is shown in Fig. 3-2-3-3.

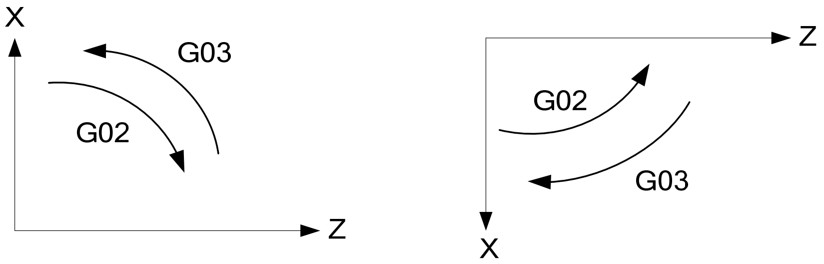


Fig. 3-2-3-3

Example: G02 compiles a program shown in Fig. 3-2-3-4:

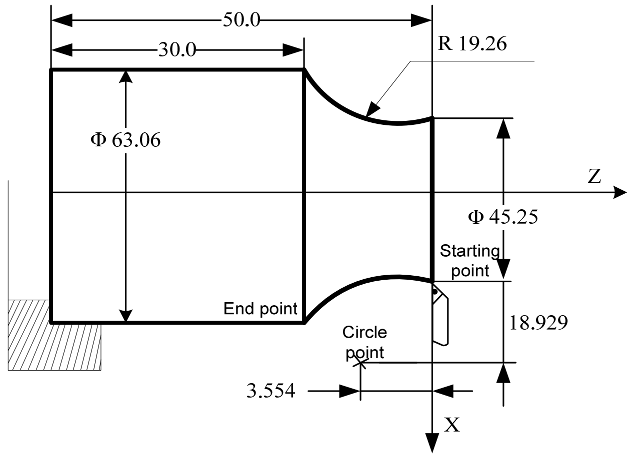


Fig. 3-2-3-4

Program（the current point of the tool is the starting point）:

G02 X63.06 Z-20 R19.26 F300 ； or

G02 U17.81 W-20.0 R19.26 F300 ； or

G02 X63.06 Z-20 I18.929 K-3.554 F300 ； or

G02 U17.81 W-20.0 I18.929 K-3.554 F300 ；

Note 1: the arc center is specified by I, K which separately corresponds to X, Z (Fig. 3-2-3-1, Fig.3-2-3-2). I, K mean the vector component from starting point to circle center, with signs.

Note 2: when I, K or, R is not specified in the command format, the system executes the interpolation along G01 path.

Note 3: one or all of X(U), Z (W) can be omitted. Omitting one means the starting point and end point of the axis are consistent; omitting all means the starting point and end point are in the same position.

Note 4: when X (U), Z (W) are omitted simultaneously, and I, K are circle center, which means the arc path is a full circle; I, K and R cannot be command simultaneously, otherwise, the system alarms.

Note 5: when I=0 or K=0, they can be omitted.

Note 6: when R is specified, R>0, the specified arc is less than 180 degree; when the arc is more than 180 degree, I, K are specified; R, I, K are simple.

Note 7: when R error range exceeds the value set by No. 98, No.26 alarm occurs.

Note 8: in MDI mode, I, K are valid but I, K values are not displayed.

3.2.4 Chamfering/Cornering Arc

Command format: **C_: chamfer**

B_: chamfer arc transition

Function: the above is added to the end of the block in G01 or G02, G03, the chamfering at the cornering or the arc transition is executed automatically.

Explanation: the chamfering following C specifies the distance from the virtual inflection point to corner starting point and end point, the virtual inflection point is presumed not to execute the corner, the corner point actually exists, which is shown below:

(1) G01 U100 W-50 C10;
(2) W-100;

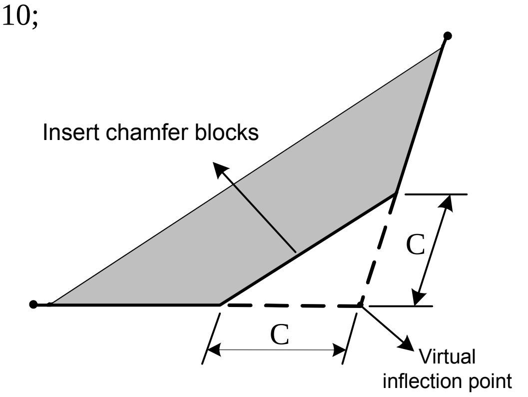


Fig. 3-2-4-1

After the chamfer arc transition is B, the chamfer arc radius is specified, which is shown below:

（1）G01 U100 W-50 B10；
（2）W-100；

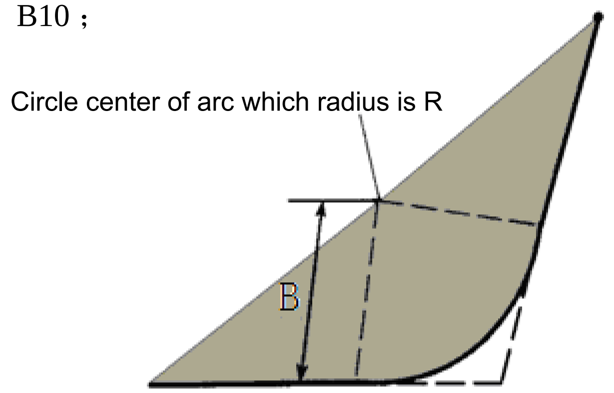


Fig. 3-2-4-2

Limits:

1. Cornering and chamfering arc are executed only in the specified plane, and cannot be executed by the parallel axes.
2. After a block for cornering and chamfering arc transition, G01 or G02, G03 must be followed, and the cornering and chamfering arc transition are disabled when there is no these commands in the following block.
3. The system alarms when the inserted block for the chamfering or arc transition causes the

tool exceeding the previous interpolation movement range.

4. The Chamfering or cornering arc transition cannot be specified in the block where the coordinate system is changed or which is followed by the reference point return is executed.
5. The chamfer arc transition cannot be specified in the block used for the thread machining.
6. The chamfering and cornering values cannot be negative.

3.2.5 Dwell G04

Command format：G04 P__ ；or

G04 X__ ；

G04 U__ ；

Function：When G04 executes the dwell operation, the system delays the specified time to execute the next block. The specified time range: 0.001s～9999.999s.

When P, X or U is specified, it means the exact stop between block, dwell time is influenced by No. 031, the bigger the parameter set value, the longer the dwell time is;

Explanation：when No:11#0 is set to 1, the unit is shown in Fig. **3-2-5-1**:

Table 3-2-5-1

Address	P	X	U
Unit	0.001s	1s	1s

When No:11#0 is set to 0, the unit is shown in Fig. **3-2-5-2**:

Address	P	X	U
Unit	0.001s	0.001s	0.001s

Note 1: when G04 is execute, P and X or P and U are in the same block, X, U are valid, P is invalid.

Note 2: G04 and M98 cannot be in the same block, otherwise, the alarm (eer76) occurs.

Note 3: G04 and other commands in the Group 01 cannot be in the same block, otherwise, the alarm (err131) occurs.

3.2.6 Programs Specifying Parameter Function （G10）

Command format：G10 P（parameter number） Q（numerical value）

Function：When G10 is specified, the actual output of related parameters from the one specified by G10 is executed

Example：G10 P45 Q1000

After the block is executed, No. 45 value is automatically changed into 1000

Explanation：（1）Parameter range specified by P：all data parameter of the system；

（2）Q range and unit are related the specified parameter;

（3）Parameter（Q）set value has no decimal point. It automatically cleared away by the system when the decimal point is input when programming.

3.2.7 Machine Zero Return（G28）

Command format：G28 X（U）_ Z（W）_ ；

Function：The commanded axis goes through the middle point specified by X (U), Z(W) to return to the machine zero. One or two axes can be specified in G28.

Table 3-2-7-1

Command	**Function**
G28 X（U）	X returns to machine zero, Z remains unchanged
G28 Z（W）	Z returns to machine zero, X remains unchanged
G28	X, Z remain unchanged
G28 X（U）___Z（W）	X, Z return to machine zero simultaneously

Explanation： G28 operation process (Fig3-2-7-1):

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

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

(3) When the machine is not locked and after the machine reference point return is completed, the machine zero return indicator is ON and the machine coordinates are cleared.

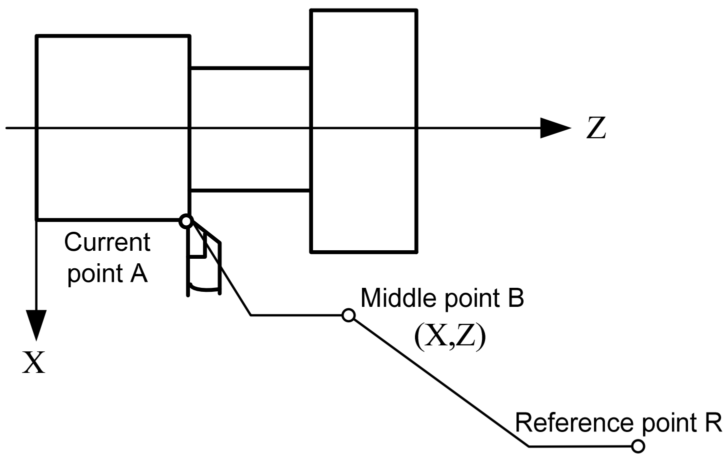


Fig. 3-2-7-1

Note 1: After the system is turned on, the system has not executed the manual machine zero return firstly, the path from the middle point to the machine zero is the same that of the manual machine zero return when G28 is executed.

Note 2: A point→B point and B point →R point, X, Z moves at its respective speed, so, their paths are not linear.

Note 3: The machine zero return function cannot be executed when there is no the machine zero installed on the machine.

Note 4: When No.006: Bit6 is set to 1, the system is firstly turned on to execute G28 and after the machine zero return is executed, the set coordinate system is valid; when G28 is executed again and after the machine zero return is executed, the set coordinate system is invalid.

3.2.8　Workpiece Coordinate System Setting（G50）

1）Workpiece coordinate system setting

Command format：G50 X_ Z_ ；

Function：Set a workkpiece coordinate system. Two code parameters specify the absolute coordinate value of tool nose position of current tool post on a new workpiece coordinate system. G50 does not make the motion axis move.

Explanation： X ：X absolute coordinate of current tool nose in the workpiece coordinate system;

Z ：X absolute coordinate of current tool nose in the workpiece coordinate system;

1. After the coordinate system is established, the position of the absolute command is presented in the coordinate system till one new coordinate system is established by G50.
2. When the diameter programming is specified, the diameter is specified to X; when the radius programming is specified, the radius is specified to.

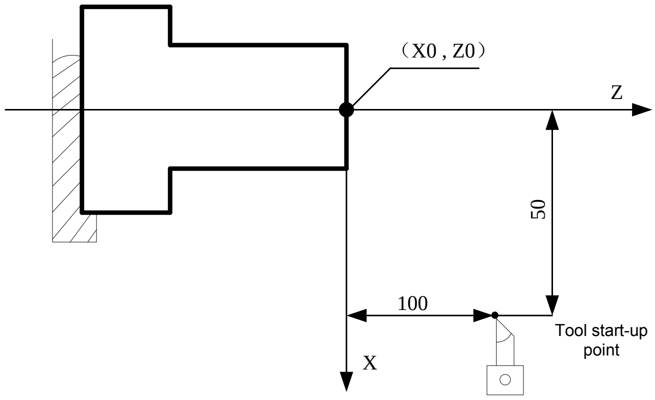


Fig. 3-2-8-1

After G50 X100 Z100 is executed, the workpiece coordinate system is thus set (shown in Fig. 3-2-8-1) and the tool nose's current position in the workpiece coordinate system is defined. Refer to Section 4.4 Tool Setting for coordinates setting methods.

Note 1: G50 cannot be in the same block with the commands included to Group 01, otherwise, ERR131 alarm occurs.

Note 2: When using G50 sets the coordinate system in the tool offset cancel mode. After the coordinate is set, the absolute coordinates are those of G50 set value; the tool offset cancel is executed in MDI mode: "T0100 G00 U0 W0". Suppose that the current tool offset state is T0101.

Note 3: When using 50 sets the coordinate system in tool offset state, there are two kind of displaying absolute coordinates:

A. The tool offset has been executed(have movement command after tool offset) and after the setting is done, the absolute coordinates are the one set by G50, which is shown below:

Table 3-2-8-1

Program(execute tool compensation in coordinate offset mode)	Absolute coordinate display value	No. 01 tool compensation value
G0 X0 Z0	X：0 Z：0	X：-12 Z：-23
T0101	X：12 Z：23	
G0 X0 Z0	X：0 Z：0	
G50 X20 Z20	X：20 Z：20	

B. The tool offset has not executed(have no movement commands after tool offset), including tool offset cancel and tool offset setting, after the setting is done, the absolute coordinates are the tool offset values, which is shown below:

Table 3-2-8-2

Program(execute tool compensation in coordinate offset mode)	Absolute coordinate display value	No. 01 tool compensation value
G0 X0 Z0	X：0 Z：0	X：-12 Z：-23
T0101	X：12 Z：23	

G0 X50 Z50	X：50 Z：50	
T0100	X：38 Z：27	
G50 X20 Z20	X：8 Z：-3	

Table 3-2-8-3

Program(execute tool compensation in coordinate offset mode)	Absolute coordinate display value	No. 01 tool compensation value
G0 X0 Z0	X：0 Z：0	X：-12 Z：-23
T0101	X：12 Z：23	
G50 X20 Z20	X：32 Z：43	

2）Coordinate system translation

Command format：G50 U_ W_ ；

Function：The tool nose position in the previous absolute coordinate system translates the distance set by a parameter. Namely, the tool nose position in the new coordinate system corresponding to the previous absolute coordinate system is X+U，Z+W.

Explanation：When the diameter programming is specified, X is done by diameter; when the radius programming is specified, X is done by radius.

Note: There is no M, S, T in the block of G50, otherwise, the system alarms.

3.2.9 Workpiece Coordinate System Offset（G51）

Command format：G51 X（U） Z（W）

Function: G51 is specified to complete the workpiece coordinate system offset when programming.

Explanation：When the absolute coordinate programming is used**,** for example**,** G51 X_ Z_， means that the current coordinate system offsets to the position specified by X, Z;

When the incremental coordinate programming is used, for example, G51 U_ W_，means the current coordinate system offsets the displacement vlaues specified by U, W;

Specify G51, X0, Z0 to cancel the workpiece coordinate offset and recover the previous workpiece coordinate system.

Example 1：Use “G51 X_ Z_ ” to execute the offset

Program	Absolute coordinates	Machine coordinates	Explanations
G50 X100 Z100	X100 Z100	X100 Z100	G50 sets coordinate system

G51 X-10 Z-10	X110 Z110	X100 Z100	G51 executes the offset corresponding to the previous workpiece coordinate system instead of the current coordinate system.
G51 X20 Z20	X80 Z80	X100 Z100	
G0 X10 Z10	X10 Z10	X30 Z30	
G51 X0 Z0	X30 Z30	X30 Z30	Cancel the workpiece coordinate system offset

Example 2：Use “G51 U_ W _ ” to execute the offset.

Program	Absolute coordinates	Machine coordinates	Explanations
G50 X100 Z100	X100 Z100	X100 Z100	G50 sets coordinate system
G51 U-10 W-10	X110 Z110	X100 Z100	G51 executes the offset corresponding to the previous workpiece coordinate system instead of the current coordinate system.
G51 U20 W20	X90 Z90	X100 Z100	
G0 X10 Z10	X10 Z10	X20 Z20	
G51 X0 Z0	X20 Z20	X20 Z20	Cancel the workpiece coordinate system offset

Note 1: Executing G51, G50, G28 and the manual machine zero return operation (machine zero return can automatically set the coordinate system) can automatically cancel offset value.

3.2.10 Feed per minute (G98)

Command format：G98 Fxxxx

Function：G98 is the feed per minute. The distance the tool traverses per minute is commanded by the numerical value following F, unit: mm/min.

Explanation：G98 is modal. Once G98 state is commanded, it is valid till G99 is executed.

3.2.11 Feed per Rev (G99)

Command format：G99 Fxxxx

Function： G99 is the feed per rev. The tool feed amount per the spindle rev is commanded by the numerical value following F, unit: mm/r.

Explanation： G99 is modal. Once G99 state is commanded, it is valid till G98 is executed.

Table 3-2-11-1　Difference between feed per minute and feed per rev

	Feed per minute	**Feed per rev**
Address designation	F	F
Command designation	G98	G99
Range	1 mm/min～8000mm/min (F1～F8000)	0.001 mm/r～500.00mm/r (F0.001～F500)
Limited value	Feed per minute, feed per rev are limited in some speed. The limited values are set the machine manufacturer. (The limited value is the numerical value following the override)	
Override	Feed per minute, feed per rev are adjusted by the override 0～150％.	

Note 1: When the position encoder's speed is below 1 r/min, the speed becomes uneven. The speed below 1 r/min can be used when the even speed is not required. Below 1r/min, the slower the speed is, the more uneven the speed is.

Note 2: G98, G99 are modal. They are valid after they are commanded till another command is executed.

Note 3: F code can permit inputting up to 7 digits. The system alarms when the input feedrate exceeds the limit.

Note 4: The position encoder must be installed on the spindle when the feed per rev is executed.

3.2.12　Constant Surface Control (G96, G97)

Command format: G96 S__ ；constant surface control, and the numerical value following S specifies the surface speed of tool nose in the tangential direction.

G97 S__ ；cancel the constant surface control code. The numerical value following S specifies the spindle speed.

Function： The constant surface is defined that the surface speed following S is constant. along the tool's position changing, the spindle speed is counted according to the previous given surface speed, the spindle speed is converted into the corresponding voltage to output the spindle control unit, which gets the constant relations between the instantaneous position of tool and the workpiece's surface.

Explanation： The surface speed unit is shown below:

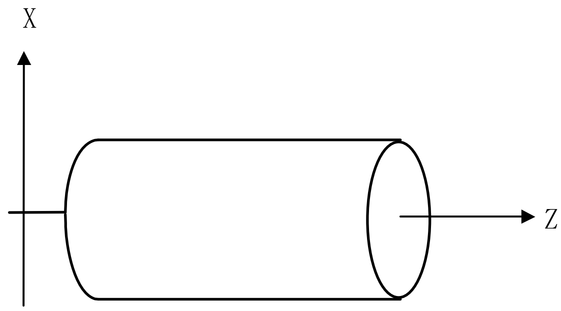
Part Two　Programming

Table 3-2-12-1

Input unit	Surface speed unit
Metric	m/min

The surface speed units are different according to the machine manufacturers.

When the constant surface control is executed, the rotary axis must be set on Z (X=0) of the workpiece coordinate system.



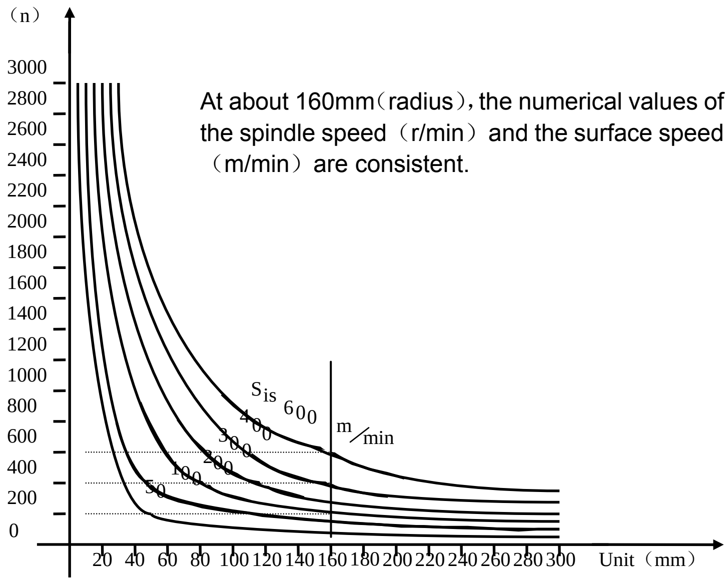


Fig. 3-2-12-1

1） Spindle speed override

The specified surface speed or speed can use the override 50%，60%，70%，80%，90%，100%，110%，120％ according the spindle override selection.

2） Max. spindle speed limit

Using the numerical value following G50 S can command max. spindle speed (r/min) in the constant surface control state.

G50 S__ ；

In the constant surface control state, the spindle speed is limited to its max. speed when it exceeds the above specified value.

3） Constant surface control in G00

For the block in G00, when the constant surface speed control is executed, the system counts the surface speed at the end point of the block instead of the surface speed of the tool position which is changed constantly, which reason is that the rapid traverse does not execute the cutting.

Note 1: When the system is turned on, the state which does not set max. spindle speed is the unlimited state. For the limit, the system is only in G96 instead of G97.

Note 2: The lower limit value of the spindle speed in the constant surface speed control is controlled by P78. P78=100: when G50 S0 is executed, the spindle speed is 100 r/min.

Note 3: The commanded S value in G97 still retains in G96, i.e. it remain unchanged in G97. its value recovers when the system returns to G96 state.

G96 S50；（50m/min）

G97 S1000；（1000r/min）

G96 X3000；（50m/min）

Note 4: It does not run when the machine is locked, X coordinate value in the corresponding program changes and the system executes the constant surface control.

Note 5: The constant surface control is valid when thread cutting is executed, so, the constant surface control is invalid in G97, which makes the spindle rotate at the same speed.

Note 6: Feed per rev (G99) is still valid without useful meaning in G96.

Note 7: When G96 state is changed into G97 and there is no S code (r/min) in G97, the final speed in G96 is used as S code in G97.

N100 G97 S800；（800r/min）

…

N200 G96 S100；（100m/min）

…

N300 G97；（Xr/min）

X is speed of the block which is front of N300, i.e. when G96 state is changed into G97, the spindle speed does not change. When G97 state is changed into G96, S value in G96 is value. S=0 m/min when S value is not commanded.

3.2.13 Skip Function （G31）

Command format: G31 X（U）__ Z（W）__ F__

Function 1：About the standard G31 function, the system skips to execute the next block when the external skip signal is input.

Example 1： G01 X0 Z0 F200

G31 X100 Z100 F100 when the block is running, the skip signal input is value, the system skips to the next block from the current block.

G01 X-50 Z-50 F150

Function 2： About the extended G31 function, the system skips to execute the next block when the external skip signal is input. The system continuously runs limitlessly in some direction when there is no skip signals.

Example 2: G01 X0 Z0 F200

G31 X100 Z100 F100 the system skips to execute the next block when the external skip signal is input validly. The system continuously runs limitlessly in some direction when there is no skip signals.

G01 X-50 Z-50 F150

Example ：

Function 1：P145=0: when there is the skip signal input, the system skips to the next block from the current; the system runs to the position（X100，Z100） specified by the block, and then executes the next block when there is no skip signal input.

Function 2：P145=1: when there is the skip signal input, the system skips to the next block from the current; The system continuously runs limitlessly in some direction till the skip signal is input or the limit is executed when there is no skip signals.

Note 1: When G31 extension function is used, the system runs limitlessly in some direction, so, the hard limit should be installed on the machine.

Note 2: The input interface of skip signal is defined to XS39 12 when delivery.

3.2.14 Thread Cutting with Invariable Pitch（G32）

Command format： G32 X（U）__Z（W）__ F（I）__；

Function： Two axes moves simultaneously from the starting point (the position before G32 runs) to the end point specified X (U), Z (W) to execute the thread cutting, which is path shown in Fig. 3-2-14-1). The straight thread with invariable pitch, taper thread and end thread can be machine in G32. the tool retraction groove is needed in G32.

Explanation： X（U）：absolute (incremental) coordinate of X thread cutting's end point；

Z（W）：absolute (incremental) coordinate of Z thread cutting's end point；

F：metric thread pitch, i.e., the movement corresponding to the workpiece when the spindle rotates one rotation, range：0.001 mm～500mm, it is modal；

I ：tooth number per inch for inch thread, range: 0.06 tooth/inch～25400 tooth/inch，it is modal.

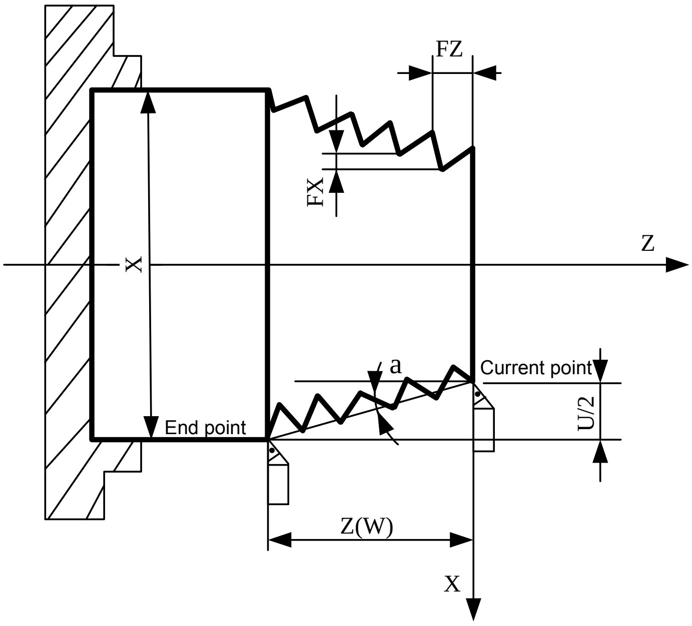


Fig. 3-2-14-1

The system uses a long axis and a short axis, which is shown in Fig.3-2-14-2.

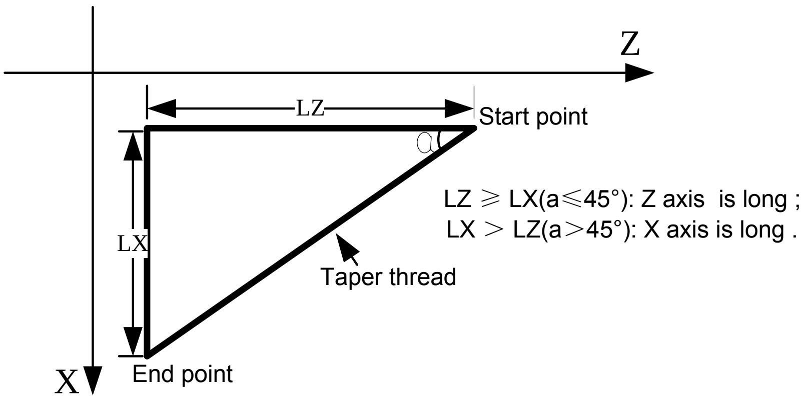


Fig. 3-2-14-2

Notes:

1. **Start and end of thread cutting, some lead is wrong because of acceleration and deceleration, the commanded thread length should be longer than the required, which is shown in Fig. 3-2-14-3.**
2. **The federate override is invalid, and is the constant value 100% in the course of thread cutting.**
3. **The spindle must be started when thread cutting, otherwise, the system alarms; the spindle cannot stop in the course of thread cutting..**
4. **The spindle override is invalid in the course of the thread cutting. Changing the spindle override brings wrong thread because of acceleration/deceleration.**
5. **The feed hold function is invalid in the course of the thread cutting. After he first non thread cutting block is executed when the thread cutting is executed, stopping the single block run stops the thread cutting.**
6. **When the previous block is for thread cutting and the current is also for it, the system does not**

check the one rotation signal of the spindle position encoder at beginning of thread cutting.

7. The spindle speed must be constant. When the spindle speed changes, the thread appears deviation.

8. The system alarms when F, I are in the same block simultaneously.

Example 1：Using G32 compiles a program which path is shown in Fig. **3-2-14-3**，its thread pitch：4mm

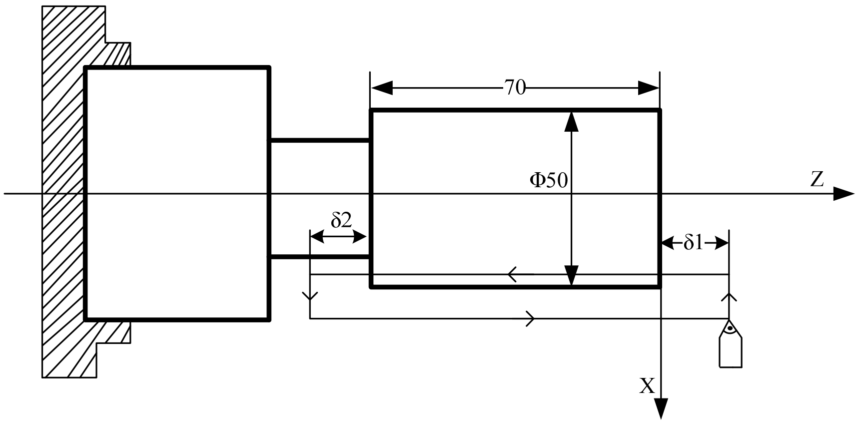


Fig. 3-2-14-3

A program：δ1 = 3mm，δ2 = 1.5mm，tool cutting depth 1mm（single side），cut-in two times.

G0 X100 Z50；（rapid positioning）

M03 S200；（start spindle, speed 200）

T0101；（call thread tool）

G0 X49 Z3；（rapid positioning，the first cut-in 1mm）

G32 W-74.5 F4.0；

G00 X55；

W74.5；

X48；（rapid positioning，the second cut-in 1mm）

G32 W-74.5 F4.0；

G00 X55

W74.5；

G0 X100 Z50 M05；

M30；

Example 2：Using G32 compiles a program which path is shown in Fig. **3-2-14-4**，the length axis is Z, it thread pitch：3mm

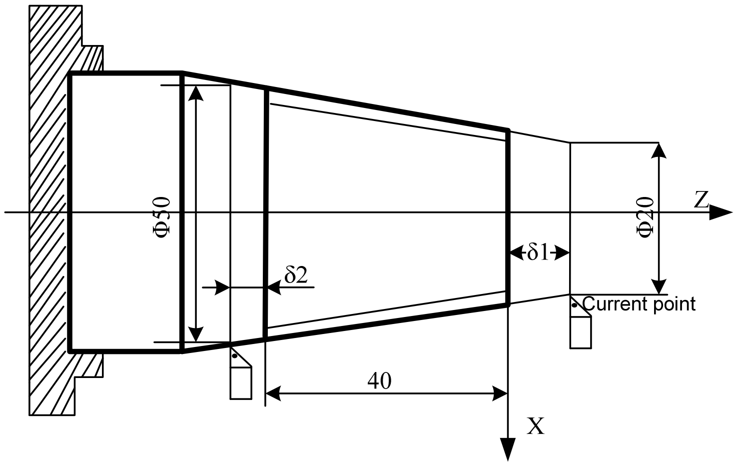


Fig. 3-2-14-4

A program：δ1 = 2mm，δ2 = 1mm，tool cutting depth 1mm（single side）, cut-in two times.

G0 X100 Z50；（rapid positioning）

M03 S200；（start the spindle，speed 200）

T0101；　（call thread tool）

G00 X19 Z2；　　　　（rapid positioning，the first cut-in 1mm）

G32 X49 Z-41 F3；

G00 X55；

Z2；

G0 X18；　　　　　（rapid positioning，the second cut-in 1mm）

G32 X48 Z-41 F3；

G0 X55；

Z2；

G0 X100 Z50 M05；

M30；

3.2.15　Thread Cutting with Variable Pitch（G34）

Command format：G34 X（U）__Z（W）__ F（I）__K__；

Function：　Two axes moves simultaneously from the starting point (the position before G34 runs) to the end point specified X (U), Z (W) to execute the thread cutting. The straight thread with variable pitch, taper thread and end thread can be machine in G32. the tool retraction groove is needed in G34.

Explanation：　X（U）：absolute (incremental) coordinate of X thread cutting’s end point；

Z（W）： absolute (incremental) coordinate of Z thread cutting's end point；

F：metric thread pitch, pitch of thread's starting point, range：0.001 mm～500mm, it is modal；

I： tooth number per inch for inch thread, range: 0.06 tooth/inch～25400 tooth/inch，it is modal；

K：pitch's increment or decrement of one rotation of the spindle. K range：±0.000001mm～±500mm.

±0.000001 inch～±19.685 inch

When K value increasing or decreasing makes pitch exceed the permissive value or reduce to 0 or the negative value, the system alarms; at the same time, when the pitch changes greatly, the acceleration/deceleration in the course of thread machining greatly becomes slow, which cause the wrong pitch.

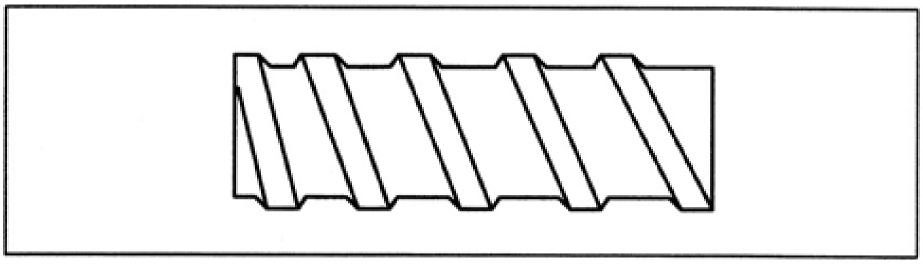


Fig. 3-2-15-1

Note: In MDI mode, I, K value can be input and cannot be displayed. Do not run G34 in MDI mode.

3.2.16 Tapping Cycle（G33）

Command format：G33 Z（W）__ F（I）__；G33X(U)__F(I)__；

Function1: Z moves from starting point (the position before G33 runs) to end point specified by Z (W) to execute the rigid tapping (its path is shown in Fig.3-2-16-1), the spindle rotates reversely and Z returns to the starting point of the cycle.

Function2：X moves from starting point (the position before G33 runs) to end point specified by X (U) to execute the rigid tapping, the spindle rotates reversely and X returns to the starting point of the cycle. (Its execution is the same that of Z but its direction is different from that of X)

Explanation：Z（W）：absolute (incremental) coordinate of Z thread cutting's end point；

X（U）：absolute (incremental) coordinate of X thread cutting's end point；

F: metric thread pitch, i.e. the tool movement corresponding to the workpiece when the spindle rotates one rotation, range：0.001 mm～500mm，it is modal；

I：tooth number/inch for inch thread，range：0.06 tooth/inch～25400 tooth/inch，it is modal.

G 33 Z tapping cycle process:

1） Z tool infeed tapping；

2） Stop the spindle；

3） Wait the spindle to exactly stop；

4） The spindle rotates reversely；（opposite to previous rotation direction）

5） Z executes the tool retraction to starting point of machining；

6） The spindle remains rotation, but the rotation direction and the tool retraction direction are consistent, G33 runs completely;

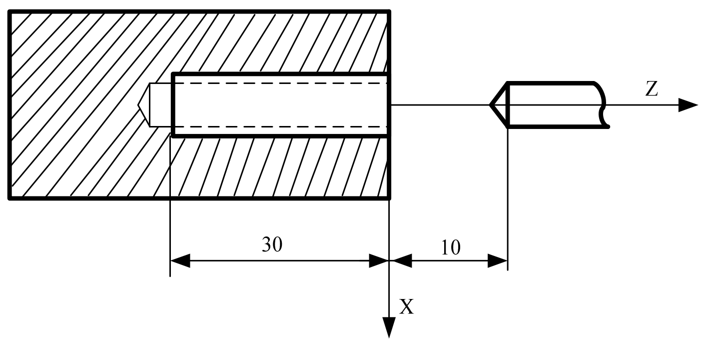


Fig. 3-2-16-1 G33 tapping cycle

Example：single-thread with pitch 2mm

N0010	M3 S300；	start the spindle
N0020	G00 X0 Z3；	rapid positioning
N0030	G33 Z-30 F2 ；	tapping cycle
N0040	G0 Z100 X30 ；	return to starting point
N0050	M30 ；	end of program

Note 1: G33 is rigid tapping. The spindle decelerates in some time after the spindle stop signal is valid. At the moment, Z rotates along with the spindle till the spindle exactly stops, the bottom hold of the thread when the actual machining should be deeper than the actual demand, and the concrete length is determined by the spindle speed when tapping and by whether there is spindle brake.

Note 2: Other notes are the same those of G32.

Note 3: G33 are modal.

Note 4: I value can be input in MDI mode and cannot be displayed. Do not execute G33 in MDI mode.

3.3 Single Fixed Cycle Command

For some special roughing, the same machining path are cut repetitively because of much cutting, the system can use a fixed cycle function, adopting a block to complete the machining operations by many blocks. When the repetitive cutting is executed, the corresponding numerical value is changed, which is valid to simply programs. The single fixed cycle commands include outer (inner) cutting cycle G90, thread cutting cycle G92 and end cutting cycle G94.

The diameter is specified in the following figures. When the radius is specified, U/2 replaces U, X/2 replaces X.

3.3.1 Outer（Inner）Cutting Cycle（G90）

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

Function：Using G90 can realize the single cycle machining of cylindrical surface, and taper surface, and the tool returns to the starting point after the cycle is completed. The dotted lines(R) (R) in Fig.3-3-1-1, Fig. 3-3-1-2 mean to be the rapid traverse, and the real lines (F) means to be cutting feed. The sign of numerical value following the address U is determined by X direction of the path 1 and the one following W is determined by Z direction of the path 2.

Explanation：X.Z：absolute coordinates of circular end point, unit：mm；

U.W：coordinates of circular end point corresponding to the circular starting point, unit：mm；

R：it is radius difference between starting point and end point of taper cutting, unit：mm；

F：it is combined speed of X, Z in the cycle, and it is modal.

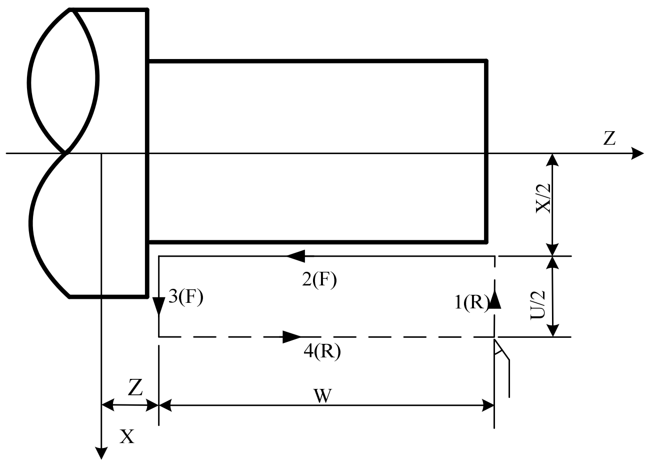


Fig. 3-3-1-1

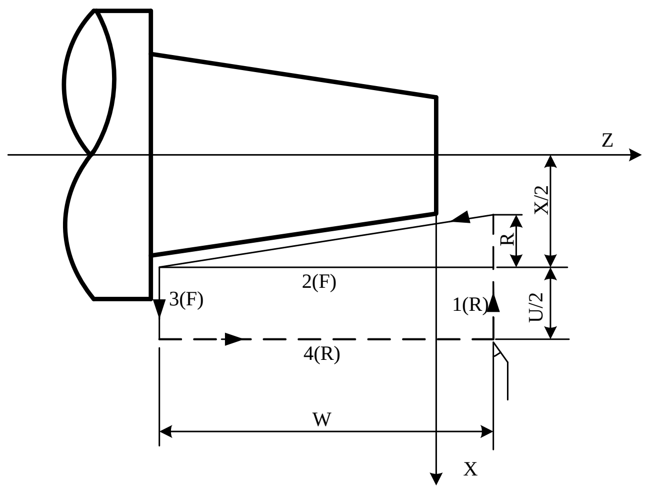


Fig. 3-3-1-2

G90 has four kind of paths, which are shown in Fig. 3-3-1-3 according to the different start-up tool positions.

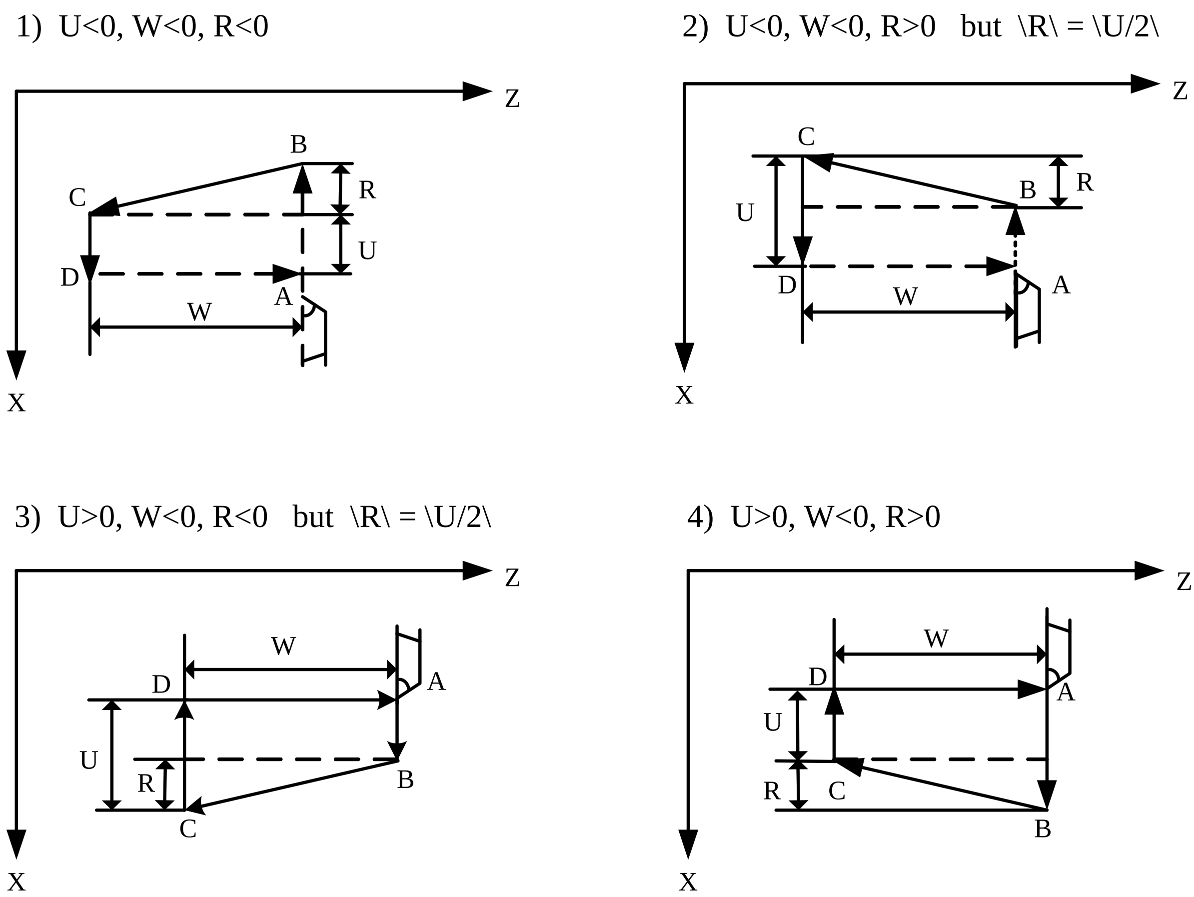


Fig.3-3-1-3　　G90 path

Example: using 90 compiles a program which path is shown in Fig. **3-3-1-4**.

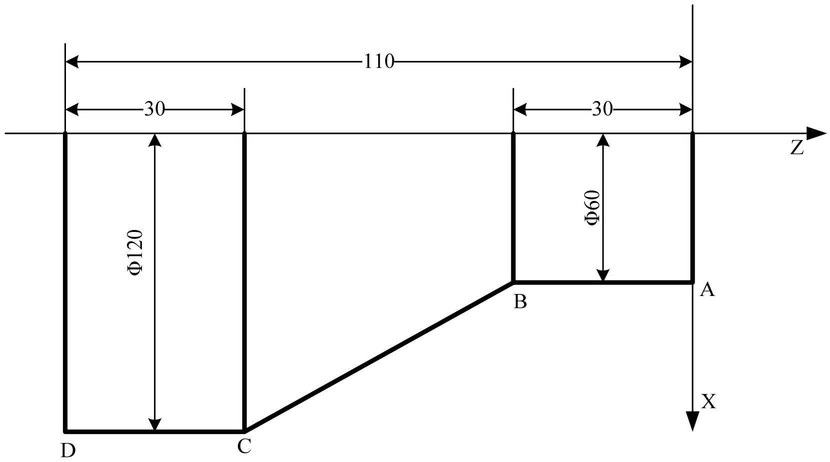


Fig. 3-3-1-4

A program：

O0001；

M3 S300；

G0 X130 Z5；

G90 X120 Z-110 F200；（C→D）

X60 Z-30；（A→B）

G0 X130 Z-30；

G90 X120 Z-80 R-30 F150；（B→C）

M5 S0；

M30；

3.3.2 End Cutting Cycle（G94）

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

Function：Executing G94 can perform a single cycle machining of end, the tool returns to the starting point after the cycle is completed.

R means the rapid traverse in Fig.3-3-2-1. Fig. 3-3-2-2，F means the cutting feed. The sign of numerical value following U in incremental programming is determined by X direction of path 2, that following W is done by Z direction of path 1.

Explanation: X, Z：absolute coordinate values of circular end point，unit：mm；

U, W：coordinates of circular end point corresponding to circular starting point, unit：mm；

R: Z coordinate component from starting point of end cutting to end point displacement，unit：mm；

F：it is combined federate of X, Z, it is modal.

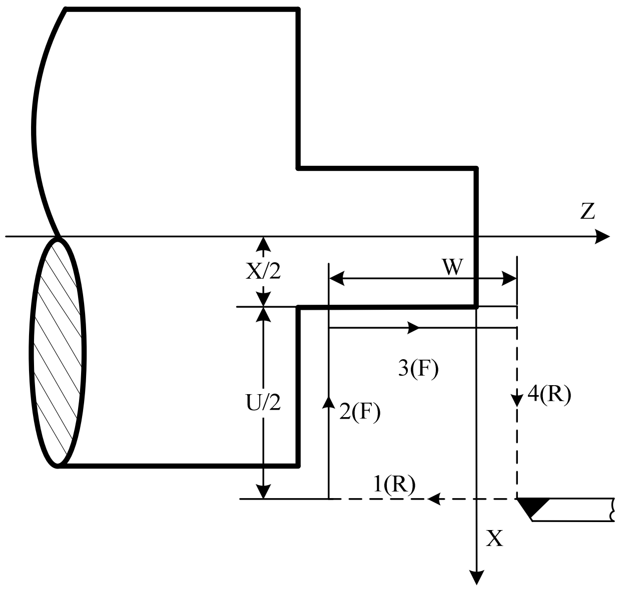


Fig. 3-3-2-1

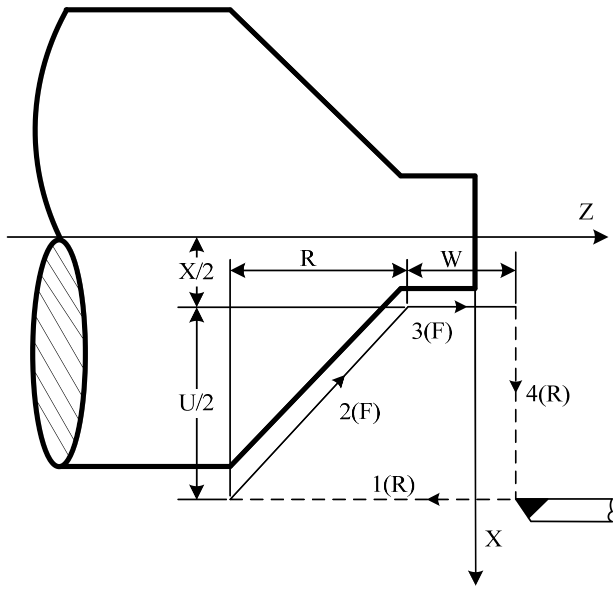
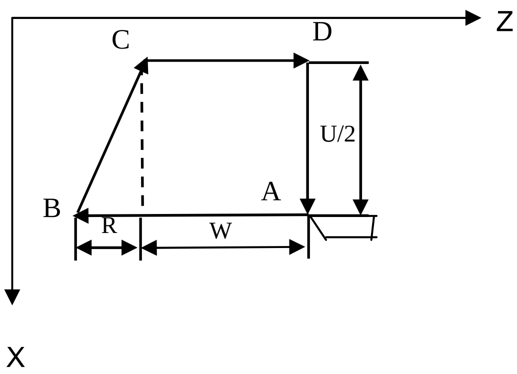


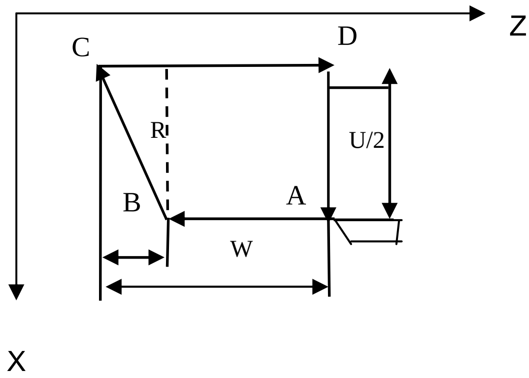
Fig. 3-3-2-2

G94 has four kind of paths according to start-up tool position, which are shown in Fig. 3-3-2-3:

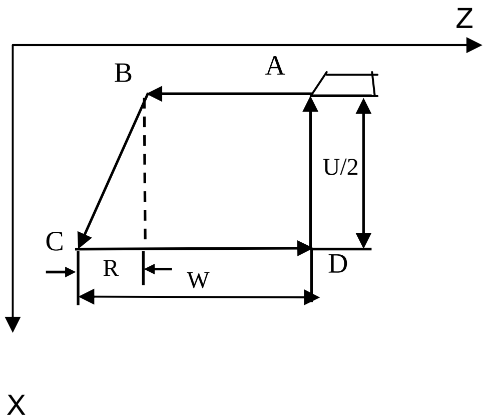
1) U<0，W<0，R<0



2) U<0，W<0，R>0(│R│≤│W│)



3) U>0，W<0，R>0 (│R│≤│W│)



4) U>0，W<0，R<0

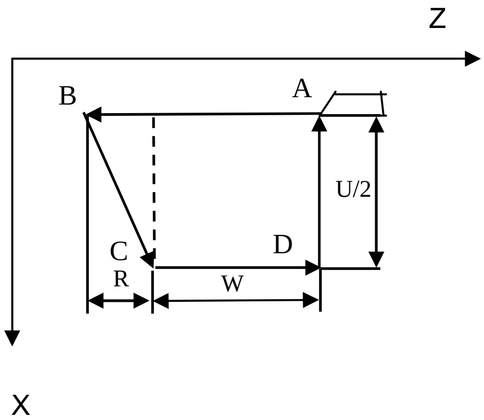


Fig.3-3-2-3 G94 path

Example: using G94 compiles a program shown in Fig. 3-3-1-4.

A program：

```
O0002；
M3 S1；
G0 X130 Z5；
G94 X120 Z-110 F100；（D→C）
G0 X120 Z0；
G94 X60 Z-30 R-50；（C→B→A）
M5 S0；
M30；
```

3.3.3 Thread Cutting Cycle（G92）

Command format：

G92X(U)__ Z(W)__J__K__ F__ L__ P__； （metric thread）

Assign thread pitch(F)

G92X(U)__ Z(W)__ J__K__ I __ L__ P__；（inch thread）

Assign thread pitch (tooth/inch)

Function： Executing G92 can machine the straight thread with constant pitch and the single cycle thread machining of taper thread, the tool returns to the starting point after the cycle is completed. The grooving is not demanded when thread cutting. When the user does not use J, K to set length of thread run-out, the thread run-out length=set value set by P68 X0.1X pitch. In incremental programming, the sign of numerical value following U is determined by X direction of path 1, that following W is done by Z direction of path 2. Dotted lines in Fig. 3-3-3-1 and Fig. 3-3-3-2 means rapid traverse, real lines means cutting feed. When J, K sets values, G92 executes X, Z thread run-out according to J, K set value; when only J or K value is set, 45° thread run-out is executed.

Explanation： X.Z：coordinats of circular end point, unit：mm；

U.W：coordinates of circular end point corresponding to circular starting point，unit：mm；

J：proportion of X thread run-out, without sign. Unit：mm，J is the radius designation；

K：proportion of Z thread run-out，without sign. Unit：mm；

R：radius difference between thread starting point and end point，unit：mm；

F：metric thread pitch，range: 0.001～500，unit：mm，modal；

I： inch thread/inch tooth，range: 0.06～25400，unit：tooth/inch，modal；

L：number of thread head, range:（1～99），unit：head，modal；it is 1 when it is not assigned；

P：length of thread run-out，range:（0～255），unit：0.1 pitch；

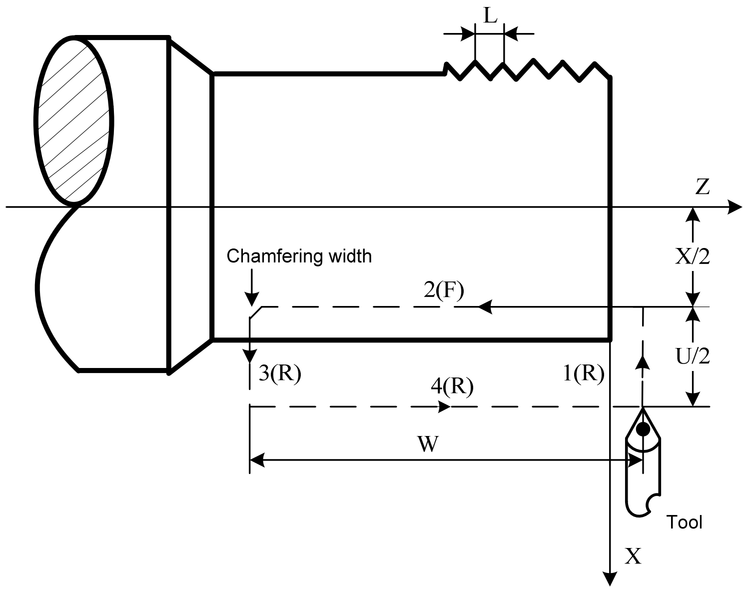


Fig. 3-3-3-1

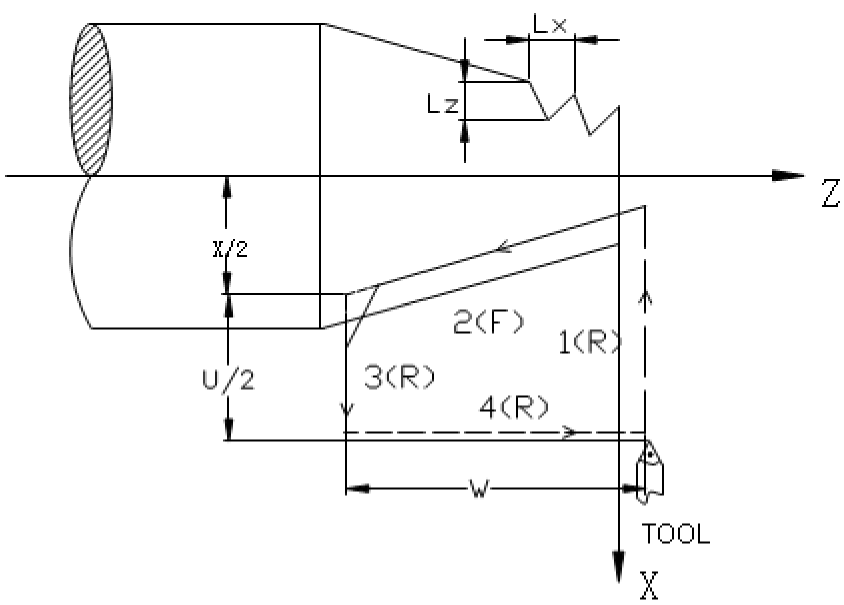


Fig. 3-3-3-2

Notes:

1. **Notes of G92 are the same those of G32, refer to Section 3.2.14.**
2. **When there is the feed hold signal (pause) input in thread cutting cycle in the thread cutting cycle, the cycle is executed till the operations of 3 completes to stop.**
3. **Thread pitch range, spindle speed limit are the same those of G32.**
4. **When using G92 machines the straight thread, start-up tool point and thread end point are the same in X direction, the system alarms because the system cannot identify the inner thread or outer.**
5. **R value in G92 is referred to Fig. 3-3-1-3.**
6. **When one of J, K is set to 0，or is not assigned，the system takes it as 45° thread run-out.**
7. **I value can be input in MDI mode and cannot be displayed. Do not execute G92 in MDI mode.**

Example: using G90 compiles a program shown in Fig. 3-3-3-3, and using G92 executes machining thread.

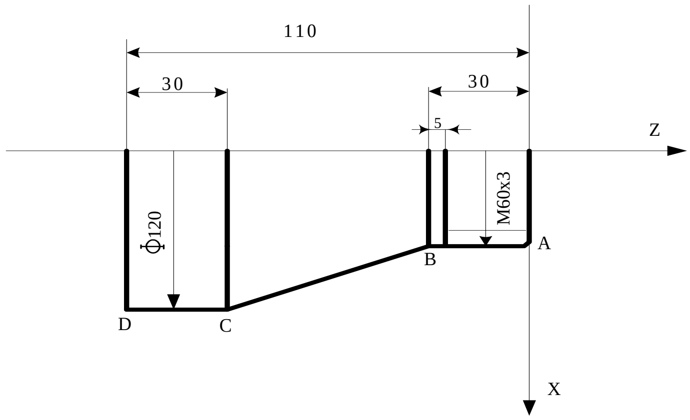


Fig. 3-3-3-3

A program：

```
O0001；
M3 S300；
G0 X150 Z50；
T0101；                    （outer turning）
G0 X130 Z5；
G90 X120 Z-110 F200；      （C→D）
X60 Z-30； （A→B）
G0 X130 Z-30；
G90 X120 Z-80 R-30 F150；  （B→C）
G0 X150 Z150；
T0202；                    （thread cutter）
G0 X65 Z5；
G92 X58.5 Z-25 F3；        （machine thread，4 times cut-in）
X57.5 Z-25；
X56.5 Z-25；
X56 Z-25；
M5 S0；
M30；
```

3.3.4 Notes in Single Fixed Cycle Commands

1） In the single fixed cycle, X（U），Z（W），R are modal. When the new X（U），Z（W），R are assigned，the previously commanded data are valid.

2） In the single fixed cycle, X（U），Z（W），R are cleared when the non modal G codes except G04 or other commands except for G90, G92, or G94 in Group 01 are commanded.

3） When there is only blocks without movement codes after G90, G92, G94, the fixed cycle is not executed repetitively.

（Example） N003 M3；

…

…

N010 G90 X20.0 Z10.0 F2000；

N011 M8；（do not repetitively execute G90）

…

…

4） In the fixed cycle, when M, S, T are commanded, the fixed cycle and M, S, T are executed simultaneously. Cancel the fixed cycle after M, S, T is executed, execute it again.

（Example） N003 T0101；

…

N010 G90 X20.0 Z10.0 F2000；

N011 G00 T0202；

N012 G90 X20.5 Z10.0；

3.4 Compound Fixed Cycle Commands

To simply programming, the system provides six compound cycle commands, including outer (inner) roughing cycle G71; end roughing G72, closed cutting cycle G73/ finishing machining G70; end deep hole machining cycle G74; outer grooving cycle G75 and compound thread cutting cycle G76. The system only specifies tool cutting values of finishing and roughing, and automatically counts machining paths and number of cutting.

3.4.1 Outer（Inner）Roughing Cycle（G71）

Command format： G71u（Δd）R（e）；

G71 P（NS） Q（NF） U（Δu） W（Δw）F S T ；

N（NS） . . . . . ；
. . . . ；
. . . . F；
. . . . S；   blocks for finishing path
. . . . T；
.
N（NF）. . . . . ；

Function： According to the workpiece's finishing path, cutting amount, tool infeed and tool retraction specified by the block, the system automatically count the roughing path, which is shown in Fig.3-4-1-1. Cutting is executed by operations which are parallel to Z. The non forming rod can be formed one time.

Explanation： Δd：cutting depth without sign at one time. The cut-in direction is determined by AA' (specified by the radius), range: 0.001mm～9999.999mm. It is modal and valid before the next is specified. It can be specified by P71, the parameter value also changes according to the program command.

e： Tool retraction（specified by the radius），unit：mm，range: 0.001mm～9999.999mm. It is modal and valid before the next is specified. It can be specified by P72, the parameter value also changes according to the program command.

NS：It is the first block's serial number in blocks for finishing path.

NF：It is the last block's serial number in blocks for finishing path.

Δu：Distance and direction of X finishing allowance, range: -9999.999mm～9999.999mm.

Δw：Distance and direction of Z finishing allowance, range: -9999.999mm～9999.999mm.

F： Cutting feedrate, range: feed per minute is 1mm/min～8000mm/min, feed per rev is 0.001mm/r～500mm/r.

S： Spindle speed；

T： Tool, tool offset number;

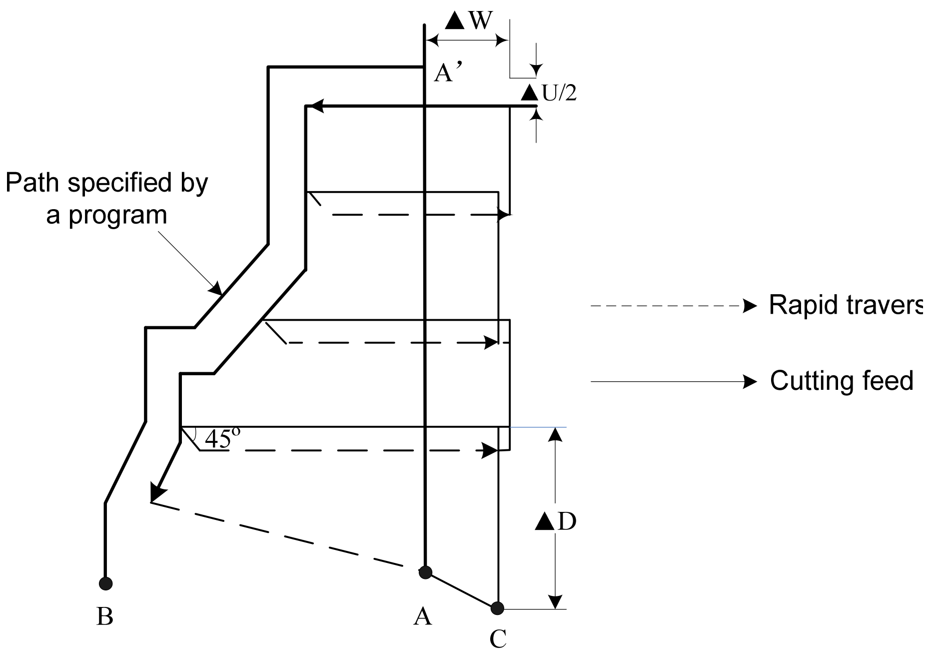


Fig. 3-4-1-1 G71 path

1. △d，△u are specified by the same address U, and their difference is whether their blocks specify P, Q.
2. The circular operations are executed G71 specified by P, Q.
3. In G71，F, S, T function in the blocks NS～NF are invalid and all are neglected. G71's blocks or the previous specified F, S, T are valid. F, S, T in the NS～NF are valid only to G70.
4. With the constant surface speed control selection function, G96 or G97 in the blocks NS～NF is invalid, G71 or the previous blocks are valid.
5. According to the cut-in directions, G71 has four paths (Fig.3-4-1-2), which are executed according to operations which are parallel, signs of Δu.Δw are shown below:

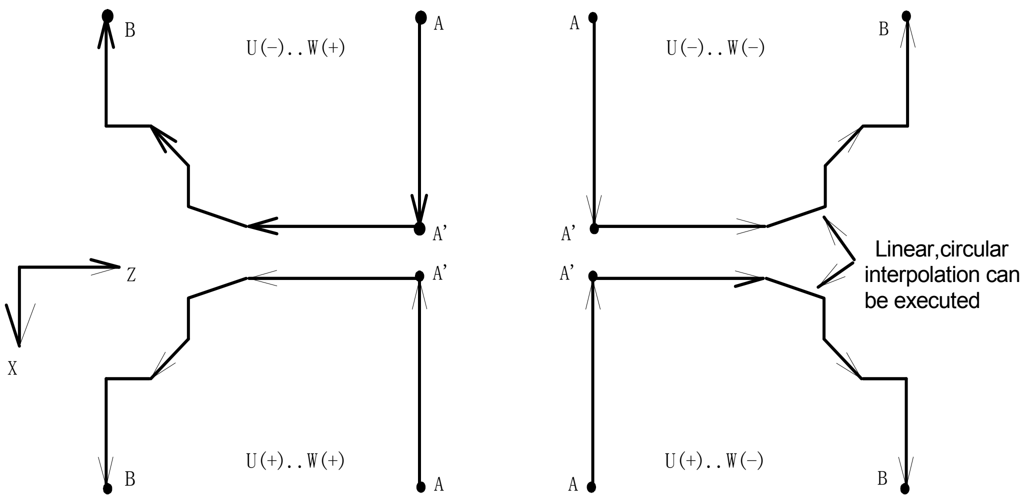


Fig. 3-4-1-2

6. Only G00 or G01 is specified in the blocks which serial number is NS between A and A', and coordinates of A and A' are consistent.
7. Between A' and B，X.Z values must monotonously increase or decrease, but their path cannot be monotonous.
8. The system cannot call subprograms between NS and NF.
9. There are most 100 blocks between NS and NF, ERR137 alarm occurs when the blocks exceed 100.

Example: Using G71 compiles a program shown in Fig.3-4-1-3.

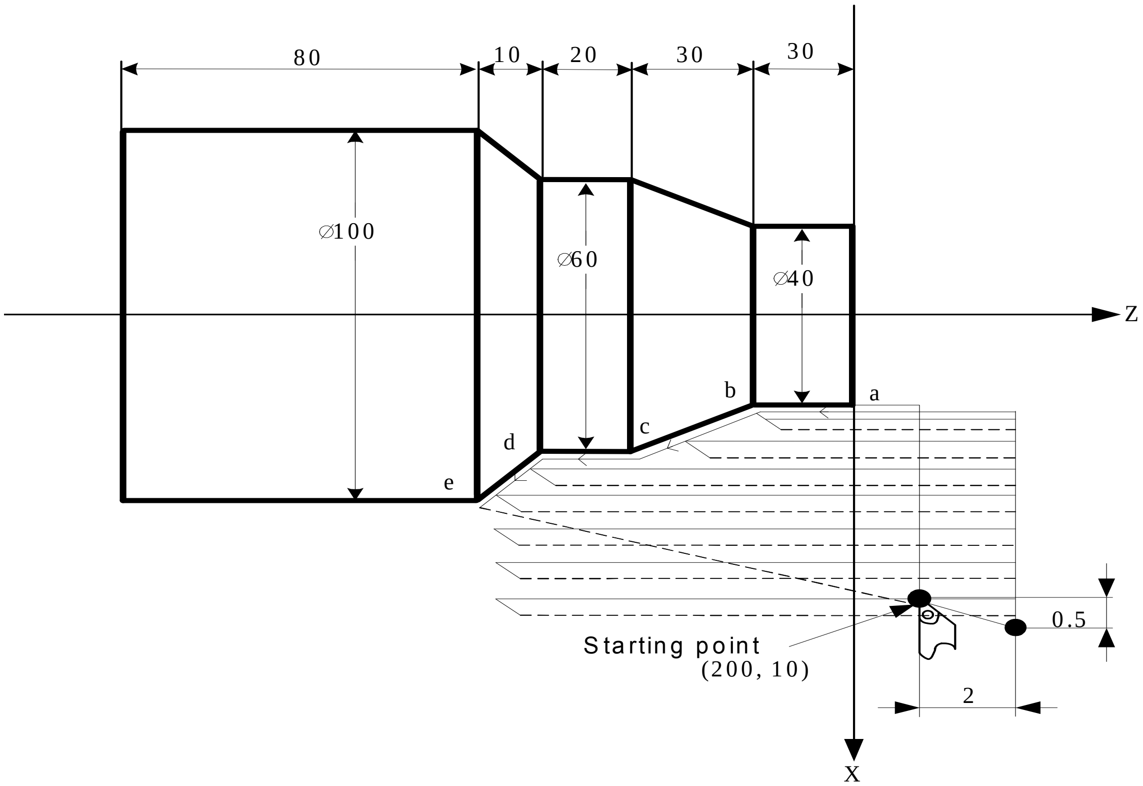


Fig. 3-4-1-3

A porgram：

O0001；

N010 G0 X220.0 Z50；	（positioning to safety position）
N020 M3 S300；	（the spindle rotating CCW, speed 300r/min）
N030 M8；	（cooling ON）
N040 T0101；	（use a roughing tool ）
N050 G00 X200.0 Z10.0；	（rapid positioning，approach the workpiece）
N060 G71 U0.5 R0.5；	（cut-in 1mm (diameter) at one time，tool retraction 1mm[diameter]）

N070 G71 P080 Q120 U1 W2.0 F100 S200； （a—d roughing，X allowance 1mm，Z 2mm）
N080 G00 X40.0； （positioning to X40）
N090 G01 Z-30.0 F100 S200；（a→b）
N100 X60.0 W-30.0；（b→c） blocks for finishing path a→b→c→d→e
N110 W-20.0；（c→d）
N120 X100.0 W-10.0；（d→e）
N130 G00 X220.0 Z50.0； (execute rapid tool retraction to safety position）
N140 T0202； (use No. 2 finishing tool, execute No 2. tool offset）
N150 G00 X200.0 Z10.0； （positioning to the cycle starting point of G7）
N160 G70 P80 Q120； (finishing a--- e）
N170 M05 S0； (stop the spindle, speed）
N180 M09； （cooling OFF）
N190 G00 X220.0 Z50.0 T0100； (rapid return to safety position, use reference tool, clear tool offset）
N200 M30； （end of program）

3.4.2 End Roughing Cycle（G72）

Command format：G72 W（Δd）R （e）；

G72 P（NS） Q（NF）U（Δu）W（Δw）F__ S__ T__；

N（NS） . . . . . . ；
. . . . ；
. . . . F；
. . . . S； blocks for finishing path
. . . . T；
.
N（NF）. . . . . . ；

Function： According to the workpiece's finishing path, cutting amount, tool infeed and tool retraction specified by the block, the system automatically count the roughing path Cutting is executed by operations which are parallel to X. The non forming rod can be formed one time.

Explanation：

Δd：cutting depth without sign at one time. The cut-in direction is determined by AB (specified by the radius), range: 0.001mm～9999.999mm. It is modal and valid before the next is specified. It can be specified by P71, the parameter value also changes according to the program command

e： Tool retraction，unit：mm，range: 0.001mm～9999.999mm. It is modal and valid before the next is specified. It can be specified by P72, the parameter value also changes according to the program command.

NS：It is the first block's serial number in blocks for finishing path.

NF：It is the last block's serial number in blocks for finishing path.

Δu：Distance and direction of X finishing allowance，range: -9999.999mm～9999.999mm.

Δw: Distance and direction of Z finishing allowance，range: -9999.999mm～9999.999mm.

F：Cutting feedrate，range：feed/min : 1mm/min～8000mm/min，feed/r: 0.001mm/r～500mm/r.

S：Spindle speed；

T：Tool, tool offset number；

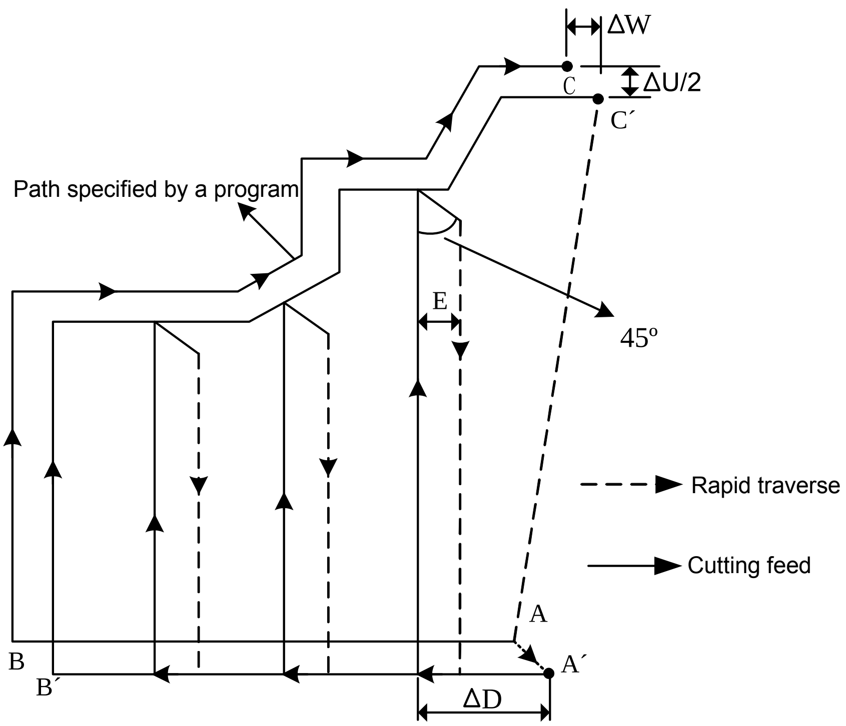


Fig. 3-4-2-1

Explanations：

1. △d，△u are specified by the same address W, and their difference is whether their blocks specify by P, Q.
2. The circular operations are executed G72 specified by P, Q.
3. In G72，F, S, T function in the blocks NS～NF are invalid and all are neglected. G72's blocks or the previous specified F, S, T are valid. F, S, T in the NS～NF are valid only to G70.
4. With the constant surface speed control selection function, G96 or G97 in the blocks NS～NF is invalid, G72 or the previous blocks are valid.
5. According to the cut-in directions, G72 has four paths (Fig.3-4-2-2), which are executed

according to operations which are parallel, signs of Δu, Δw are shown Fig.3-4-2-2.

6. Only G00 or G01 is specified in the blocks which serial number is NS between A and B, and coordinates of A and B are consistent.
7. Between B and C，X.Z values must monotonously increase or decrease, but their path cannot be monotonous.
8. The system cannot call subprograms between NS and NF.
9. There are most 100 blocks between NS and NF, ERR137 alarm occurs when the blocks exceed 100.

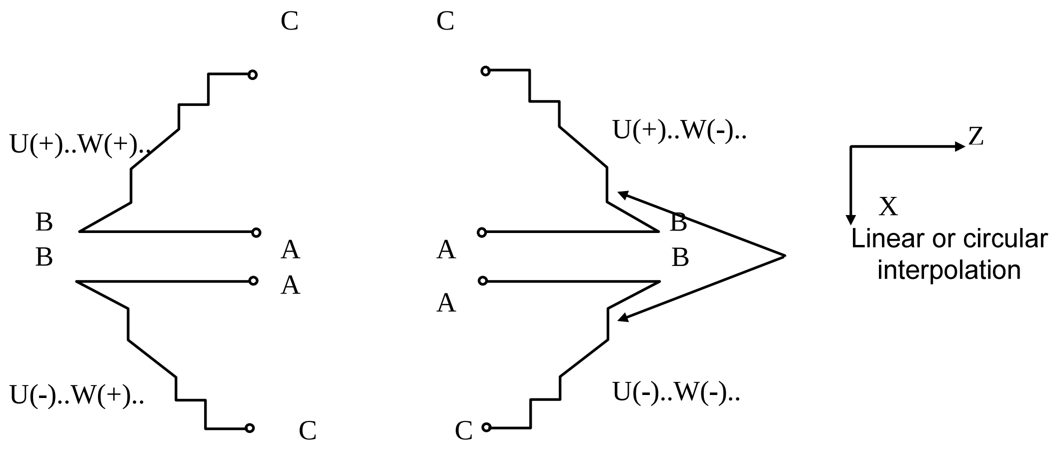


Fig. 3-4-2-2 four shapes of G72 path

Example: using compound fixed cycle G72 compiles a program shown in Fig. 3-4-2-3.

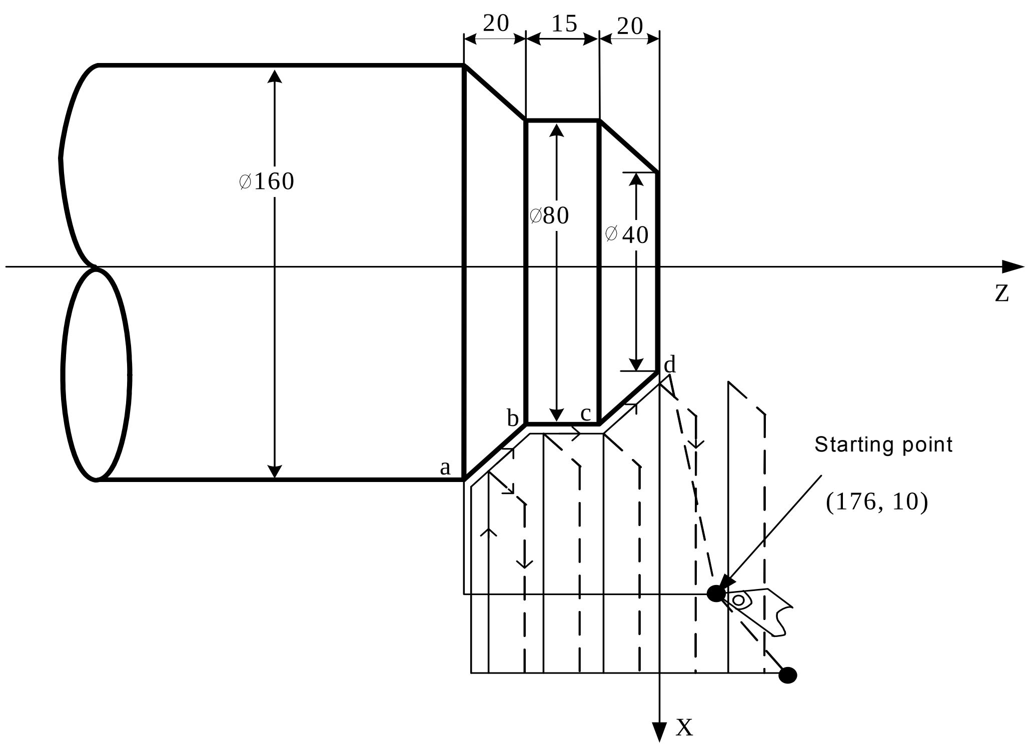


Fig. 3-4-2-3

Program：

O0002；

N010 G0 X220.0 Z50.0；（position to safety position）

N015 T0202； （execute No. 2 tool，No. 2 tool offset）

N017 M03 S200； （the spindle rotating CCW，speed 200）

N020 G00 X176.0 Z10.0；（rapid positioning，approach the workpiece）

N030 G72 W2.0 R1.0； （tool infeed 2mm，tool retraction 1mm）

N040 G72 P050 Q090 U1.0 W1.0 F100 S200； （a—d roughing，X allowance 1mm，Z allowance 1mm）

N050 G00 Z-55.0 S200 ；（rapid positioning）

N060 G01 X160.0 F120； （tool infeed to point a）

N070 X80.0 W20.0； （machining a—b） blocks for finishing path

N080 W15.0； （machining b—c）

N090 X40.0 W20.0 ； （machining c—d）

N100 G0 X220.0 Z50.0； （rapid tool retraction to safety position）

N105 T0303； （execute No. 3 tool, No. 3 tool offset）

N108 G00 X176.0 Z10.0； （rapid return G70）

N110 G70 P050 Q090； （finishing a—d）

N115 G0 X220.0 Z50.0； (traverse to safety position to execute tool change)

N120 M5 S0 T0200； （stop the spindle, execute No. 2 tool and cancel tool compentation）

N130 G0 X220.0 Z50.0； （rapid return to starting point）

N140 M30； （end of program）

3.4.3 Closed Cutting Cycle（G73）

Command format： G73u （Δi）W（Δk） R （d）;

G73 P（NS）Q（NF）U（Δu）W（Δw）F S T ；

Function： G73 can execute the repetitive cutting along the path specified by the blocks NS～NF, the tool moves forward once every cutting. For the smithing, casting and other roughing has formed prelimiarily, which can improve the machining efficiency.

Explanation： Δi：distance and direction of X tool retraction (radius value), unit：mm; it is modal and valid before the next is specified. It can be specified by P73, the parameter value also changes according to the program command.

Δk：Distance and direction of Z tool retraction (radius value), unit：mm; it is modal and valid before the next is specified. It can be specified by P74, the parameter value also changes according to the program command.

D: Times of closed cutting, unit: times; it is modal and valid before the next is specified. The parameter value also changes according to the program command, when set by P75.

NS: It is the first block's serial number in blocks for finishing path;

NF: It is the last block's serial number in blocks for finishing path;

Δu: Distance and direction of X finishing allowance, range: -9999.999mm～9999.999mm. the specified value is the radius or diameter value according to N0:1#3;

Δw: Z finishing allowance, range: -9999.999mm～9999.999mm;

F: Cutting feedrate, range: 1 mm/min～8000mm/min;

S: Spindle speed;

T: Tool, tool offset number;

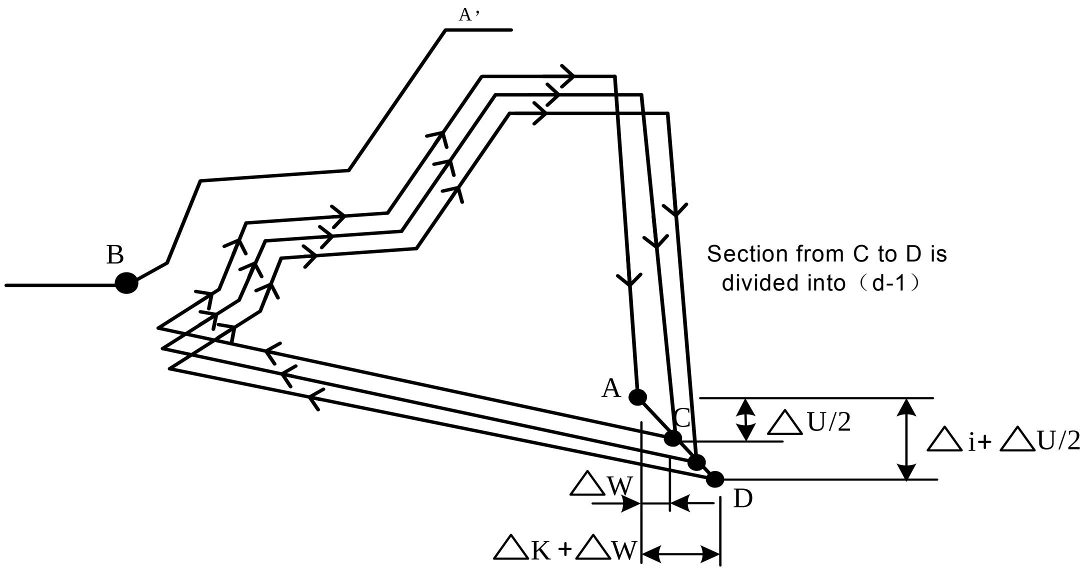


Fig. 3-4-3-1 G73 path

1. F, S, T functions in any blocks between NS～NF are invalid. F, S, T functions only in G73 are valid.
2. Δi, Δk, Δu, Δw are specified by the same address U, W, and their difference is whether their blocks specify by P, Q.
3. The system cannot call subprograms between NS and NF in G73.
4. NS～NF blocks realize the cycle machining, signs of Δu.Δw.Δi.Δk are paid more attention. The tool returns to A after the cycle is completed.
5. When one of Δi, Δk is 0, U0 or W0 is compiled in the course of programming; or P73/P74 are set 0, otherwise, it is influenced by last G73's set value.
6. There are most 100 blocks between NS and NF, ERR137 alarm occurs when the blocks exceed 100.

Example Using 73 compiles programs shown in Fig.3-4-3-2.

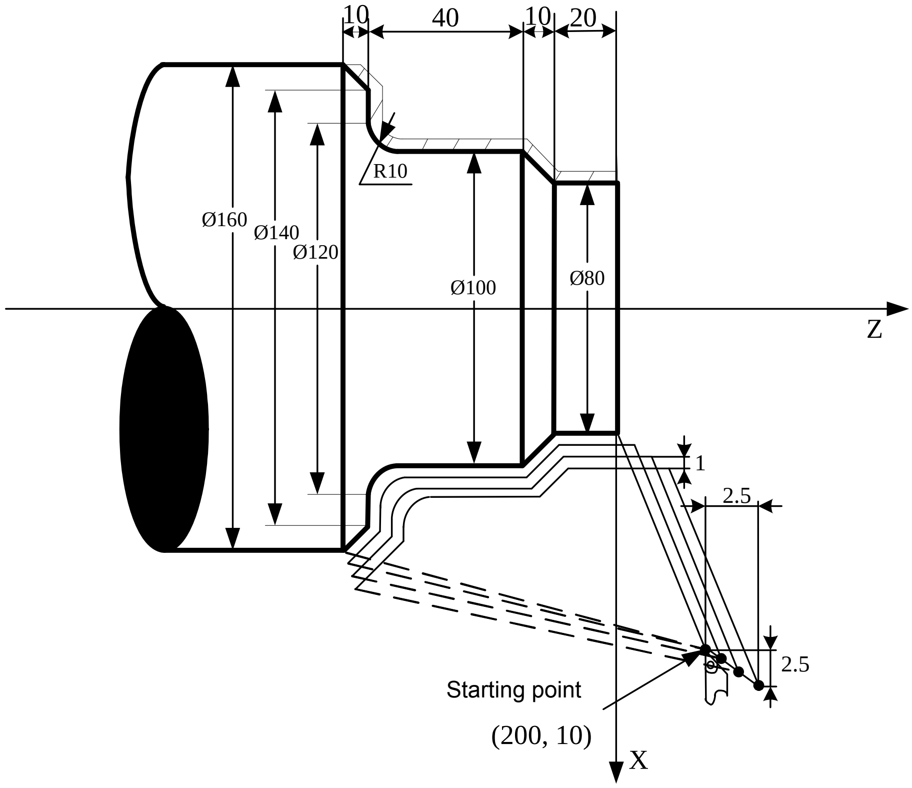


Fig. 3-4-3-2 G73 sample

A program is shown below:（the diameter is designed，metric input, ，least blank dimension is Ø86）

N008 G0 X260.0 Z50.0 ; （positioning to safety position）

N009 T0101; （execute No. 1 tool, its tool offset）

N010 G98 M03 S300; （the spindle rotating CW，speed 300）

N011 G00 X200.0 Z10.0; （rapid position to starting point）

N012 G73 U2.0 W2.0 R3 ; （X tool retraction 4mm，Z tool retraction 2mm，roughing 3 times，diameter feeding 2mm at every tool infeed）

N013 G73 P014 Q020 U0.5 W0.5 F100 ;（X allowance 0.5mm，Z finishing allowance 0.5mm）

N014 G00 X80.0 W-10.0 S500 ;
N015 G01 W-20.0 F120 ;
N016 X100.0 W-10.0 ;
N017 W-30.0 ; blocks for finishing shape
N018 G02 X120 W-10.0 R10.0 F100 ;
N019 G01 X140.0 ;

N020 G01 X160.0 W-10.0 ;

N021 G0 X260.0 Z50.0；(traverse to safety position to execute tool change)

N022 T0303；（execute No. 3 tool, its tool offset）

N023 G00 X200.0 Z10.0；（rapid return to G70's positioning）

N024 G70 P014 Q020；（finishing）

N025 M5 S0 T0200；（stop the spindle，execute No. 2 tool and cancel its tool compensation）

N026 G0 X260.0 Z50.0；（rapid return to starting point）

N027 M30；（end of program）

3.4.4 Finishing Cycle（G70）

Command format：G70 P（NS）Q（NF）；

Function：when G70 is executed, the tool executes the finishing along the workpiece finishing path specified by NS～NF from the initial point.

Using G70 executes finishing after G71, G72, G73 executes roughing.

Explanation：NS: It is the first block's serial number in blocks for finishing path;

NF: It is the last block's serial number in blocks for finishing path.

G70 path is determined by NS～NF blocks. Relative position of NS, NF in G70～G73 is shown below:

```
........
........
G71/G72/G73 P（NS） Q（NF） U（Δu） W（Δw） F   S   T  ;
N（NS） ......
  ........
· F
· S
· T
.
.
.
N（NF）......
           .
G70 P（NS） Q（NF）;
           .
```

1. F, S, T function among "NS" and "NF" in G71 G72 G73 are invalid, but they are valid in G70.

2. The tool returns to starting point and the system reads the next block when G70 completes.
3. G70 includes S, T, F. In M code, M30 is valid when M30 and G70 are in the same block, and it is invalid when it and other commands are in the same block.
4. Blocks between NS and NF in G70 cannot call subprograms.

   Example：See G71.G72 sample.
5. The address specified by P.Q should be single and cannot be repetitively commanded. When a program defines the same serial number, the system runs the block for finishing neighboring G70.

3.4.5 End Deep Hole Machining Cycle（G74）

Command format： G74 R（e）;

G74 X（U）Z（W）P（Δi）Q（Δk）R（Δd）F;

Function： When G75 is executed, the system confirms the cutting end point according to blocks (the point confirmed by X, Z coordinates) e.Δi.Δk and Δd confirms the tool path. In the cycle, it can execute the chip breaking of end cutting. When X (U) are omitted, only Z is executed, which is the deep hole machining. G74 path is shown in Fig. 3-4-5-1.

Explanation： e：Tool retraction amount after cutting Δk along Z, unit：mm，range: 0.001mm～9999.999mm. It is modal and valid before the next is specified. It can be specified by P76, the parameter value also changes according to the program command.

X： X absolute coordinate value of cutting end point B2, unit：mm；

U： Difference of X absolute coordinate between cutting end point B2 and starting point A, unit：mm；

Z： Z absolute coordinate value of cutting end point B2, unit：mm；

W： Difference of Z absolute coordinate between cutting end point B2 and starting point A, unit：mm；

Δi： X circular movement at one time（without sign, radius value）, unit：mm；

Δk： Z cutting movement at one time（without sign），unit：mm；

Δd： X tool retraction when cutting to end point（radius value），unit：mm；

F： Cutting feedrate. Range: feed per minute is 1mm/min～8000mm/min，feed per rev is 0.001mm/r～500mm/r”.

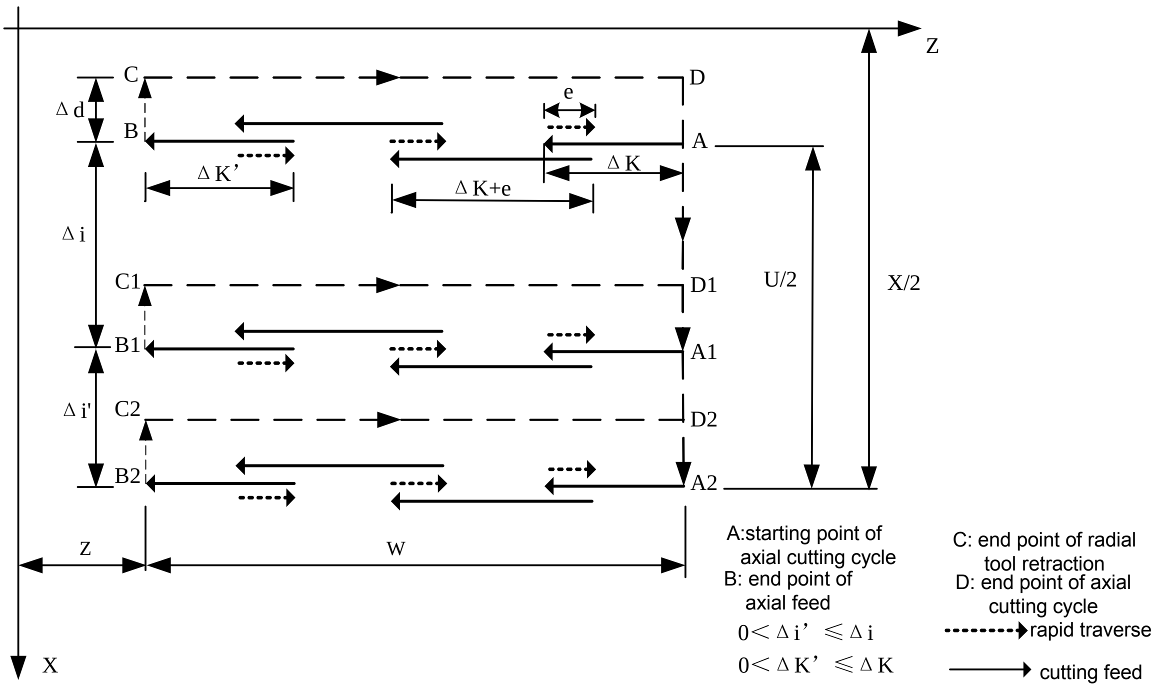


Fig. 3-4-5-1

1. e and Δd are specified by the address R, and their difference are whether they are specified by Z（W）, namely, when X（U）is commanded, it is Δd，when X（U）is commanded, it is e;
2. When the cycle operation include G74 of Z（W）and Q（△k）, "G74 R（e）" is executed, the cycle operation cannot be executed.

Example: Using G74 compiles a program shown Fig. 3-4-5-2.

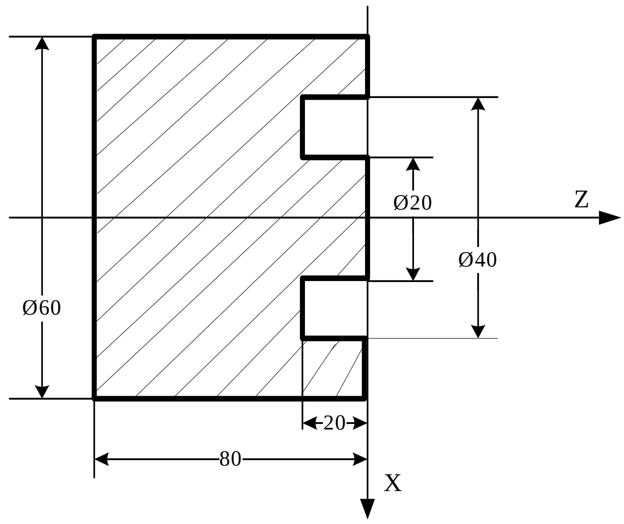


Fig. 3-4-5-2

A program：

O0001；　　　　（program name）

G0 X100 Z50；　　（rapid positioning ）

T0101；　　　　　　（tool width 2mm）

M3 S500 G97；　　　（start the spindle，its speed 500）

G0 X36 Z5；　　　　（positioning to starting point of machining, tool width having been added to X value）

G74 R1 ；　　　　　（Z tool retraction）

G74 X20 Z-20 P2 Q3.5 F50；（X cycle movement 4mm at one time 4mm，Z 3.5mm）

G0 Z50；　　　　　　（Z tool retraction）

X100；　　　　　　　（X tool retraction）

M5 S0；　　　　　　（stop the spindle）

M30；　　　　　　　（end of program）

3.4.6　Outer Grooving Cycle（G75）

Command format：G75 R（e）；

G75 X（U）Z（W）P（Δi）Q（Δk）R（Δd）F ；

Function：When G75 is executed, the system confirms the cutting end point according to blocks (the point confirmed by X, Z coordinates) e.Δi.Δk and Δd confirms the tool path. It is similar that X, Z are exchanged in G74, and in the cycle, it can execute the chip breaking of end cutting, execute the grooving and cutting-down machining of outer diameter(omit Z, W, Q). G75 path is shown in Fig. 3-4-6-1.

Explanation：e：Tool retraction amount after cutting Δi along X, unit：mm，range: 0～9999.999mm. It is modal and valid before the next is specified. It can be specified by P76, the parameter value also changes according to the program command;

X：X absolute coordinate value of cutting end point B2，unit：mm；

U：Difference value of X absolute coordinate between cutting end point B2 and A，unit：mm；

Z：Z absolute coordinate value of cutting end point B2，unit：mm；

W：Difference value of Z absolute coordinate between cutting end point B2 and A, unit：mm；

Δi：X circular movement at one time（without sign, radius value），unit：mm；

Δk：Z cutting amount at one time（without sign），unit：mm；

Δd：Z tool retraction amount when cutting to end point，unit：mm；

F ：Cutting feedrate.

G74, G75 are used to cutting, grooving or hole machining, make the tool automatically retract.

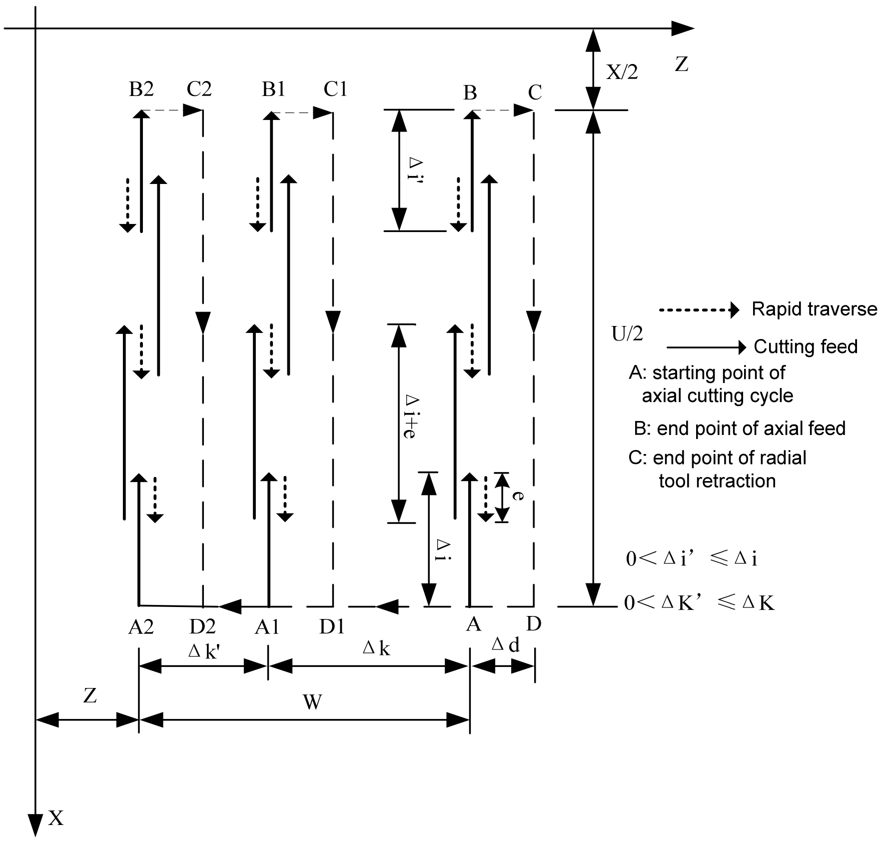


Fig. 3-4-6-1

1. e and Δd are specified by the address R, and their difference are whether they are specified by X（U），namely, when X（U）is commanded, it is Δd，when X（U）is commanded, it is e;
2. The circular operations is executed by G75 specified by X (U).

Example: Using G75 compiles a program shown in Fig. 3-4-6-2.

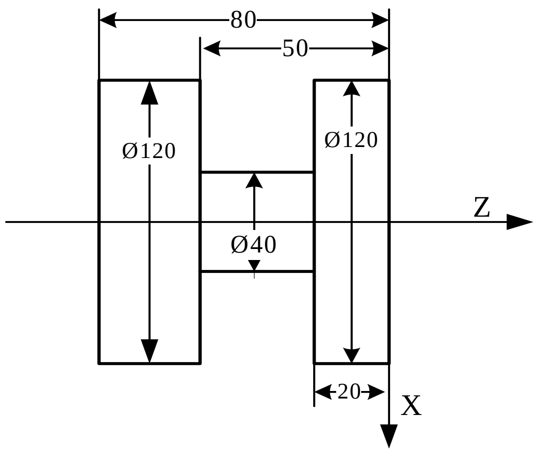


Fig. 3-4-6-2 G75 sample

A program：

O0001；（program name）

G0 X150 Z50；　　（rapid positioning）

T0101；　　（too width 4mm）

M3 S500 G97；　　（start the spindle，its speed 500）

G0 X125 Z-24；　　（positioning to starting point of machining, the tool width having been added to Z）

G75 R1 ；　　（X tool retraction）

G75 X40 Z-50 P2 Q3.5 F50；（X cycle movement 4mm at one time 4mm，Z 3.5mm）

G0 X150；　　（X tool retraction）

Z50；　　（Z tool retraction）

M5 S0；　　（stop the spindle）

M30；　　（end of program）

3.4.7　Compound Thread Cutting Cycle（G76）

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

G76 X（U） Z（W）　R（i）P（k）Q（△d）F（I） ；

Function：The system automatically counts and executes thread cutting cycle many times according to the commanded address, and G76 path is shown in Fig. 3-4-7-1.

Explanation：X.Z：　coordinates of thread end point (bottom of thread)，unit：mm；

U.W：coordinates of thread end point corresponding to machining starting point，unit：mm；

m：it is repetitive times of last finishing, modal, valid before it is specified next time. It is set by P77, the parameter value is also changed according to the commands. Range: 1～99；

r：　Chamfer value of thread. When L is the lead, in 0.1L～9.9L, 0.1L is taken as the first gear, using 00~99 two numerical values is specified. It is modal and valid before it is specified next time. Using P68 also change the parameter value according to the commands. After G76 set the thread chamfering value, it influences in G92 thread cutting cycle.

a：Tool nose angle（angle of thread tooth can select the angle 80°，60°，55°，30°，29°，0°）.

The original of angle value is specified by two numerical value. It is modal, and valid before it is specified next time. It is set by P78, and the parameter value is changed according to the command. The tool nose angle can select 80°.60°.55°.30°.29°.0°；

△dmin：least cut-in value, unit：mm. When the first cut-in value（$\triangle D\times\sqrt{N}$-$\triangle D\times\sqrt{N-1}$） is less than △dmin, △dmin is taken as the first cut-in value. It is modal, and valid before it is specified next time. It is also set by P79, the parameter value can be changed by the commands. The least cut-in value is 0～9999999，unit：0.001mm；

d：Finishing allowance, unit：mm. It is modal, and valid before it is specified next time. It is set by P80, the parameter value is changed by the command. The finishing allowance range is 0～9999999, unit：0.001mm；

i： Radius difference of thread part, unit：mm，i=0 : cutting the straight thread;

k：Tooth height of thread（X distance is commanded by the radius value）, unit： mm;

△d：The first cutting depth, radius value, unit：mm.

F：Thread lead, unit：mm；

I：Tooth/inch.

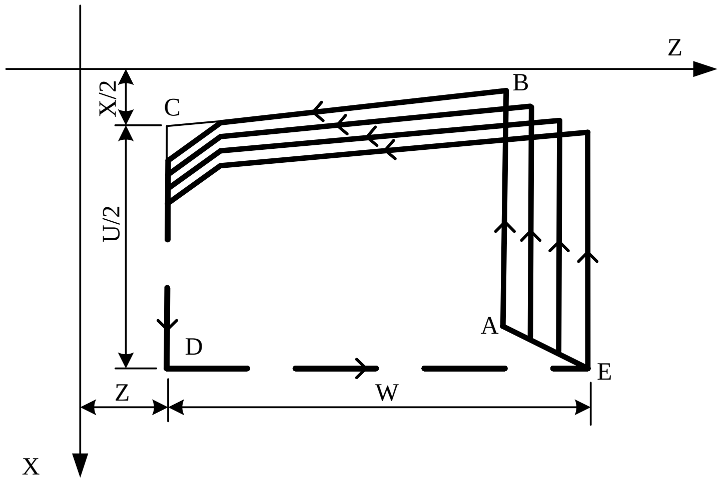


Fig. 3-4-7-1

Cut-in method is shown in Fig. 3-4-7-2:

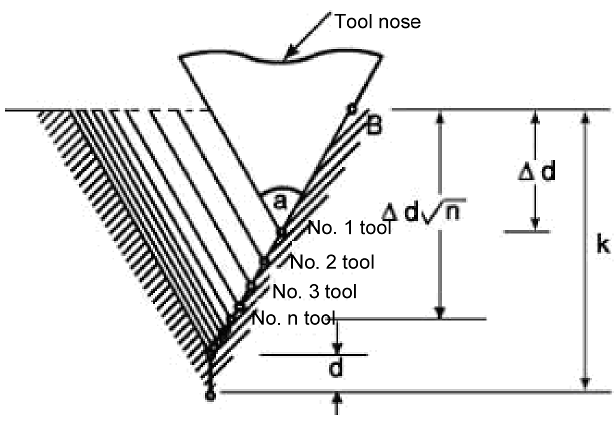


Fig. 3-4-7-2

1. Data specified by P, Q, R are distinguished by whether they have X (U), Z (W) or not.
2. The cycle operation is executed by G76 specified by the address X（U），Z（W）.
3. In cycle machining, the tool uses the one-side to machine, and the tool nose load can be reduced.
4. The first cut-in amount is △d，the N times cut-in is △D× $\sqrt{N}$ ，each cut-in amount is fixed.
5. There are four machining shapes, which can machine inside thread. B, C section uses the feedrate commanded by F, others are rapid traverse in Fig. 3-4-7-1.
   Increment signs are determined as follows:
   U：It is determined by the direction from the path A to C;
   W：It is determined by the direction from the path C to D;
   R（I）：It is determined by the direction from the path A to C;
   P（K）：positive；
   Q（△D）：positive.
6. Its notes are the same those of G32.
7. The thread chamfering value designation is valid to G92 thread cutting cycle.
8. m，r，a are specified by the address p one time.

Example: using G76 compile a program shown in Fig. 3-4-7-3, machining thread is M68X6.

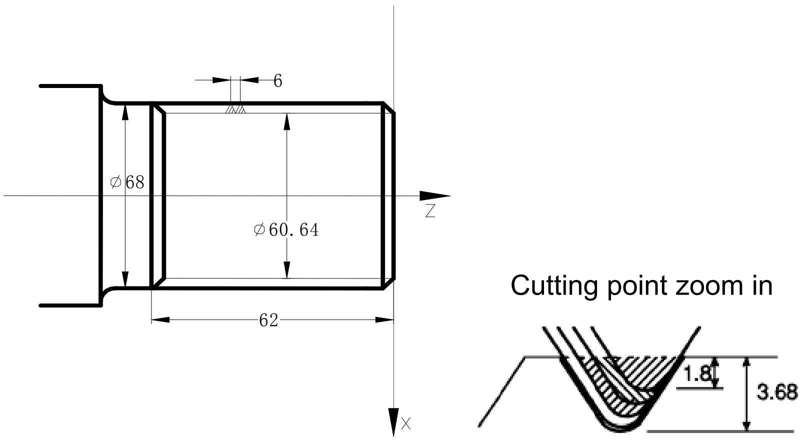


Fig. 3-4-7-3

A program：

G00 X100 Z50；	（positioning to safety position）
M03 S300；	（start the spindle, specify its speed）
G00 X80 Z10；	（rapid positioning to starting point of machining）
G76 P011060 Q0.1 R0.2；	（execute thread cutting）
G76 X60.64 Z-62 P3.68 Q1.8 F6.0；	
G00 X100 Z50；	（return to starting point of the program）
M5 S0；	（stop the spindle）
M30；	（end of program）

3.4.8 Notes of Compound Fixed Cycle Codes

1. There must be P，Q，X，Z，U，W，R in blocks of the commanded compound fixed cycle, each block must be commanded correctly.
2. In blocks of G71, G72, G73, when their serial numbers are specified by P, the blocks of corresponding serial number must command G00 or G01 in Group 01, otherwise, P/S alarm occurs.
3. In MDI mode, the system cannot execute G70，G71，G72，G73，G74，G75，G76. Namely, they cannot be executed even if they are commanded.
4. In G70，G71，G72，G73, the blocks which serial numbers are specified by P and Q cannot have the followinG Codes:
   - ★Codes in Group 01 except for G00, G01, G02, G03;
   - ★ M98/M99；
   - ★ G04 is valid in the last forming tool of roughing and valid in finishing.
5. When the compound fixed cycle （G70～G76）is executed, the operation can be stopped to insert the manual operation, but the system must return the previous operation before the compound fixed cycle is executed again. If not, the manual movement amount will not be added to the absolute value, the following operation will miss position, which value is equal to manual movement amount.
6. When G70，G71，G72，G73 is executed，the serial numbers specified by P，Q cannot be repetitive in the program.
7. Notes of G76 are same those of G32, G92, are valid for thread chamfering designation and G92.

Part Two Programming

CHAPTER FOUR MISCELLANEOUS FUNCTION M CODE

The movement command and M code must be executed simultaneously when they are in the same block.

(Example) N1 G01 X50.0 Z-50.0 M05 ；(spindle stop)

4.1 Miscellaneous Function （M function）

Miscellaneous function M code consisted of the address M and two-digit numerical value. The system sends the corresponding control signal to the machine, used to control the corresponding function ON/OFF of the machine. There must be only one M code in the same block.

The system supports the following M codes:

M03 ： Spindle rotation CW.

M04 ： Spindle rotation CCW.

M05 ： Spindle stop.

M08 ： Cooling ON.

M09 ： Cooling OFF（do not output signals）.

M10 ： Tailstock forward.

M11 ： Tailstock backward.

M12 ： Chuck clamping.

M13 ： Chuck releasing.

M32 ： Lubricating ON.

M33 ： Lubricating OFF（do not output signals）.

M00 ： Program pause; press 'Cycle Start' key to continuously execute programs.

M30 ： End of program; the program returns to the initial.

M41~M44 ：Automatic gear-change function of spindle（see Machine Connection Section, Automatic Gear-Change Control of Spindle）

M51~M70 ：User-customized output control

M91~M94 ：User-customized input control

Except for M00, M30, execution time (not pulse width) of other M codes is set by P86.

Set value：0～9999999

Set time =set value ×4 ms.

Part Two Programming

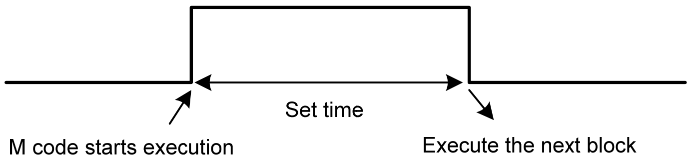


Fig. 4-1-1

Note 1: When other M codes except for the above, the system alarms the following and stops the execution.

NO.181: M code is wrong, illegal M codes are input into the program.

Note 2: After M, S, T, function is activated, the mode still remains even if it is changed, 'Reset' key is pressed to close the activation. (NO:12#0 sets whether M, S, T function is valid.)

The following M codes describe special usages:

1. M30 end of program

1) M30 means the end of main program.

2) It means the system stops the automatic run, is in reset state.

3) It means the system returns to the beginning of the main program.

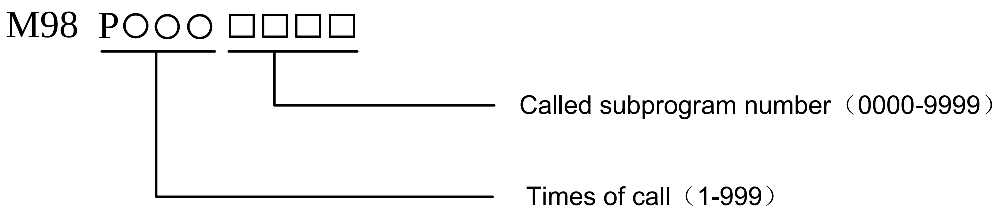
4) It means 1 is added to the number of machining workpiece.

2.M00：Program pause

After the block of M00 is executed, the system stops the automatic run. It is the same that of the single block stop, and the system saves the previous modal information. The system starts the automatic run after it runs.

3. M98/M99 (call subprogram/ subprogram return)

Format：



It can be execute repetitively when it is used for calling a subprogram or the end of program is M99. The four-embedding of subprogram can be executed and is referred to Section 4.7 Subprogram Control.

Note **1: The next block of M00, M30 cannot be saved into the buffer storage even if it exists.**

Note **2: The code signals are not output when M98, M99 is executed.**

4.2 Special M Codes（M21.M22.M23.M24 ）

Relevant Parameters:

0	1	3

		M23	**M21**				

M21 =0：The corresponding output pin does not execute the output

=1：The corresponding output pin executes the output

M23 =0：The corresponding output pin does not execute the output

=1：The corresponding output pin executes the output

Input/output signals:

M21I ：input signal（user-customized input pin） M21O：output signal（user-customized output pin）

M23I ：input signal（user-customized input pin） M23O：output signal（user-customized output pin）

Use method:

Function 1: M21 When M21=0, the corresponding output pin does not execute the output; when M21=1, it outputs M21O.

Example：G0 X100 Z100

M21 execute the normal M code

G0 X0 Z0

M22 close output M21O

M30

Function 2：M21 P when M21 is set 1 and after M21 P function is executed in the time specified by P, M21 output ends.

Example：G0 X100 Z100

M21 P10000 the duration of M21 execution is 10s, and M21O is closed.

G0 X0 Z0

M22

M30

Function 3: M21Q When M21 is set 1 and M21 Q function is executed, the corresponding output pin has the output signal and waits for the input signal, and there is an input signal, the system waits for the end to execute the next block; when there is no input signal, the system always waits.

Example：G0 X100 Z100

M21 Q1 the system checks input interface M21I，and waits always without it.

G0 X0 Z0

M22

M30

M22 close the output M21O.

Note 1: The time unit is ms when P is specified.

Note 2: When Q is specified, its numerical value is not 0, it is HIGH, otherwise, it is LOW.

Note 3: M23 use methods are the same those of M21; M24 of M22.

4.3 M Code Calling Subprograms

When NO.4#2(CM98) is set to 1 and the system executes others except for M, S, T, the system calls a corresponding subprogram instead of alarm. The user can extend miscellaneous function codes according to requirements combined with macro input/output interface variable.

M code: when the system executes others except for standard codes, the called subprograms are :

M□□：called subprogram **90**□□.

4.4 S Codes Calling Subprograms

When NO.4#2(CM98) is set to 1 and the system executes others except for M, S, T, the system calls a corresponding subprogram instead of alarm. The user can extend miscellaneous function codes according to requirements combined with macro input/output interface variable

S code: when the system executes **S10~S99**, the called subprograms are:

S□□：called subprogram **91**□□.

Note 1: S codes cannot call subprograms when the spindle analog voltage output is selected.

Note 2: When the system executes the non-standard M, S, the corresponding subprograms must be input otherwise, NO.78 alarm occurs.

Note 3: The non-standard M, S, T codes cannot run in MDI mode, otherwise, M/S or T code wrongly alarms or NO.130 alarm occurs.

Note 4: In the corresponding subprogram, the axis motion command can be input, or the output point can be controlled (ON/OFF), or the skip or cycle is executed, some DI signal is taken as an end signal of M/S/T. See Macro Program about DI/DO.

4.5 T Codes Calling Subprograms

When NO.4#2(CM98) is set to 1 and the system executes others except for M, S, T, the system calls a corresponding subprogram instead of alarm. The user can extend miscellaneous function codes according to requirements combined with macro input/output interface variable.

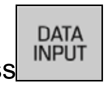
T code: when the system executes others except for standard codes, the called subprograms are :

T□□：called subprogram **92□□**.

For example: T28 calls the subprogram 9228.

4.6 Parameters of Miscellaneous Function

The parameters are set by the user according to the actual.

Setting: Open the parameter protection switch, select the parameter window in MDI mode, move cursor to the front of the required serial number to input data, press 

and then the data is input into the corresponding parameters.

Methods for searching a serial number are shown below:

1) Use Page Up/Down to find out the serial number window, move the cursor by the direction key to position to the required serial number position.

2) Search method: press 

→input the required searched parameter number →press 

.

4.7 Alarm Related to Miscellaneous Function

Alarms related to miscellaneous function are displayed on the external message window. When the system alarms, it automatically switches to external message window, and the detailed contents of alarm are displayed on the top of LCD, ‘ALARM’ flashed at the right of LCD.

181：M code error.

Illegal M codes are input into the program.

182：S code error.

Illegal S codes are input into the program.

183：T code error.

Illegal T codes are input into the program.

185：The tool change time is too long.

The system alarms when it has not received the specified cutter spacing arrival signal after the time of the tool post rotating forward for Ta .

186：The system has not received the tool post’s reverse locking signal within the tool post reverse locking time.

187 ：Tool change is not ready.

The system does not check whether the tool number is consistent with the current tool number.

044：Start the spindle when the chuck clamps.

045：The chuck releases when the spindle runs.

250：The tailstock advancing/retreating is executed Automatic run, starting the spindle, the spindle rotating.

CHAPTER FIVE SPINDLE FUNCTION S CODE

The code signal is sent to machine by the address S and its following numerical value, used to control the spindle of the machine. In a block, one S code can be commanded.

Refer to the manufacturer's user manual about digits and use of S code.

When the movement command and S code are in the same block, they are executed simultaneously.

5.1 Spindle Switching Value Control

NO:1#4=0: the address S and its following numerical value control the spindle's speed.

Command format： S__

The system provides 4-gear spindle mechanic gear change. Refer to the manufacturer's user manual about the corresponding relation between S code and the spindle speed, and the spindle speed gear of the machine (without selecting the analog spindle function).

Gear signal: S1～S4

Duration of S code is set by P87.

Set value：0～9999999

Set time = set value ×4 ms.

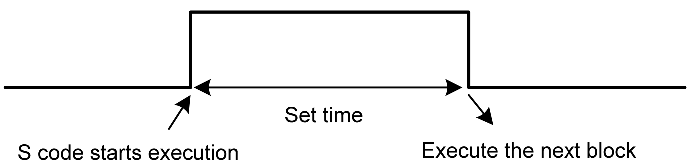


Fig. 5-1-1

Note: The system alarms below and stops the operation when the program specifies others except for the above S codes.

182：S code error, illegal S code is input to the program.

5.2 Spindle Analog Value Control

NO:1#4=1: the address S and its following numerical value directly commands the rotation quantity of the spindle (r/min), the unit of quantity of the spindle is different because of the different machine manufacturer. P55~P58 sets the max. speed of the spindle.

Command format： S__

The system realizes the spindle stepless speed regulating when the spindle analog value control is activated.

CHAPTER SIX TOOL FUNCTION T CODE

Use the address T and its following 2-digit numerical value to select the tool installed on the machine. In one block, the system can command a T code. They are executed simultaneously when the movement command and T code are in the same block.

Refer to the machine manufacturer’s user manual about T code use.

Use T code and its following numerical value to select the tool. The following two-digit numerical vaue are used for specifying the compensation number of the tool compensation.

T □□ □□

Tool compensation number

Tool selection number

The tool quantity provided by the system is set by P81, and its most setting is 8.

6.1 Tool change Process

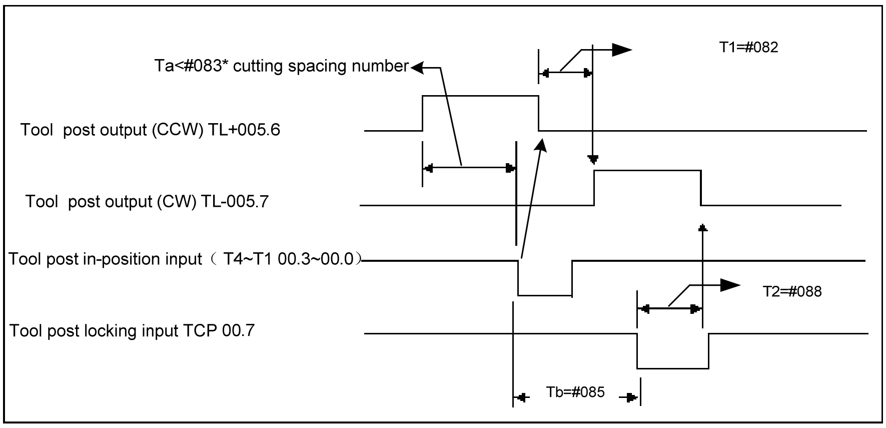


Fig**. 6-1-1**

In the above figure, P83 setting corresponds to the time parameter set by the parameter number.

When **Ta≥P8 ×** quantity of tool change, NO.185 alarm occurs: the tool change time is too long. The system alarms when it has not received the specified tool arrival signal after the time of the tool post rotating forward for Ta .

When **Ta≥P85 ×** quantity of tool change, NO.186 alarm occurs: the tool change time is too long. The system alarms when it has not received the specified tool arrival signal after the time of the tool post rotating reversely.

When T code is executed, the system output the tool post rotation (forward rotation signal) (TL+), and after it has received the tool arrival signal specified by T code, closes the signal and delays T1, then the tool post starts the reverse rotation to lock (TL-) and checks the locking signal *TCP; and after it has received the locking signal, delay the time set by P85, closes the reverse signal (TL-). When the executed tool number and the current tool number (it is recorded automatically in P49), the tool change command stops and the system executes the next block, otherwise, NO.187 alarm occurs.

After the system outputs the tool post's reverse rotation signal, has not received *TCP signal in the time set by P88, it alarms and closes the rotation signal.

Note: When the tool post mask position offsets, NO.187 alarm occurs after tool change, at the moment, the user should regulate the mask position or set No.10 BIT3 to 0.

6.2　Relevant Parameters of Tool Change

1）System parameters

The tool post in-position signal （***T8～*T1**）HIGH or LOW is set by **NO:9#1 TSGN**.

TSGN　**0 :**　the tool post in-position signal HIGH is valid.（normally-closed）

1 :　the tool post in-position signal LOW is valid. （normal open）

The tool post locking signal （*** TCP**）HIGH or LOW is set by **NO:9#0 TCPS**.

TCPS　**0 :**　the tool post locking signal LOW is valid.（normal open）

1 :　the tool post locking signal HIGH is valid.（normally-closed）

Note: When there is no tool post locking signal, it cannot be connected, it is valid when TCPS is LOW, P085 sets the reverse locking time. 980TA3 has no the signal *TCP, and TCPS is set to "0".

2）Relevant parameters of tool post

T1　: delay time from the tool post stopping forward to starting reversely rotation.

P82，set value: 0～9999999　unit：4ms

Ttool quantity : tool quantity of tool post.

P81, set value 1～8　unit：piece.

T2　：delay time after receiving locking signal of tool post reversely rotation.

P85, set value 0～9999999　unit：4ms

Tone tool：the maximum time of tool change.

P83, set value 0～9999999　unit：4ms

T total tools：the maximum time from the first to the last tool change.

P84，set value 0～9999999　unit：4ms

3） Diagnosis parameters

Tcurrent tool number：current tool number.

The diagnosis number P49. The system automatically sets its value, namely, the manual tool change is executed, i.e. automatic setting when the value is used firstly or the storage is cleared.

Ta ：The system automatically count the required maximum time from current tool change to the specified tool.

Ta = **T** tool×tool quantity of tool change.

Example: Supposing that the total tool quantity is 6.

a）The current tool is **1**，the specified is **5**，the tool of tool change is **4**.

b）The current tool is 5，the specified is **2**，the tool of tool change is **3.**

Tb ： The longest time signal for having received the tool post reverse rotation signal

6.3 Tool Offset

In the actual machining, the reference tool in programming is not almost consistent with the actual machining tool, at the moment, the difference between the standard position and atucal tool nose is called offset value.

The tool offset is controlled by T code instead of G code.

6.3.1 Basic Tool Offset

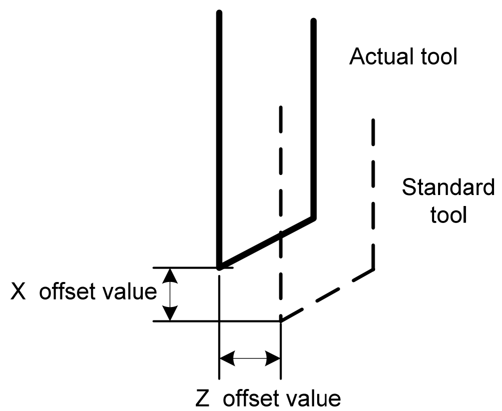


Fig. 6-3-1-1

The offset result makes the tool nose of the standard tool move in the programmed path, the tool installation is consistent with the initial point of standard tool in program, but the actual machining tool is not almost consistent with the standard. The difference between the standard position and the actual tool nose position is called offset value.

6.3.2 T Code for tool offset

T code's meanings:

T □□ □□

Tool offset number

Tool selection number

a) Tool selection

The tool selection is executed by specifying T code corresponded to the tool number. Refer to the machine manufacturer's user manual about relationship between the tool selection number and the tool

b) Tool offset number

It is used for selecting the offset value corresponded to the offset number. The offset value must be input by keyboard. The corresponding offset number has two offset values, one for X and another for Z. See Tool Compensation Display, Modification and Setting about its concrete operations.

Table 6-3-2-1

Offset value		
Offset number	X offset value	Z offset value
01	0.040	0.020
02	0.060	0.030
03	0	0
..	.	.
..	.	.
..	.	.

The tool offset is valid when T code is specified and its offset number is not 00.
When the offset number is 00, the tool offset function is cancelled.
Offset value is set:
Input in mm: -9999.999 mm～9999.999mm
NO:2#5 ORC can specify the diameter/radius specifications to X tool offset value.

6.3.3 Offset

Offset X, Z is for programmed path. T code specifies the offset value of offset number, and the offset value is added or subtracted at the end point of each block.

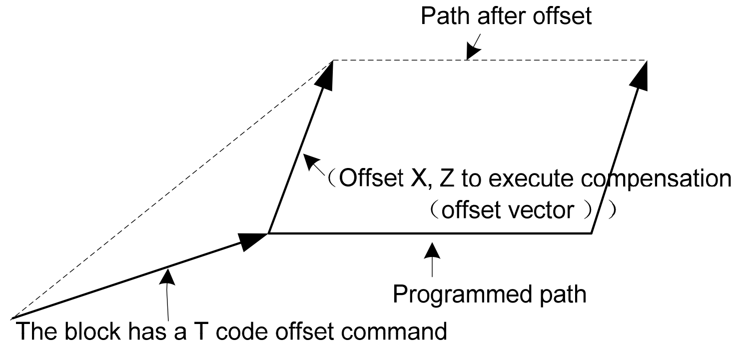


Fig. 6-3-3-1

a）Offset vector

In above figure, X, Z vector with offset is called offset vector. The compensation affects the offset vector.

b）Offset cancellation

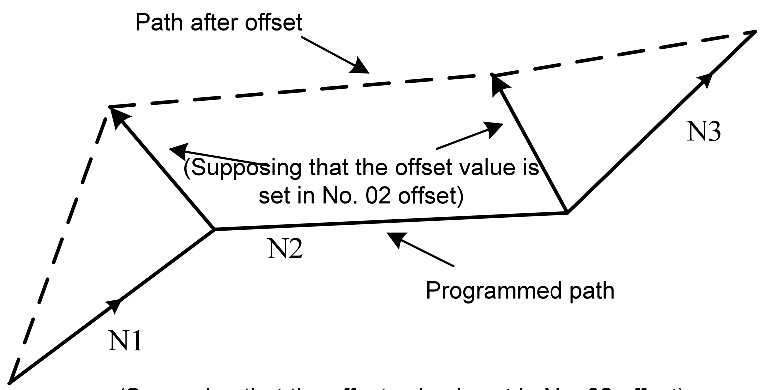
When the offset number of T code selects 00, the offset is cancelled. The offset vector is 0 at the end of canceling blocks.

N1 G01 U50.0 W100.0 T0202；

N2 W100.0；

N3 U0.0 W50.0 T0200；

Offset path



(Supposing that the offset value is set in No. 02 offset)

Fig. 6-3-3-2

Part Two Programming

6.3.4 Programming Example

Tool nose offset（Z，X）	tool number
Tool #1……B（0.120，0.200）	01
Tool #2……C(-0.180，-0.050)	02

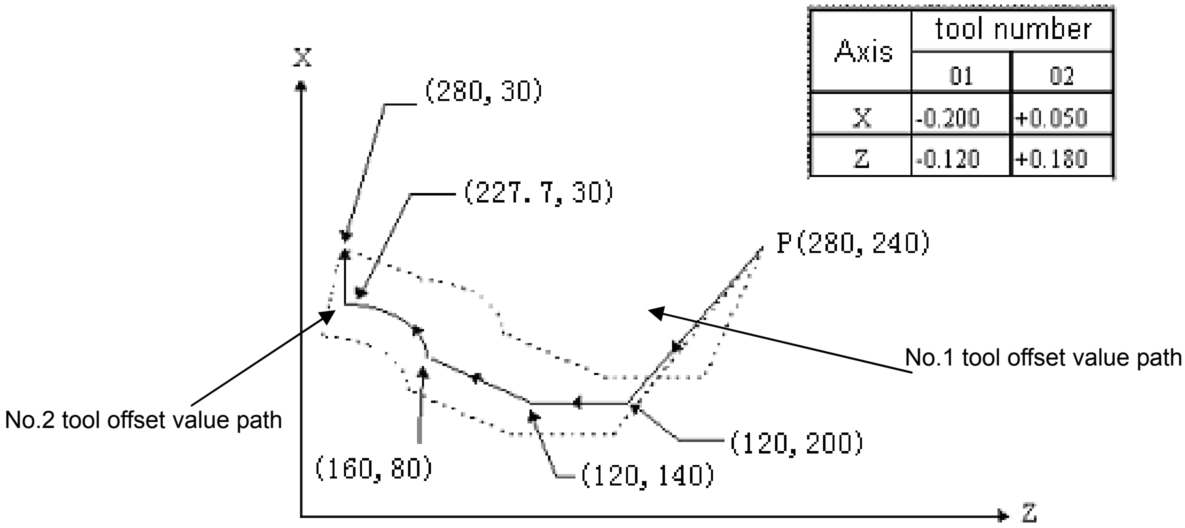


Fig. 6-3-4-1 tool compensation example- 2

(Program example 1)

G00 X280.0 Z240.0;

G00 X120.0 Z200.0 T0101;

G01 Z140.0 F30;

X160.0 Z80.0;

G03 X227.7 Z30.0 R53.81;

G00 X280.0 T0100;

#1 tool nose path is the same that of the programmed of the program.

（Program example 2）

Modifying example 1, #2 tool nose path is the same that of the programmed.

T0101→T0202 and T0100→T0200

6.3.5 Single T code

When T code and the movement function command are in the same block, the traverse speed to execute the tool length compensation is decided by the movement command to cutting feedrate or rapid traverse speed.

NO:2#0=0: when a single T code is executed, the speed of tool length compensation is determined by the current mode: when the current mode is the cutting feed, the tool length compensation is executed at the current cutting federate; when the current mode is G00, it is done at the current rapid traverse speed.

NO:2#0=1: when a single T code is executed, the tool length compensation and the first subsequent movement command are executed simultaneously, the speed of the tool length compensation is determined by the movement command.

CHAPTER SEVEN USER MACRO PROGRAM

A group of command for realizing some function is saved to storage in advance like subprograms, one command standing for these functions is used. Only the command input in programs can realize these function. The group is called user macro program body, the command is called ‘user macro command’. The user macro program body is called macro program sometime. The user macro code is also called macro program call command.

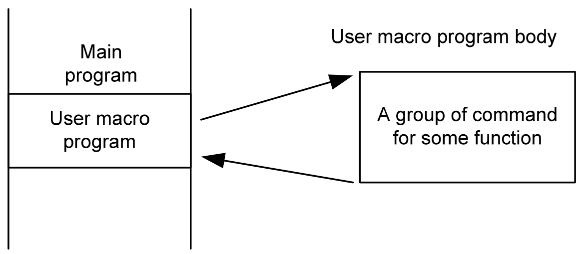


Fig. 7-1

The programmer need not remember macro program body, but does user macro codes as commands.

Obvious features of user macro programs the variable can be used in user macro program body. The variables can be operated and macro code is assigned.

7.1 User Macro Code

The user macro code is a command to call user macro program body.

Command format：

M98 P□□□□

Program number of called macro program body

Using the above command can call macro program body specified by P.

7.2 User Macro Program Body

The user macro program body can use generally CNC command, or use variable, operation or transfer command.

User macro program body begins with a program number following O and ends with M99.

O08000;	Program number
G65 H01 … …;	Operation command
G00 X#101 … …;	CNC command using variable
…;	
…;	
G65 H82 … …;	Transfer command
M99;	End of user macro program body
…;	
…;	
M99;	

Fig.7-2-1 User macro program body constitution

1） Variable usage

Variables can command the address values in user macro program body. The variable value is assigned by a main program or set by key board, or is assigned when the user macro program body is executed.

Use many variables, and they can be distinguished by their variable numbers.

a）Variable meaning

The variable number with # means the measurement, and its format is shown below:

#（i=200，202，203，204……）

(Example) #205，#209，#1005

b）Variable reference

Using variable can replace the numerical value following the address.

When a program has "<Address># i" or "<Address>—# i"，it means to take the variable value or the negative of the variable value as the address value.

(Example）F#203…#203=15: it is the same as F15.

Z-#210…#210=250: it is the same as Z-250.

G#230…#230=3: it is the same as G3.

Variable replacing its variable number can describe into ##9200 instead of ##9200, "9" following # means the replacement variable number.

The following examples are to displace the variable number.

（**Example** #200 = 205，#205 = 500:

X#200 and X500 are the same.

X-#200 and X-500 are the same.

Note 1: O and N cannot reference variables, and cannot use O#200，N#220 in programming.

Note 2: When the variable exceeds the maximum programmable dimension described by the address, it cannot be used. When #230 = 120, M#230 exceeds its maximum programmable dimension.

Note 3: Display and setting of variable value: it can be displayed on LCD window, and can be set by press key.

2）Variable classification

The variable is divided into the common and system according to the variable number, and their use and properties are different.

a）Common variable #200～#231

The common variable is common in main programs or user macro programs called by main programs. Namely, variable # i used in some user macro program and # i used in other macro programs are the same. So, the common variable #i of operation result in some macro program can be used to other macro programs.

The common variable use is defined by the user.

Common variable #200～#231 are all "0" when the power supply is turned on again, after the power supply is turned on again.

b）Common variable #500～#515

The variable use is the same those of ＃200～＃231. After the power supply is tuned off, the variable data is saved, and remains when power-on again.

c）System variable #1000～#1031

The variable use is fixed in the system.

Interface input signals #1000～#1031(select the function- need to match corresponding selected parts).

After the system reads the system variable #1000～#1031 values as the interface signals, it can learn states of interface input signals.

Diagnosis No.	Tool No.: 7	6	5	4	3	2	1	0
001	TCP	DIQP	*DECX	DITW	*SP	*ST	*DECZ	*ESP
Programmed variable number Pin	#1007 XS39：12	#1006 XS39：11	#1005 XS40：1	#1004 XS40：2	#1003 XS40：7	#1002 XS40：8	#1001 XS40：9	#1000 XS40：10

T08	T07	T06	T05	T04	T03	T02	T01
#1015 XS40：19	#1014 XS40：20	#1013 XS40：21	#1012 XS40：22	#1011 XS40：3	#1010 XS40：4	#1009 XS40：5	#1008 XS40：6

SAR	LTZ	LTX	SPEN	PCH	DOOR	GR2	GR1

003

#1023 XS41： 7	#1022 XS41：19	#1021 XS41：6	#1020 XS41：5	#1019 XS41：4	#1018 XS41：3	#1017 XS41：2	#1016 XS41： 1

004

M01	M93	M91	LTY	DECY	LCK	OWA2	OWA1
#1031 XS42：12	#1030 XS42：11	#1029 XS41 ：10	#1028 XS41：22	#1027 XS41：9	#1026 XS41：21	#1025 XS41：8	#1024 XS41：20

Value of 32-digit DI signal is automatically assigned to macro variable （#1000～#1031）inside the system. It with other logic judge, skip macro code are used together to perform various of execution.

#1000～1031 are values of corresponding points (0/1).

Interface output signal #1100～1131 (select the function- need to match corresponding selected parts).

Assign the system variable #1100～1131 to change state of output signals.

Diagnosis No.

005

	Bit No：7	6	5	4	3	2	1	0
Macro program	SPZD	DOQPJ	M05	M32	M08	M10	M04	M03
Programmed variable No. → / Pin →	#1107 XS40 ：17	#1106 XS40：4	#1105 XS39 ：16	#1104 XS39：6	#1103 XS39 ：15	#1102 XS39：2	#1101 XS39：3	#1100 XS39：7

006

TL-	TL+	DOQPS	M11	M44	M43	M42	M41
#1115 XS40：13	#1114 XS40：12	#1113 XS39：10	#1112 XS39：9	#1111 XS39：8	#1110 XS39：14	#1109 XS39：1	#1108 XS39：5

M53	M51	TLC	STAR	TRIAN	WAR	M30	MST
#1123 XS42：17	#1122 XS42：4	#1121 XS42：16	#1120 XS42：15	#1119 XS42：3	#1118 XS42：2	#1117 XS42：14	#1116 XS42：1

	M69	M67	M65	M63	M61	M59	M57	M55
008	**#1131** **XS41：13**	**#1130** **XS41：12**	**#1129** **XS42：10**	**#1128** **XS42：9**	**#1127** **XS42：8**	**#1126** **XS42：7**	**#1125** **XS42：6**	**#1124** **XS42：5**

32-digit DO signal outputs 1 or 0 by macro variable （#1100～#1131）assignment.

7.3 Operation Command and Transfer Command（G65）

General format:

G65 Hm P# i Q# j r# k；

m：01～99 means operation command or transfer command function.

i：variable name to save operation result.

j：variable name 1 to execute operation, it can be constant.

k：variable name 2 to execute operation, it can be constant.

Meaning：# i = #j O # k

It is an operator, is specified by Hm.

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

P#200 Q#201 R15….#200 = #201 O 15；

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

Note 1: The variable value does not include the decimal point. Meanings of each variable value and each address without decimal point are the same.

(Example）#200 = 10

X#200=X 10mm(input with mm)

Note 2: The constant is directly described without #.

Note 3: H code specified by G65 does not affect the offset value.

Table 7-3-1

G code	H code	Function	Definition
G65	H01	Assignment	# i = # j
G65	H02	Addition	# i = # j + # k
G65	H03	Subtraction	# i = # j - # k
G65	H04	Multiplication	# i = # j×# k
G65	H05	Division	# i = # j÷# k
G65	H11	Logic addition（OR）	# i = # j OR # k

G65	H12	Logic multiplication (AND)	# i = # j AND # k
G65	H13	OR	# i = # j XOR # k
G65	H21	Square root	# i = $\sqrt{\# j}$
G65	H22	Absolute value	# i = |# j |
G65	H23	Remainder	# i = # j -trunc(#j÷# k)×＃k
G65	H24	Decimal to binary system	# i = BIN(# j)
G65	H25	Binary to decimal system	# i = BCD(# j)
G65	H26	Composite multiplication, division operation	# i = # i×# j÷# k
G65	H27	Composite square root	# i = $\sqrt{\# j^2 + \# k^2}$
G65	H31	Sine	# i = # j×SIN(# k)
G65	H32	Cosine	# i = # j×COS(# k)
G65	H33	Tangent	# i = # j×TAN(# k)
G65	H34	Arc tangent	# i = ATAN(# j / # k)
G65	H80	Unconditional transfer	Turn to N
G65	H81	Conditional transfer 1	IF# j = # k，GOTON
G65	H82	Conditional transfer 2	IF# j ≠ # k，GOTON
G65	H83	Conditional transfer 3	IF# j ＞ # k，GOTON
G65	H84	Conditional transfer 4	IF# j ＜ # k，GOTON
G65	H85	Conditional transfer 5	IF# j ≥ # k，GOTON
G65	H86	Conditional transfer 6	IF# j ≤ # k，GOTON
G65	H99	P/S alarm	400+N P/S alarms

1. Operation command

1）Variable assignment ：# I = # J

G65 H01 P#I Q#J

（Example）G65 H01 P# 201 Q1005； （#201 = 1005）

G65 H01 P#201 Q#210； （#201 = #210）

G65 H01 P#201 Q-#202； （#201 = -#202）

2）Addition operation：# I = # J+# K

G65 H02 P#I Q#J R#K

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

3）Subtraction operation：# I = # J－# K

G65 H03 P#I Q#J R# K；

Part Two Programming

（Example）G65 H03 P#201 Q#202 R#203； （#201 = #202－#203）

4）Multiplication ：# I = # J×# K

G65 H04 P#I Q#J R#K；

（Example）G65 H04 P#201 Q#202 R#203； （#201 = #202×#203）

5）Division operation：# I = # J÷# K

G65 H05 P#I Q#J R#K

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

6）Logic addition（OR）：# I = # J.OR. # K

G65 H11 P#I Q#J R#K；

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

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

G65 H12 P#I Q#J R#K；

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

8）XOR：# I = # J.XOR. # K

G65 H13 P#I Q#J R#K

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

9）Square root：# I = $\sqrt{\#J}$

G65 H21 P#I Q#J ；

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

10）Absolute value：# I = | # J |

G65 H22 P#I Q#J ；

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

11）Remainder ：# I = # J－TRUNC(#J/#K)×# K，TRUNC：trunc

G65 H23 P#I Q#J R#K

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

12）Decimal be converted into binary system：# I = BIN (# J)

G65 H24 P#I Q#J ；

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

13）Binary be converted into decimal system：# I = BCD (# J)

G65 H25 P#I Q#J ；

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

14）Composite multiplication, division operation：# I =（# I×# J）÷# K

G65 H26 P#I Q#J R# k；

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

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

<u>G65 H27 P#I Q#J R# k；</u>

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

16）Sine：# I = # J•SIN（# K）（Unit：deg）

<u>G65 H31 P#I Q#J R#K；</u>

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

17）Cosine：# I = # J•COS（# K）（Unit：deg）

<u>G65 H32 P#I Q#J R# k；</u>

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

18）Tangent ：# I = # J•TAM（# K）（Unit：deg）

<u>G65 H33 P#I Q#J R# K；</u>

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

19）Arc tangent：# I = ATAN（# J /# K）（Unit：deg）

<u>G65 H34 P#I Q#J R# k；</u>

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

2. Transfer command

a）Unconditional transfer

G65 <u>H80</u> Pn； n：serial number

(Example）G65 H80 P120；（transfer to N120 block）

b）Conditional transfer 1 #J.EQ.# K (=)

G65 H81 Pn Q#J R# K； n：serial number

(Example) G65 H81 P1000 Q#201 R#202；

201 = #202: transfer to N1000 block; #201 ≠ #202: the system orderly executes the blocks.

c）Conditional transfer 2 #J.NE.# K (≠)

G65 H82 Pn Q#J R# K； n：serial number

(Example) G65 H82 P1000 Q#201 R#202；

201 ≠ #202: transfer to N1000 block; #201 = #202: the system orderly executes blocks.

d）Conditional transfer 3 #J.GT.# K (＞)

G65 H83 Pn Q#J R# K； n：serial number

(Example) G65 H83 P1000 Q#201 R#202；

201 ＞ #202: transfer to N1000 block; #201 ≤ #202: the system orderly executes blocks.

Part Two Programming

e）Conditional transfer 4 #J.LT.# K (＜)

G65 H84 Pn Q#J R# K； n：serial number

(Example) G65 H84 P1000 Q#201 R#202；

201＜ #202: transfer to N1000 block; #201 ≥ #202: the system orderly executes blocks..

f）Conditional transfer 5 #J.GE.# K (≥)

G65 H85 Pn Q#J R# K； n：serial number

(Example) G65 H85 P1000 Q#201 R#202；

201 ≥ #202: transfer toN1000 block; #201 ＜ #202: the system orderly executes blocks..

g）Conditional transfer 6 #J.LE.# K (≤)

G65 H86 Pn Q#J R# K； n：serial number

(Example) G65 H86 P1000 Q#201 R#202；

201 ≤ #202: transfer to N1000 block; #201 ＞ #202: the system orderly executes blocks.

h）P/S alarm

G65 H99 Pi i：alarm number +400

(Example) G65 H99 P15

P/S 415 alarm.

Note 1: When the serial number of transferred address is specified to the positive, the system firstly searches in positive direction and then in reverse direction; when it is specified to the negative, it firstly does in reverse direction, and in positive direction.

Note 2: Variable can specifies the serial number.

G65 H81 P#200 Q#201 R#202；

When the condition satisfies, the block transfers the block which serial number is specified by #200.

7.4 Notes for User Macro Program Body

1. Key board input

   Press # following the address G, X, Z, U, W, R, I, K, F, H, M, S, T, P, Q, and # is input.

2. The macro program cannot be commanded in MDI mode, otherwise, the system alarms.
3. Operation, transfer command H.P.Q.R can be input anywhere in G65 block.

   H02 G65 P#200 Q#201 R#202； …correct

   N100 G65 H01 P#200 Q10； …correct

4. Single block

   Generally, the operation, transfer blocks are executed, the single block switch is ON, the system does not stop. NO:13 #0 SBKM can make the single block stop. Such is applied to macro program debugging.

5. A variable value should be -2P32P～+2P32P-1， but the system only displays the range -9999.999～9999.999. The exceeded range is displayed to “*******”.
6. A subprogram can execute quadruple nest.
7. The variable value rounds numbers, and decimal part of operation result must be rounded.
8. Execution time of operation, transfer command is different, and the general average time is 10ms.

7.5 User Macro Program Example

1）Custom code M81（automatic feed）

Main **program**：

```
O0001；
N10 G00 X100 Z100；     (Positioning to safety position)
N20 G00 U50 F100；      (Rapid positioning)
N30 G01 U0.8            (Feed)
N40 M81；               (Call 09081 subprogram)
N50 G0 X100 Z100；      (Feed end, coordinates returning to zero）
N60 M99；               (Repetitive executeion）
```

Subprogram（custom M81）：

```
O9081；
N10 G65 H01 P#1104 Q1；    （U04=1: output feed signal indication）
G65 H82 P20 Q#1004 R1；    （Whether X14=1, X14=0: N20 program is executed）
G65 H01 P#1100 Q0；        （Cancel feed signal indication U04=0）
M99 P50；                  (Transfer to main program N50）
N20 M99 P30；              （Transfer to main program N30）
```

Part Two Programming

CHAPTER EIGHT TOOL COMPENSATION C FUNCTION

The actual tool nose is not a point but an arc. Influenced by the tool nose arc, an error occurs between the actual machined result and the workpiece program, and the tool compensation C function can realize the tool radius compensation to clear the above error. The corresponding parameter must be set to activate the function when the tool compensation C function is executed: NO:2#3 is set to 1.

0	0	2						CCMP			

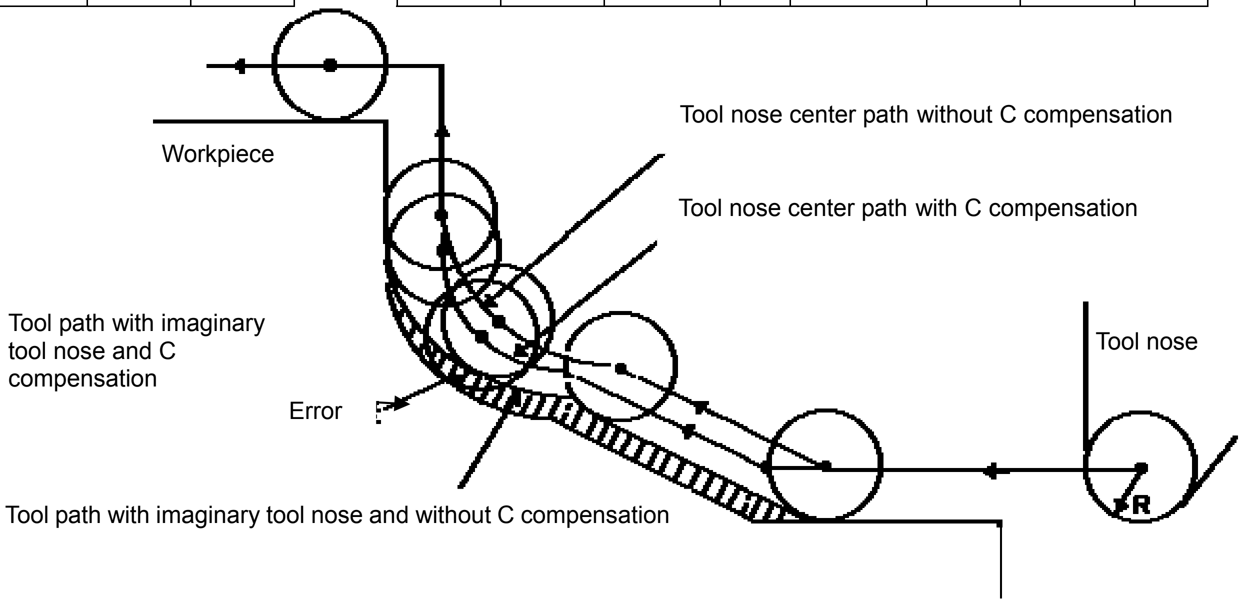


Fig. 8-1

8.1 Basic Concept of Tool Compensation C Function

8.1.1 Imaginary Tool Nose

The tool nose A point described in Fig. 8-1-1-1 is imaginary, and does not exist actually, and is called imaginary tool nose. Generally, because it is difficult to set the tool nose radius center on the initial position, and it is easy to set the imaginary tool nose on the initial position, which is shown below. Do not concern the tool nose radius when the imaginary tool nose is executed in programming.

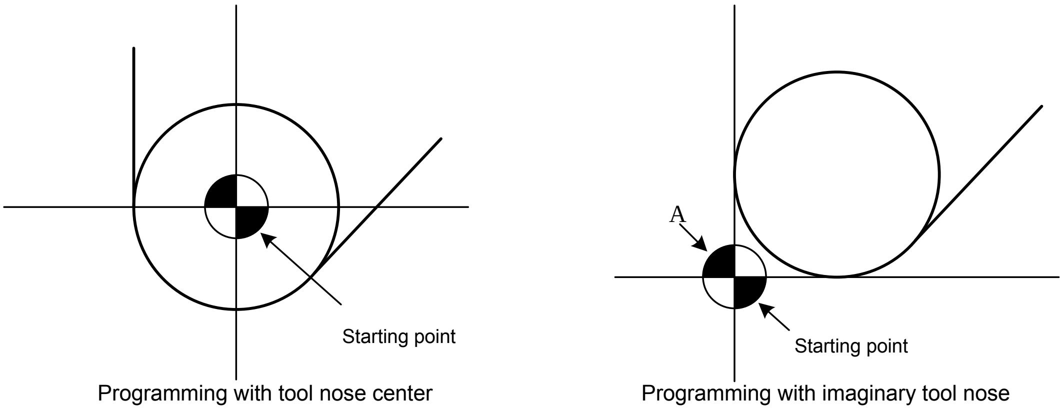


Fig. 8-1-1-1 tool nose radius center and imaginary tool nose

Note: For the machine with machine zero, a standard point such as tool post center is taken as a starting point. The distance from the standard point to the tool nose radius center or imaginary tool nose is the tool offset value.

The distance between the standard point and the tool nose radius center is offset value. For setting the tool offset value, it is easier to measure the distanced between the standard point to imaginary tool nose than the standard point to the tool nose radius center, generally, the distance between the standard point and imaginary tool nose is set to the tool offset value.

When the tool post center is taken as a starting point, the tool offset value is shown below:

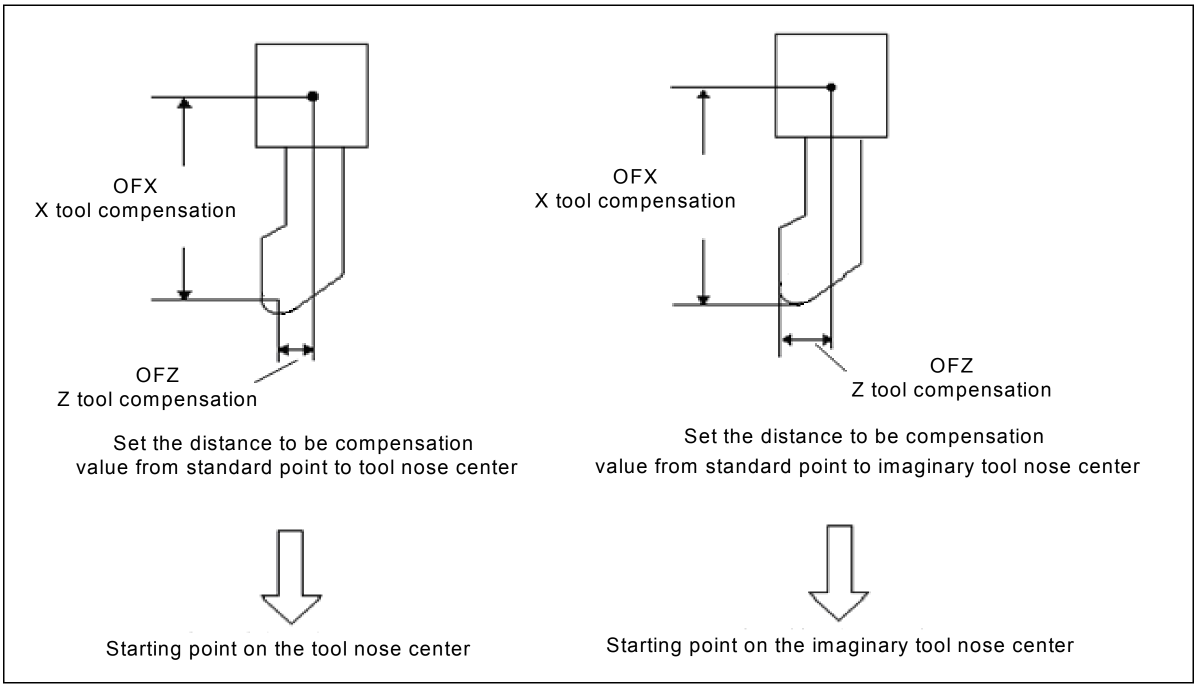


Fig. 8-1-1-2 tool offset value setting when tool post center as standard point

Fig. 8-1-1-3 and Fig. 8-1-1-4 are tool paths of tool nose center programming and imaginary tool nose programming. The left figure is radius compensation without tool nose radius, the right is one with tool nose radius compensation.

The imaginary tool nose path is the same programmed path without the tool nose radius compensation

use tool nose radius compensation used to as the precise cutting

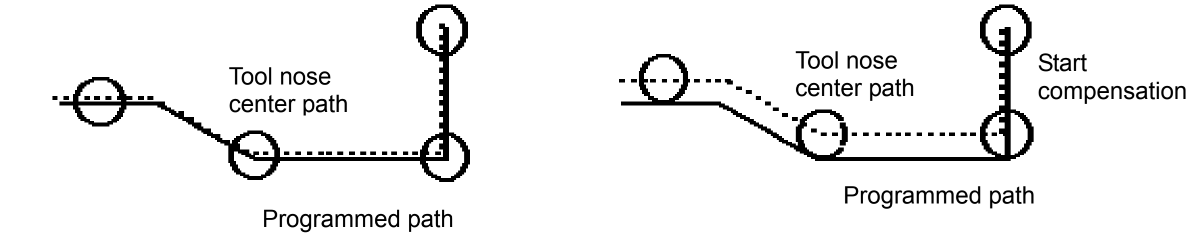


Fig. 8-1-1-3 Tool path in tool nose center programming

The imaginary tool nose path is the same to as the programmed path without the tool nose radius compensation

use tool nose radius compensation used precise cutting

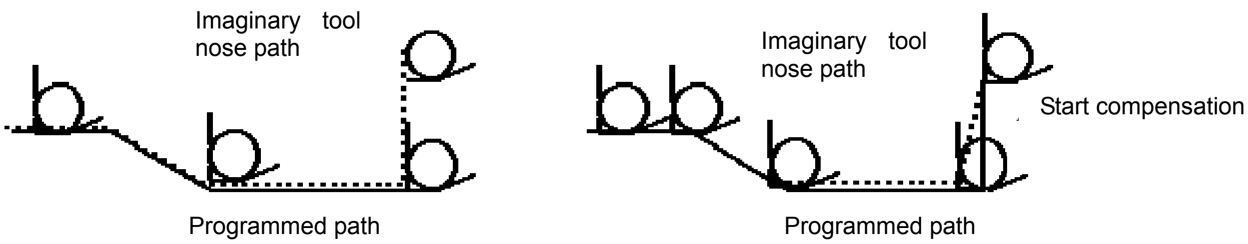
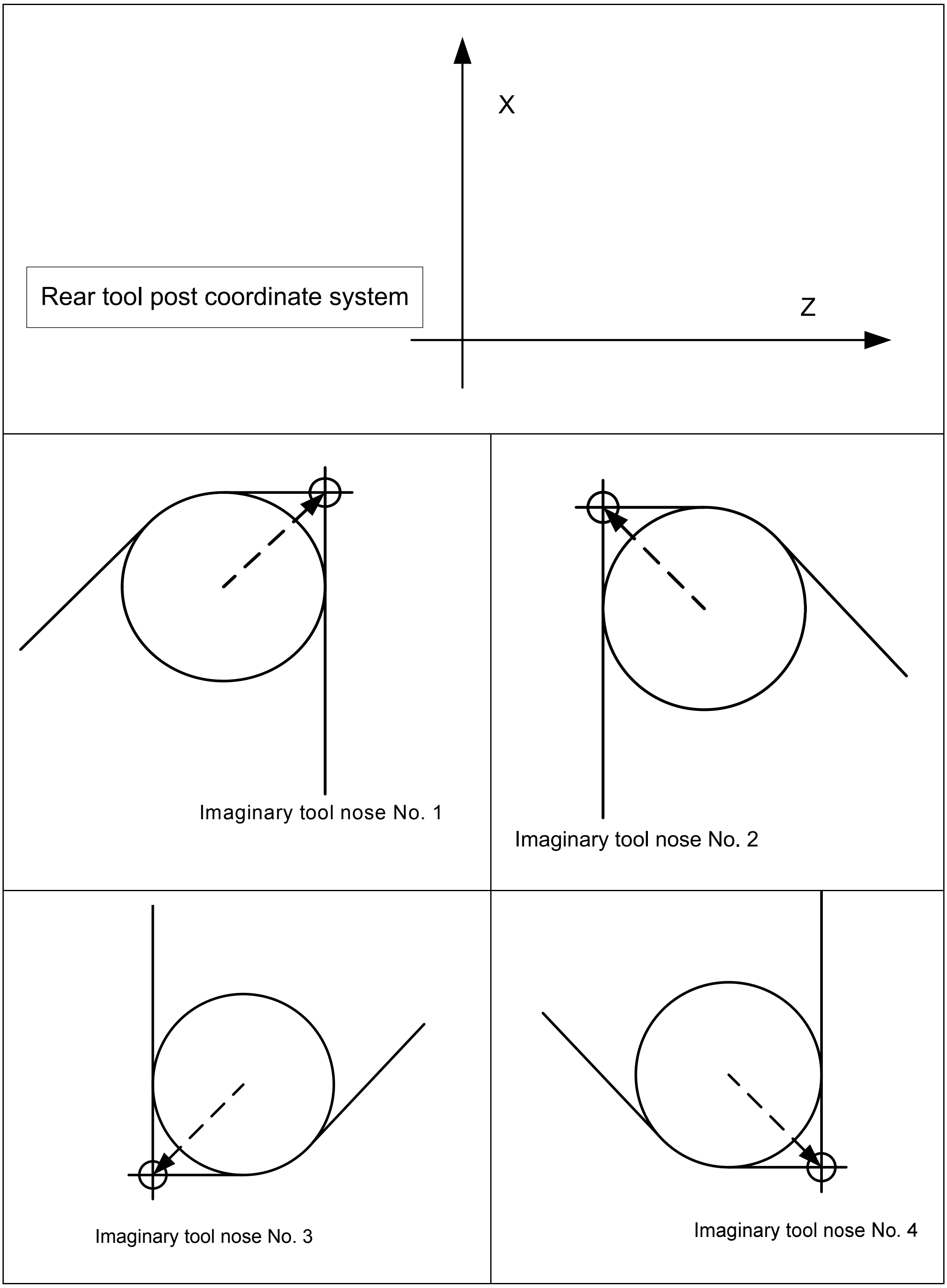


Fig. 8-1-1-4 Tool path in imaginary tool nose programming

8.1.2 Imaginary Tool Nose Direction

In actual machining, there are different position relations between the tool and workpiece. The imaginary tool nose direction is determined by the tool direction when cutting.

The imaginary tool nose number defines the position relations between the imaginary tool point and tool nose arc center. The imaginary tool nose number is divided into 10 kinds （0～9）, meaning position relations in 9 directions. The tool nose directions are different in different coordinate system (rear tool post coordinate system and front tool post coordinate system) even if they are the same tool nose direction numbers as the following figures. In figures, it represents relationships between tool nose and starting point, and end point of arrowhead is the imaginary tool nose.


X
Z
Rear tool post coordinate system
Imaginary tool nose No. 1
Imaginary tool nose No. 2
Imaginary tool nose No. 3
Imaginary tool nose No. 4

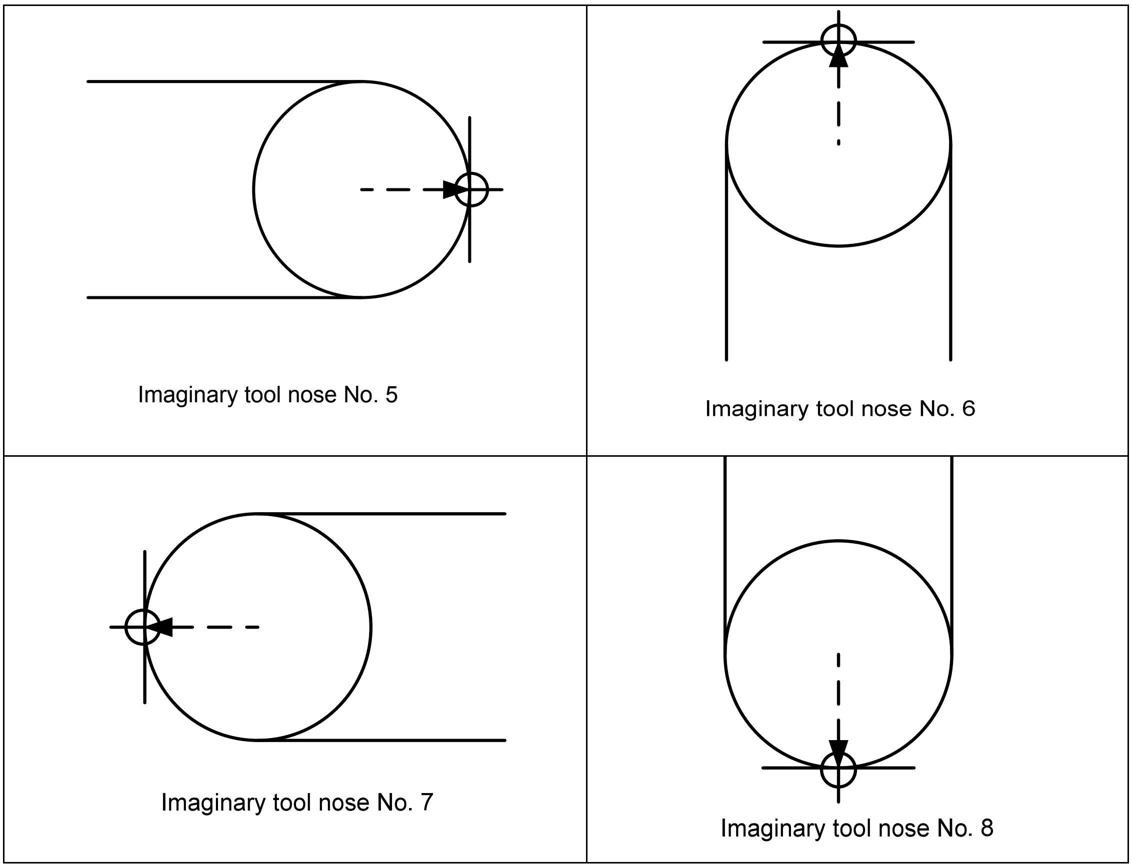
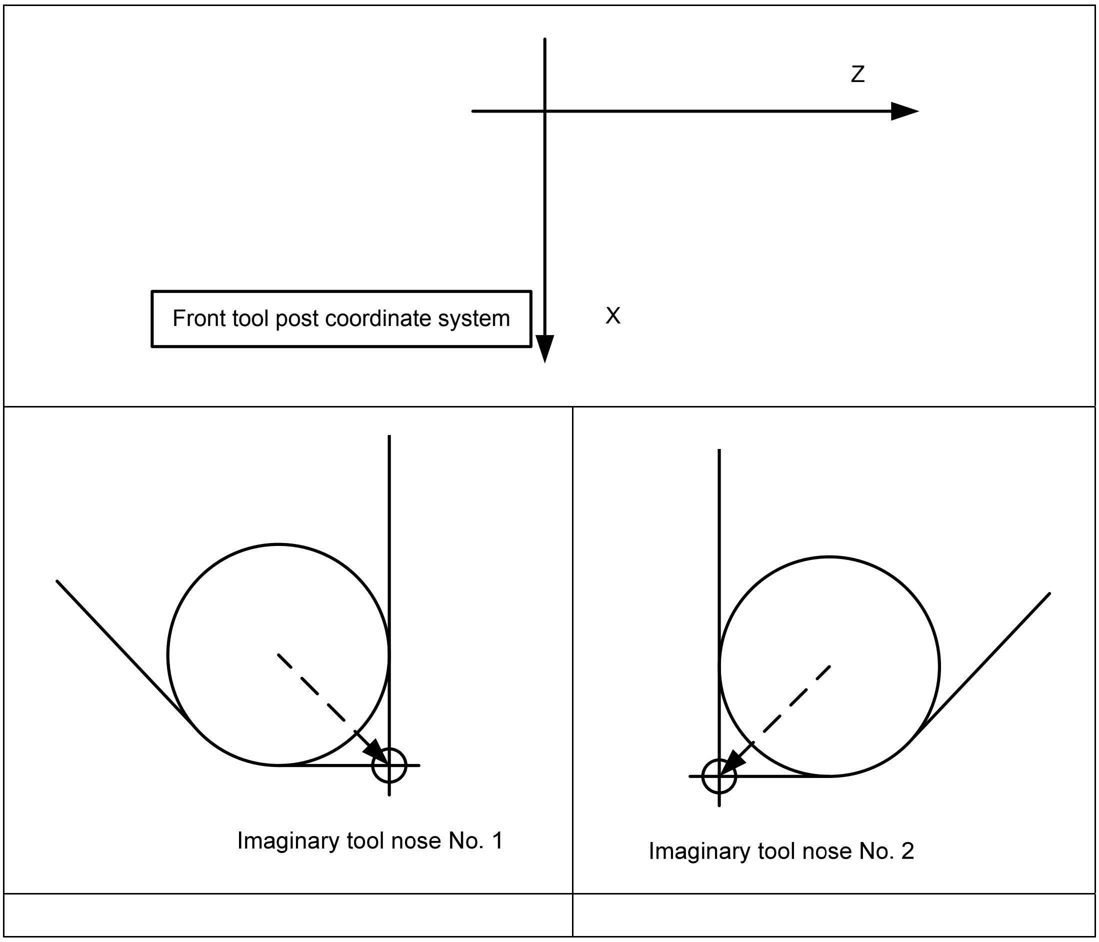


Fig. 8-1-2-1 Imaginary tool number in rear tool post coordinate system



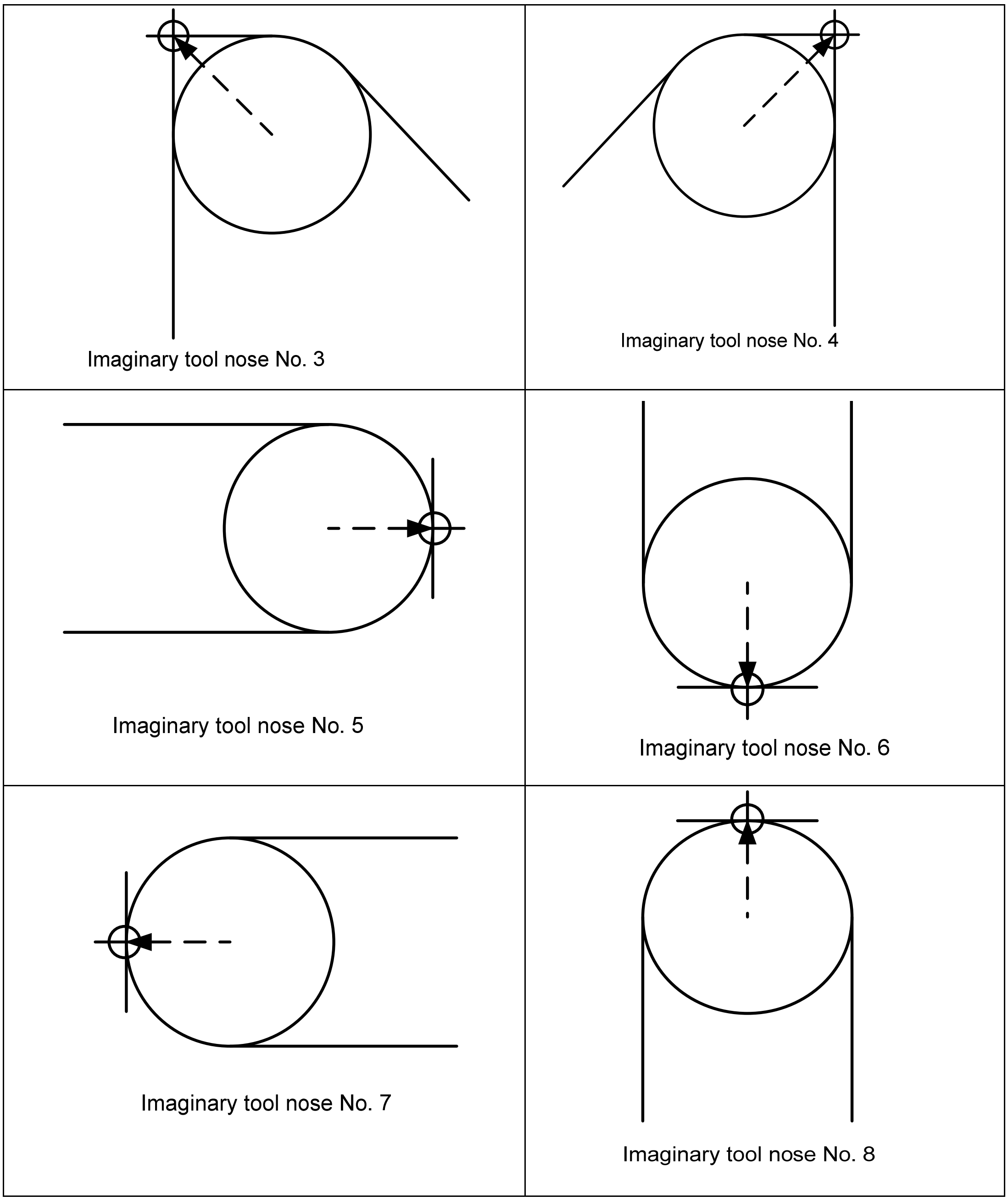


Fig. 8-1-2-2 Imaginary tool nose number in front tool post coordinate system

When the tool nose center and the starting point are consistent, the tool nose number is set to 0 or 9. The corresponding tool compensation number uses the address T to set the imaginary tool nose number.

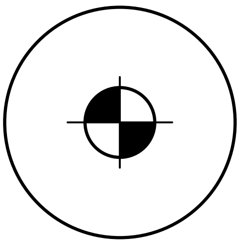


Fig**. 8-1-2-3 tool nose center and starting point be consistent**

8.1.3 Compensation Value Setting

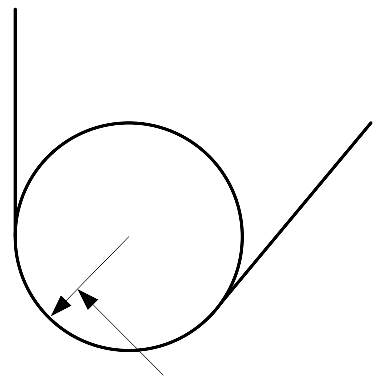


Fig. 8-1-3-1 compensation value of tool nose radius

Set the following compensation values before executing the tool nose radius compensation: X, Z, R, T. X, Z are tool offset values from the tool post center to the tool nose in X, Z direction; R is the radius compensation value of imaginary tool nose; T is the imaginary tool nose number. One value corresponds to a tool compensation number, which is set in tool compensation window. See Modify and Set Tool Compensation Value.

The concrete is shown in Table 8-1-3-1:

Table 8-1-3-1 tool nose **radius compensation value display window**

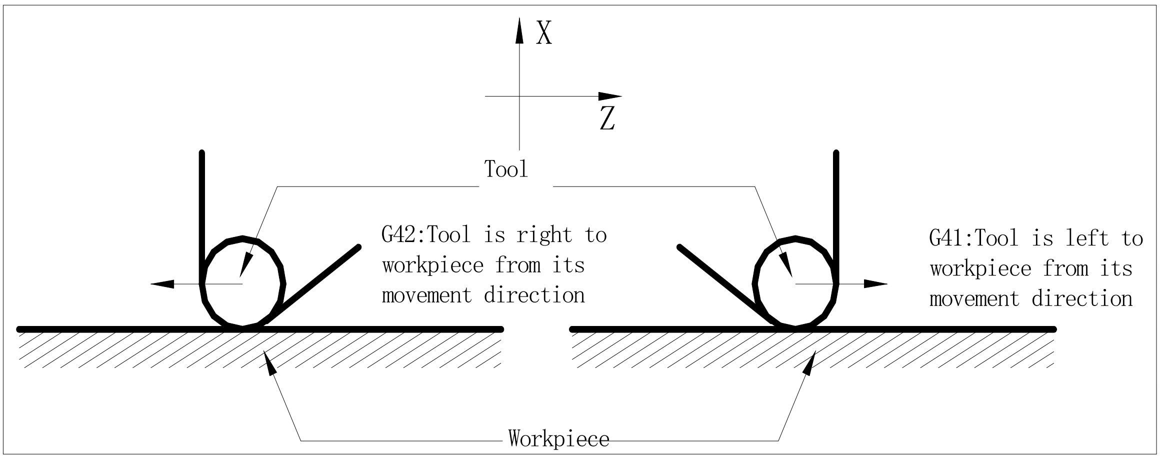
NNo.	X	Z	R	T
001	0.020	0.030	0.020	2
002	0.060	0.060	0.016	3
..	..	..	..	..
..	..	..	..	..
..	..	..	..	..
015	0.030	0.026	0.18	9
064	0.050	0.038	0.20	1

8.1.4 Relative Position between Tool and Workpiece

The relative position between the tool and workpiece must be specified when the tool radius compensation is executed. In rear tool post coordinate system, the tool center path is the right of the programmed path (part path) advancing direction, which is called right tool compensation, using G42; the tool center path is the left of the programmed path (part path) advancing direction, which is called left tool compensation, using G41. In front tool post system, they are reverse. The relative position is shown in Table 8-1-4-1 when G40, G41, G42 is commanded.

Table 8-1-4-1

Command	Function Explanation	Remark
G40	Cancel tool nose radius compensation	
G41	Tool nose radius left compensation in rear tool post coordinate system, tool nose radius right compensation in front tool post coordinate system	
G42	Tool nose radius right compensation in rear tool post coordinate system, tool nose radius left compensation in front tool post coordinate system	



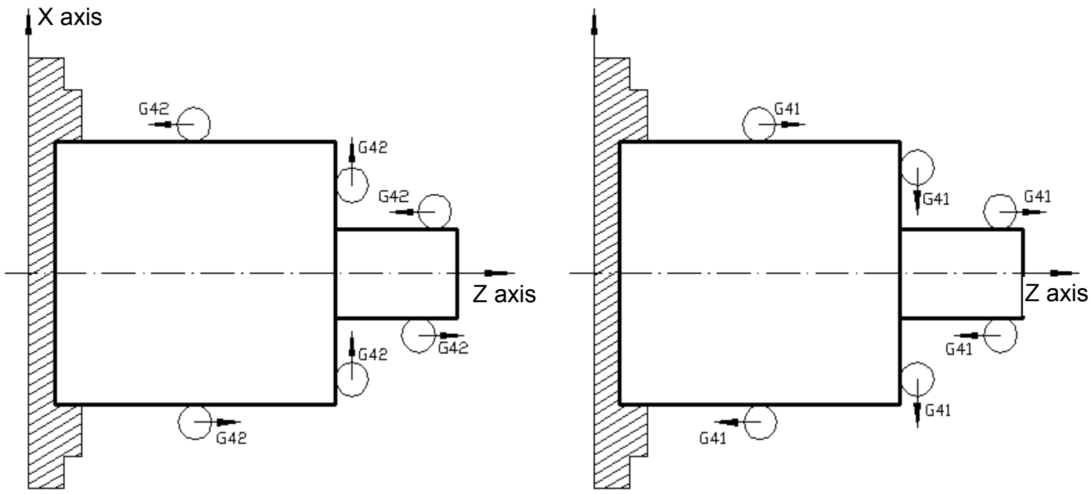


Fig. 8-14-1 tool nose radius compensation in rear tool post coordinate system

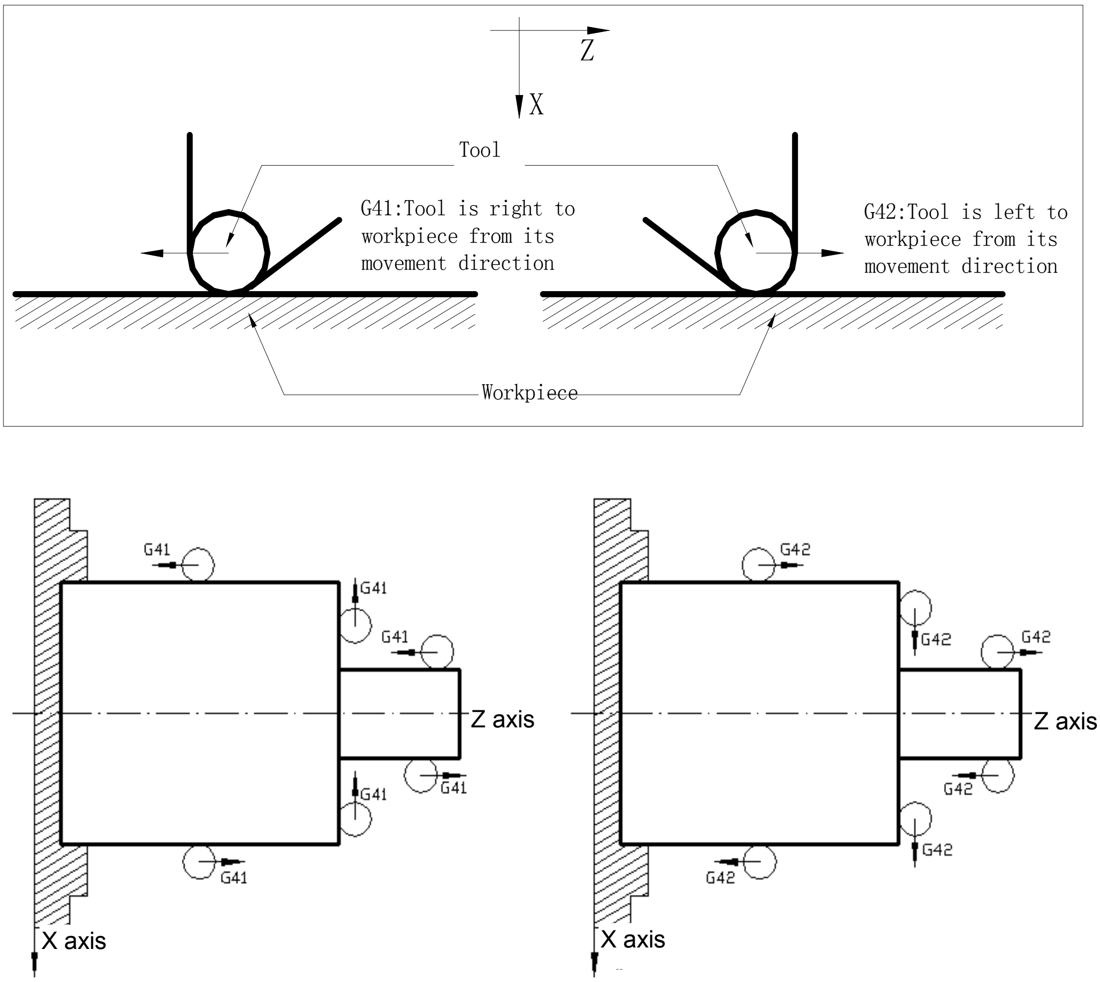


Fig. 8-1-4-2 tool nose radius compensation in front tool post coordinate system

8.1.5 Inside and outside

When the tool nose radius compensation is executed, the corners of previous and current programmed path are different, and their tool nose compensation paths are so. The intersection angle of the block intersection of two movement blocks at the side of workpiece is more than or equal to 180°, which is called inside, the intersection angle is 0～180°, which is called outside.

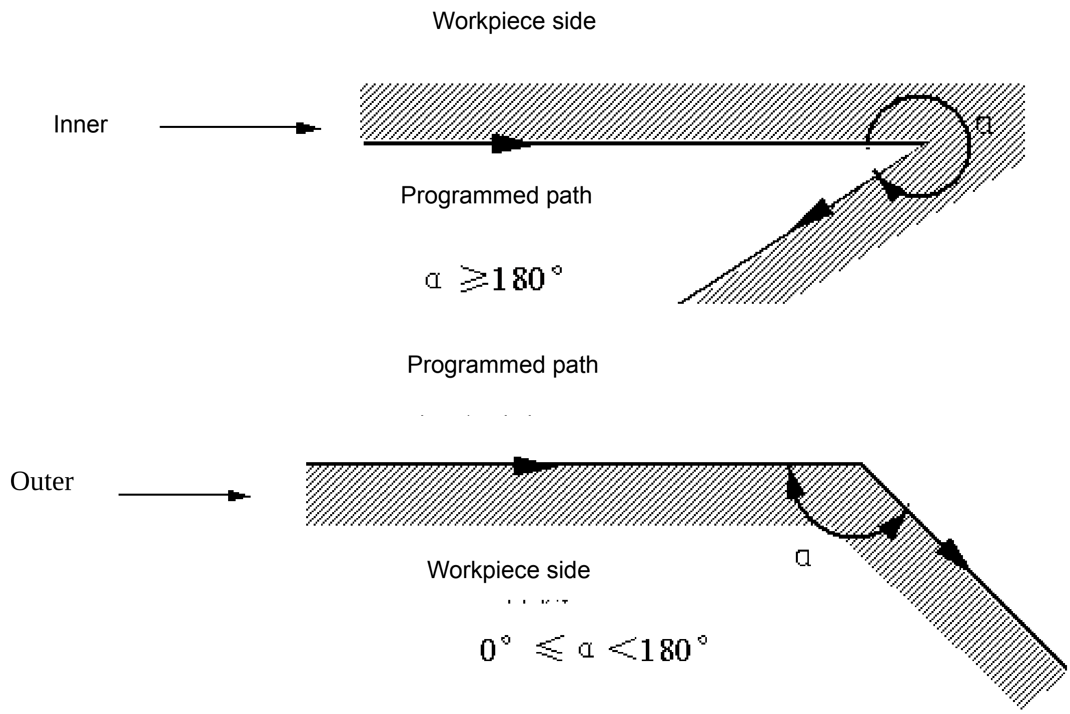


Fig. 8-1-5-1 Inner and outside

8.1.6 Command Format of G41, G42, G40

Command format：

$$\left\{\begin{matrix}G40\\G41\\G42\end{matrix}\right\}\left\{\begin{matrix}G00\\G01\end{matrix}\right\} X\text{—}\quad Z\text{—}$$

Note 1: G40, G41, G42 are modal.

Note 2: G02/G03 following G41/G42 can be used after the tool compensation is established normally.

8.2 Concrete Tool Compensation

8.2.1 Concrete Path Decomposition of Tool Nose Radius Compensation

The tool radius compensation steps are divided into three: tool compensation establishment, execution and cancel.

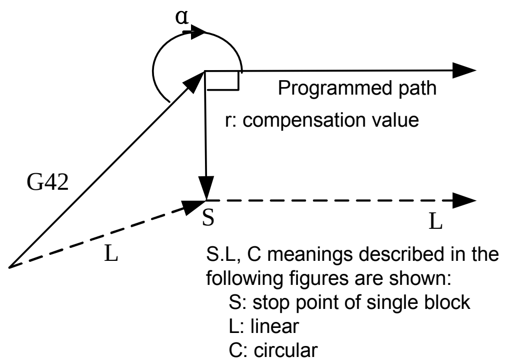
1. Tool compensation establishment

The execution from offset cancel to offset mode is called tool compensation establishment.

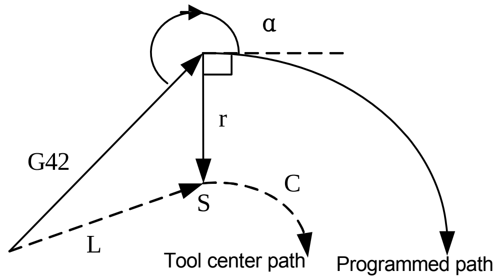
The concrete tool compensation establishment is shown in Fig. 8-2-1-1:

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

（i）Linear—linear

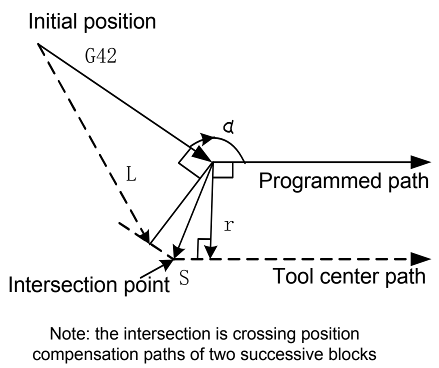


（ii）Linear—circular

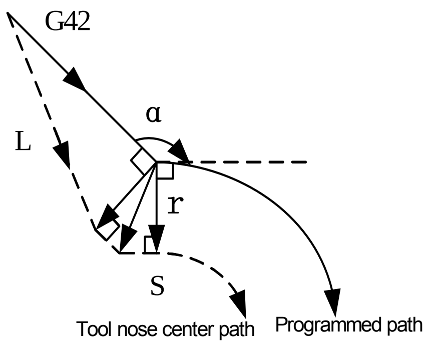


b) Tool traversing outside along corner (180°＞α≥90°）

（i）Linear-linear

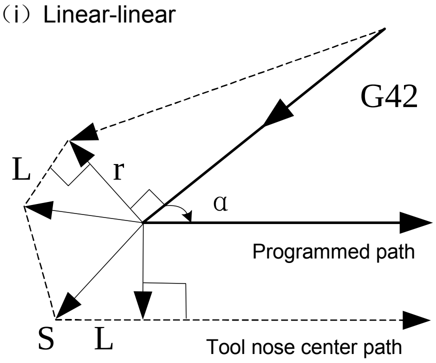


（ii）Linear—circular

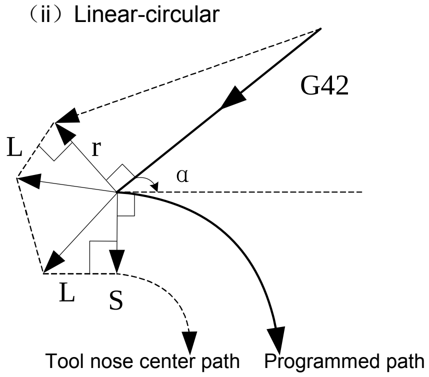


(c) Tool traversing outside along corner（α＜90°)

（i）Linear-linear



（ii）Linear-circular



(d) Tool traversing outside along corner, linear→linear.（α＜1°)

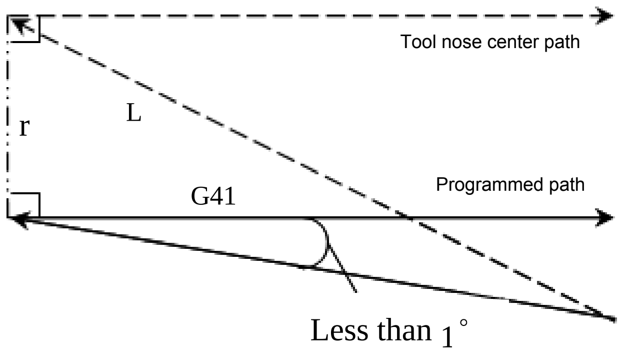


Fig. 8-2-1-1 tool compensation establishment

Note 1: When the tool compensation is established, the tool compensation number is not specified or is zero, the alarm #036 occurs.

Note 2: When the tool compensation is established, it is executed by G0 or G1; the #034 alarm occurs when it is done by G02 or G03.

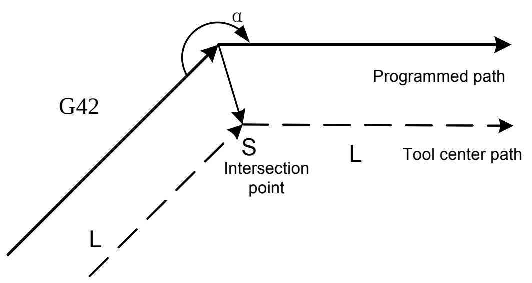
2. Tool compensation proceeding

The offset path after the tool compensation establishment and before the tool compensation cancel is called tool compensation proceeding.

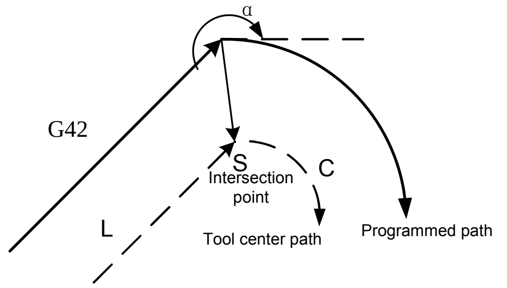
The tool compensation proceeding is shown in Fig. 8-2-1-2 and Fig. 8-2-1-3:

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

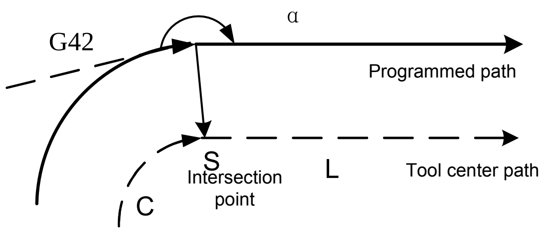
Linear → linear



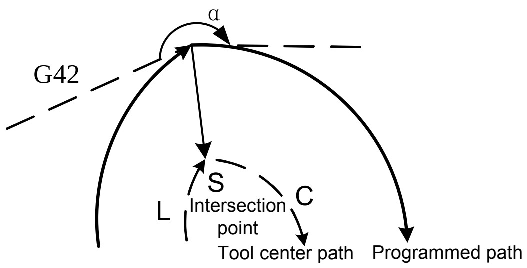
Linear → circular



Circular → linear

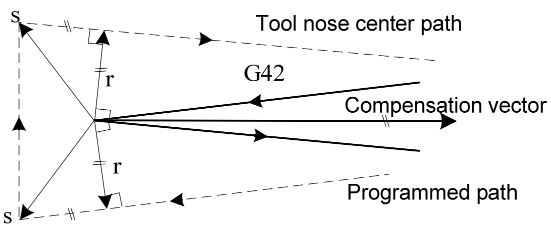


Circular → circular



（V）Inside machining less than 1 degree and zooming out compensation vector

(i) Circular → Circular



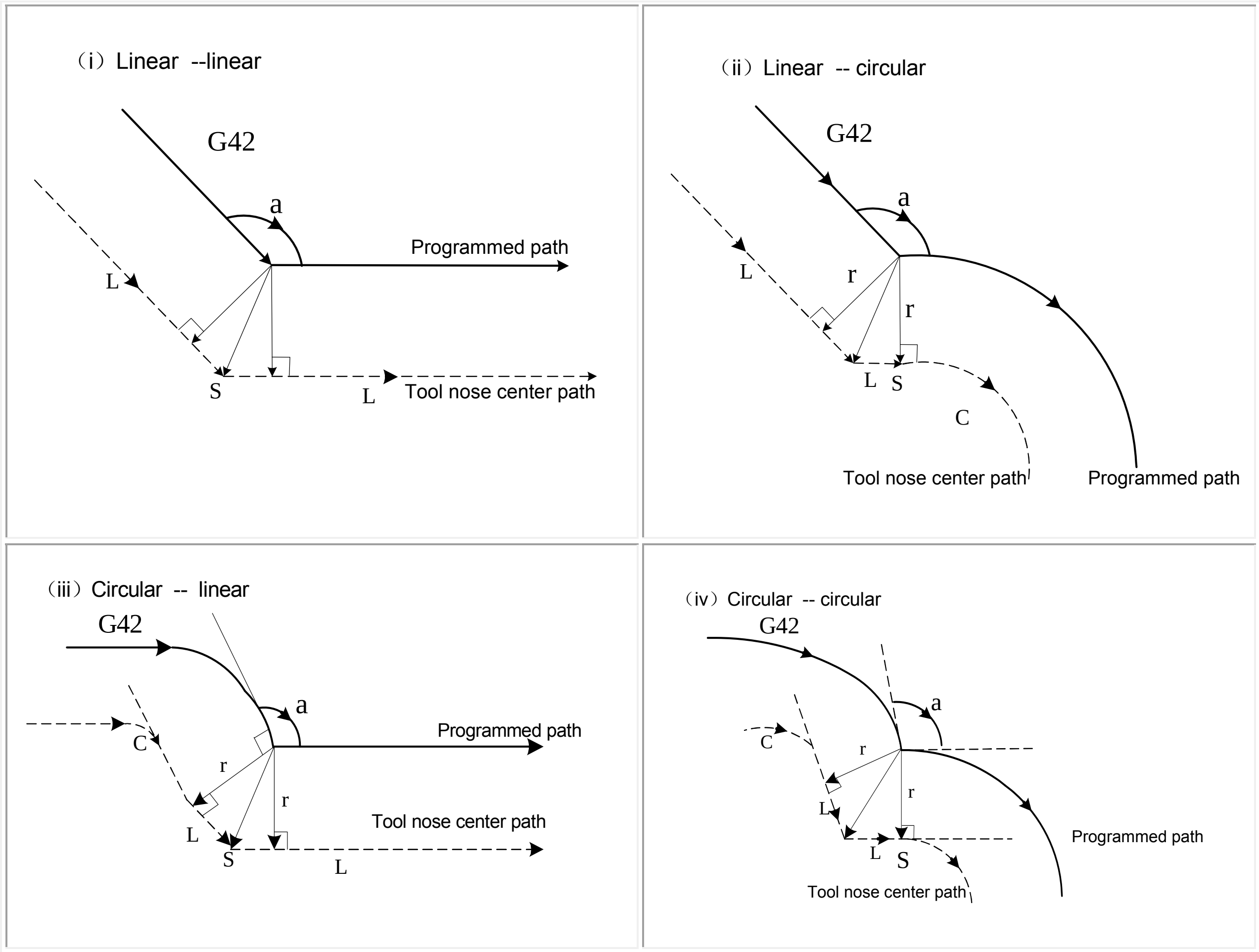
Use the same method to the followings

(ii) Circular → Linear

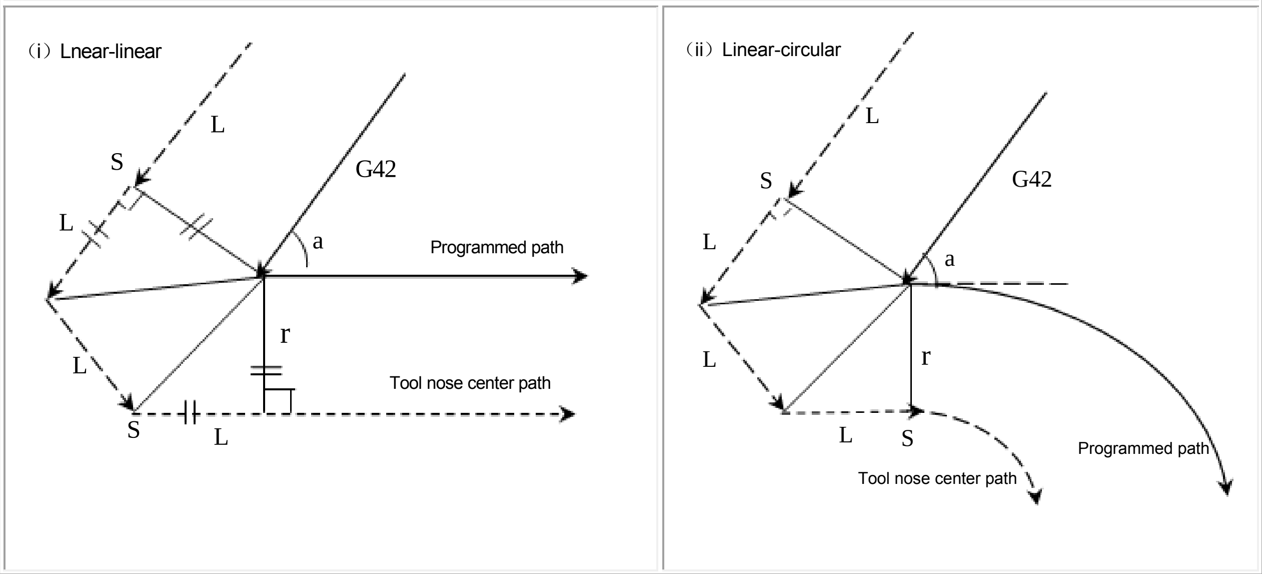
(iii) Linear → Circular

(iv) Circular → Circular

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



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



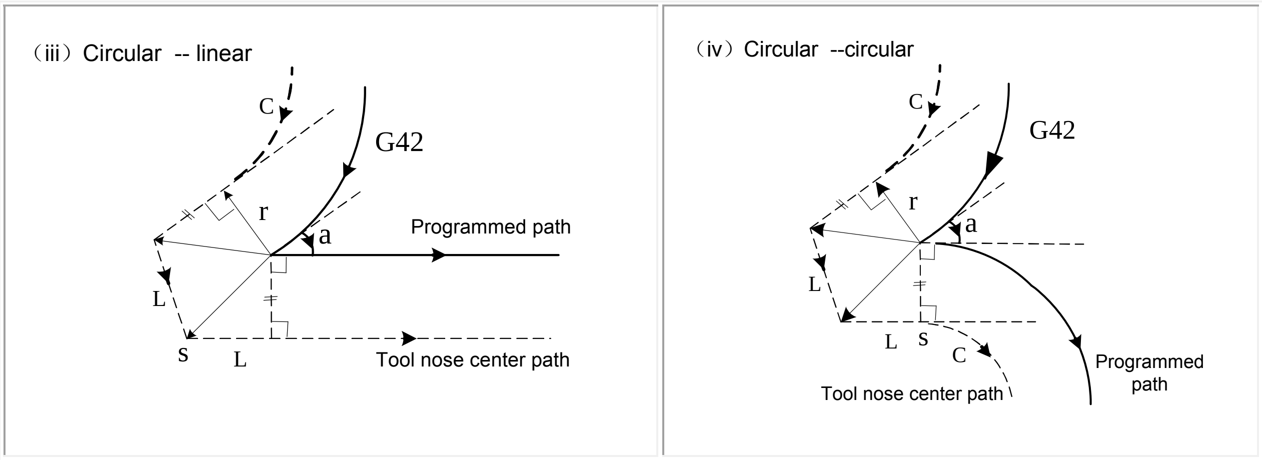
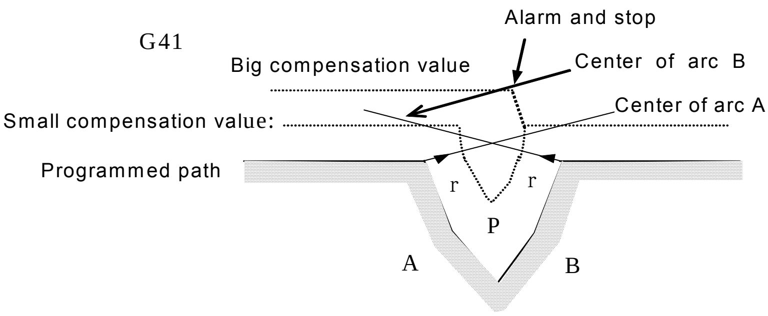


Fig. 8-2-1-2 tool compensation execution

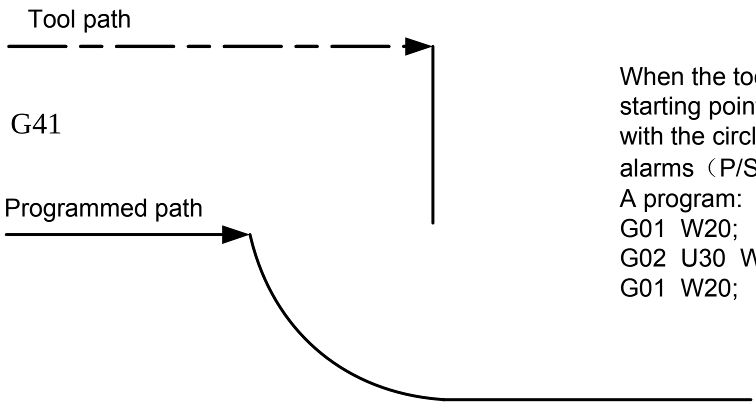
(d) Special

1）Without intersection point:



In the left figure, when the arc compensation path has an intersection point, but when the radius becomes big, the intersection point lose, at the moment, the system alarms（**P/S33**）.

2) Circle center coinciding with starting point/end point



When the tool radius is excessive big, starting point or end point coincides with the circle center , the system alarms（P/S38）

A program:

G01 W20;

G02 U30 W15 R15;

G01 W20;

Fig. 8-2-1-3 tool compensation execution

3. Tool compensation cancel

In compensation mode, when a program meets anyone of the following conditions, the system enters the compensation cancel mode, and the block operation is called tool compensation cancel.

（a） Using G40 cancels C tool compensation, executing tool compensation cancel cannot use

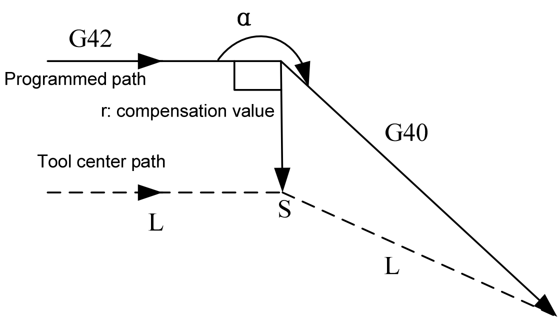
G02, G03, otherwise, NO. 34 alarm occurs and the tool stops.

（b） The tool radius compensation number is specified to 0.

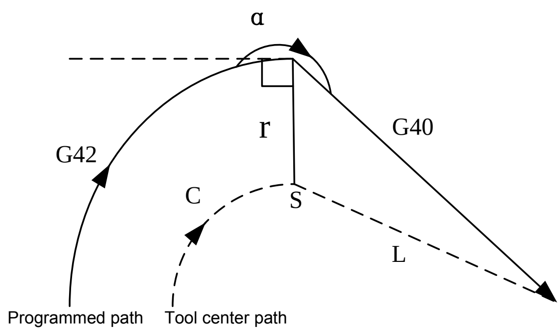
The concrete tool compensation cancel is shown below:

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

（i）Linear—linear

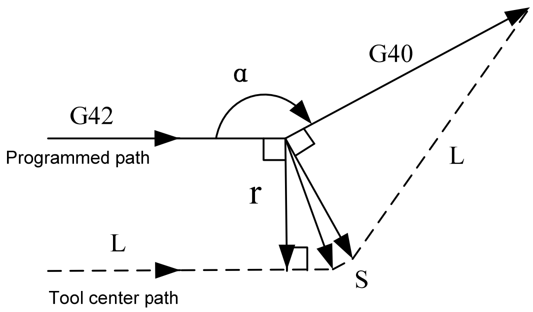


（i）Circular—linear

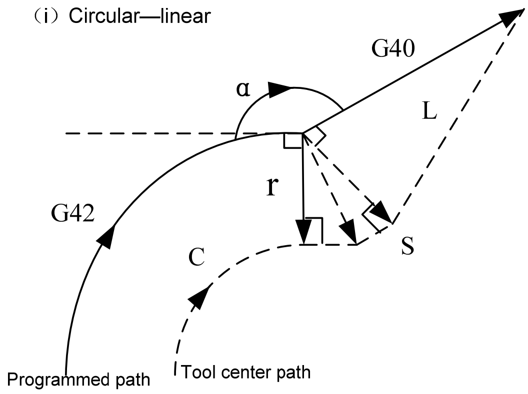


（b）Tool traversing outside along corner（180°＞α≥90°）

（i）Linear—linear

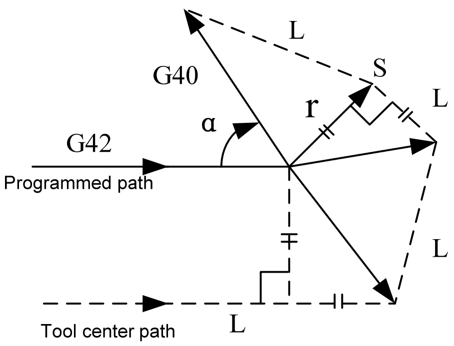


（i）Circular—linear

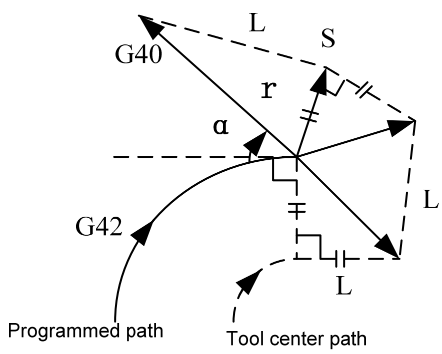


（c）Tool traversing outside along corner（α＜90°）

（i）Linear—linear



（i）Circular—linear



（d）Tool traversing outside along corner, linear→linear.（α＜1°）

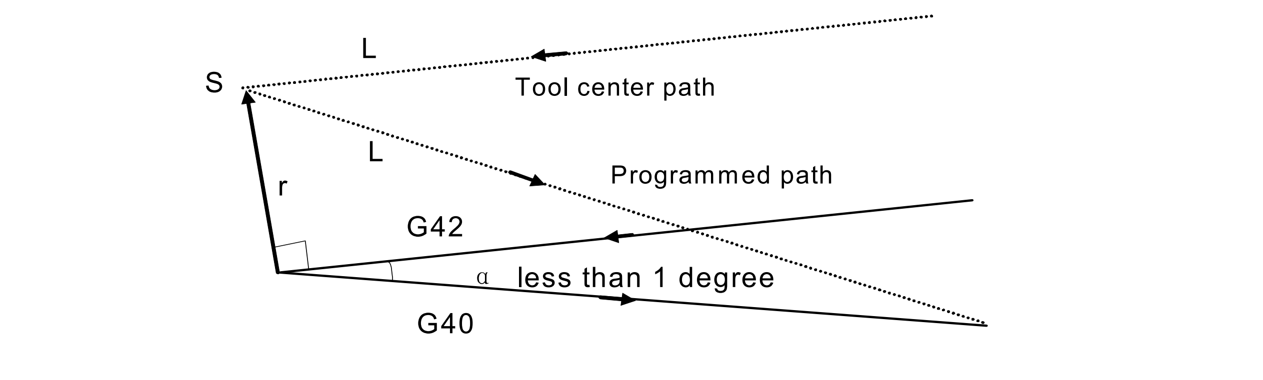


Fig. 8-2-1-4 tool compensation cancel

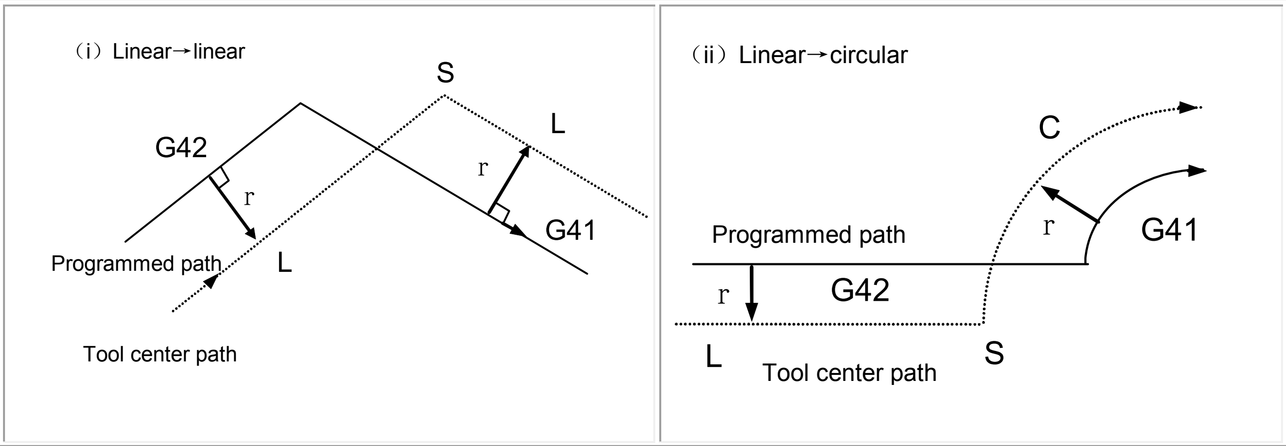
8.2.2 Compensation direction change when tool compensation proceeding

The tool compensation G code (G41 and G42) determines the compensation direction, and the compensation value sign is shown below:

Table 8-2-2-1

Compensation value signs / G code	+	-
G41	Left compensation	Right compensation
G42	Right compensation	Left compensation

In special conditions, the compensation direction can be changed in the compensation mode, but it cannot be changed in the initial of block. The following compensation value is positive.



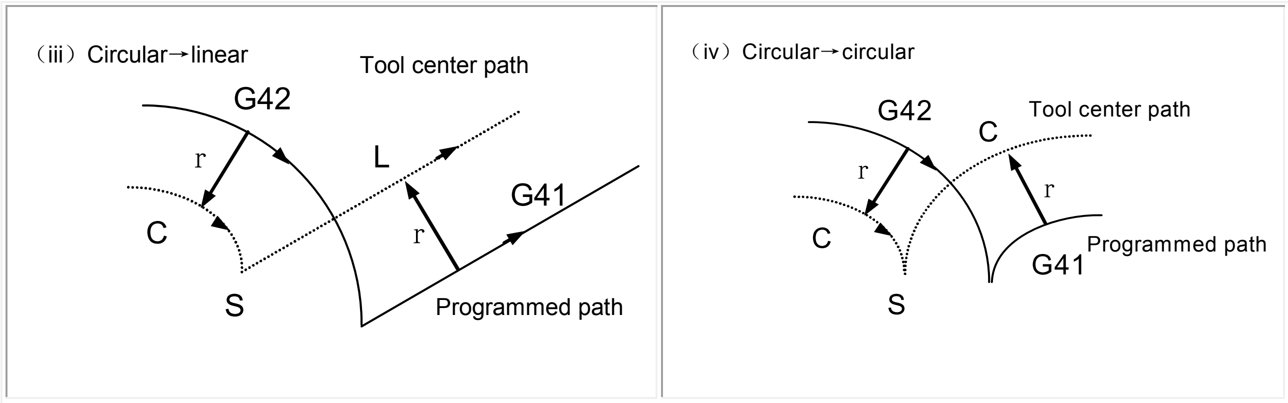


Fig. 8-2-2-1 changing compensation direction when tool compensation proceeding

When the compensation is normally executed, there is no intersection point, G41 or G42 changes compensation direction, the offset path from the block A to B is shown below:

i) <u>Linear-----linear</u>

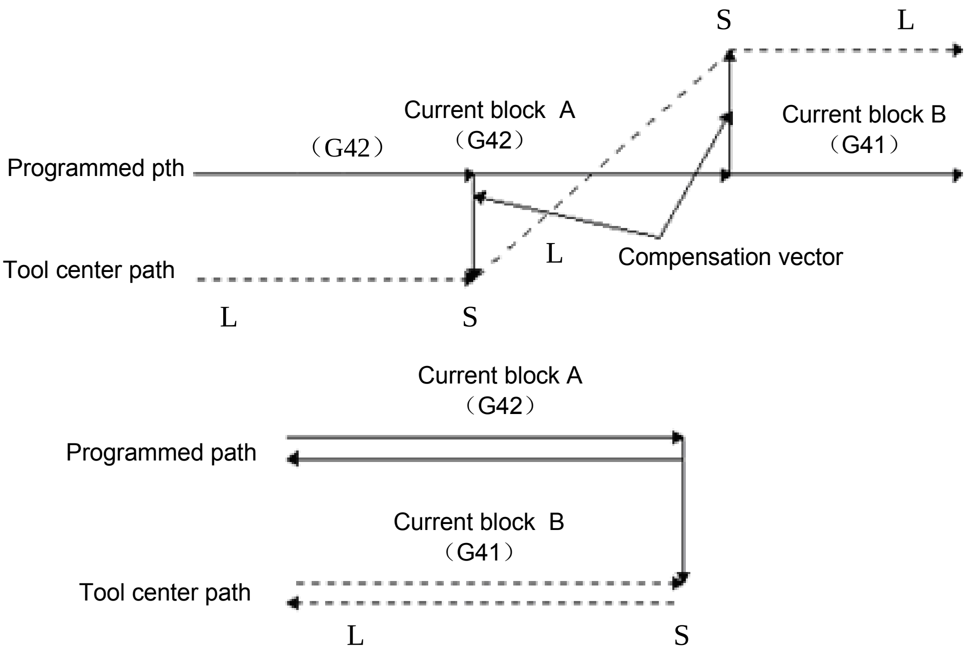


Fig. 8-2-2-2 linear — linear

ii) <u>Linear-----circular</u>

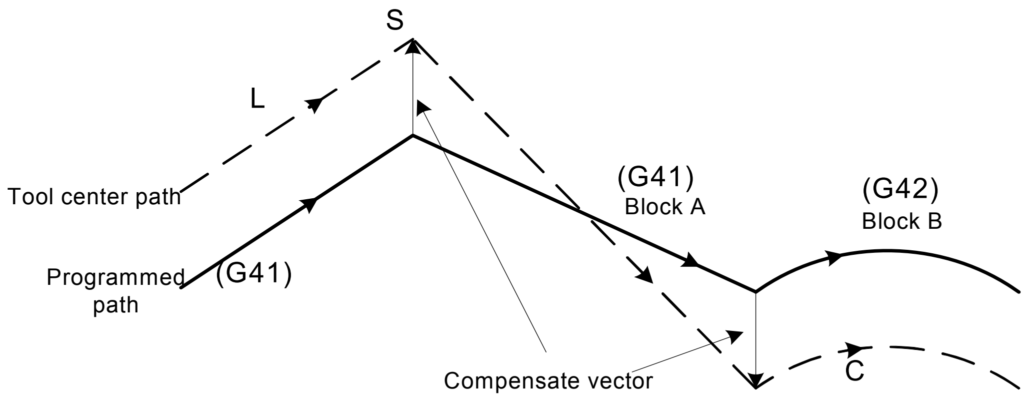


Fig. 8-2-2-3 linear—arc without intersection（change compensation direction）

8.2.3 Temporary Tool Compensation Cancel

In compensation mode, when the followinG Codes are specified, the compensation vector is temporarily cancelled, later, it automatically recovers.

At the moment, it is different from the compensation cancel mode, the tool directly traverses from the intersection point to the command point of compensation vector cancel. When the compensation mode recovers, the tool directly traverses to the intersection point.

1. Coordinate system setting（G50）

N1 T0101；

N2 G42 G00 X0 Z0；

N3 G01 U-30 W30；

N4 U30 W30；

N5 G50 X0 Z60；

N6 G01 U-30 W30；

N7 G01 U30 W30；

N8 G00 X0 Z0；

N8 M30；

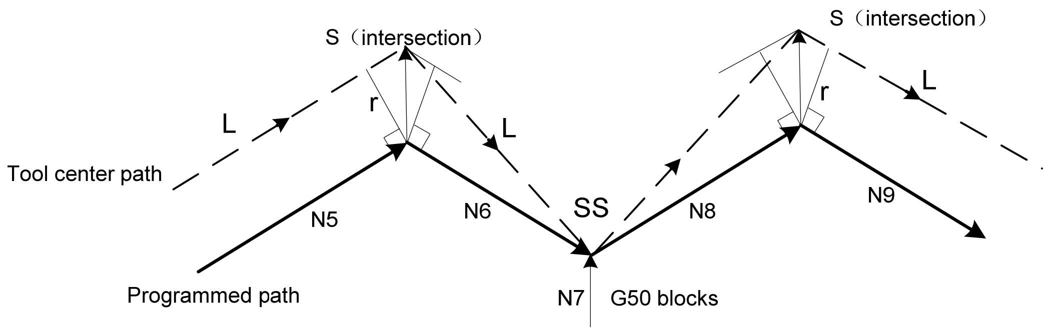


Fig. 8-2-3-1 temporarily cancel tool compensation①

Note: SS means to be the point where the tool stops twice in single block mode.

2. G90，G92，G94 are fixed cycle commands，G71～G76 are fixed cycle commands, G32/ G33/ G34 are thread machininG Codes.

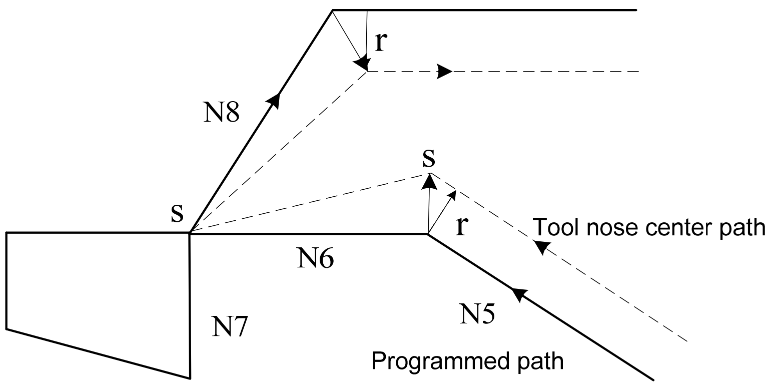


Fig. 8-2-3-2 temporarily cancel tool compensation②

N1 T0101

N2 G0 X100 Z100

N3 G0 X0 Z0

N4 G42 G90 X-20 W-50 F500 （Temporarily cancel tool compensation here）

N5 G0 X50 Z50 （Recover tool compensation ）

N6 G0 X100 Z100

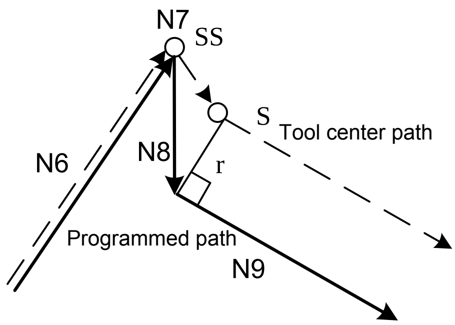
N7 M30

Note: G90/G94 is temporarily cancelled and is executed only when G41/G42 and G90/G94 are in the same block, the system automatically executes G90/G94 tool compensation when they are not in the same block. See G90/G94 tool nose radius compensation in detail.

8.2.4 Non-Movement Command when Tool Compensation

1. Non-movement command when starting compensation

The compensation vector is not executed when there are no tool traverse commands at the beginning of compensation commands.

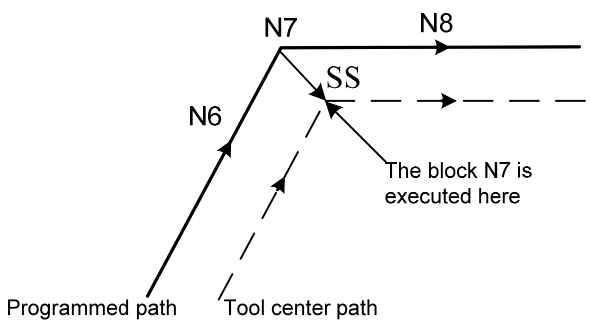


N1 T0101；
N2 G0 X0 Z0；
N3 G01 U-30 W20 F500；
N4 G42 U0；
N5 U30；
N6 U20 W20；
N7 G40 G0 X100 Z100；
N8 M30；

Fig. 8-2-4-1 non-movement command when tool compensation start

2. Non movement command when executing compensation

In compensation mode, only one block without tool traversing is commanded, the vector and tool center path is the same that the block is not commanded (refer to Section 8.2.1 Tool Compensation Proceeding), the block without tool traversing is executed at the point where a single block stops.

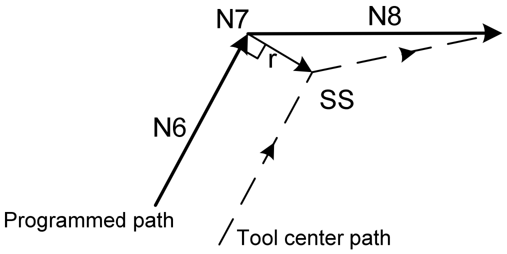


N3 T0101；
N4 G0 X100 Z100；
N5 G41 G01 X0 Z0；
N6 U-30 W20；
N7 G04 X5；
N8 W30；
N9 G40 G0 X100 Z100；
N10 M30；

Fig. 8-2-4-2 non-movement command when executing tool compensation

3. Non-movement command when tool compensation cancel

When the block which is commanded with the compensation cancel has no tool traversing, its formed length is the compensation value, its direction is vertical with the vector of the previous block movement, and the vector is cancelled at the next movement command.



N3 T0101；
N4 G0 X100 Z100；
N5 G41 G01 X0 Z0 F500；
N6 U-30 W20；
N7 G04；
N8 G0 X100 Z100；
N9 M30；

Fig. 8-2-4-3　non-movement command when tool compensation cancel

8.2.5　Tool Compensation Interference Check

The tool overcutting is called “interference”. The tool compensation interference check in advance the tool overcutting, namely, the interference check is executed even if the overcutting is not done.

(a) General conditions of interference

(1)　The directions of tool path and programmed path are different.(intersection angle of paths is between 90° and 270°).

(2)　When the arc is machined, the angle between starting point and end point of tool center path is great different from that of programmed path (more than 180°).

Example ①

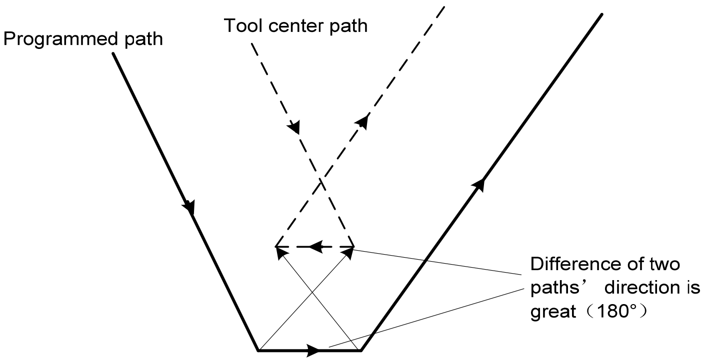
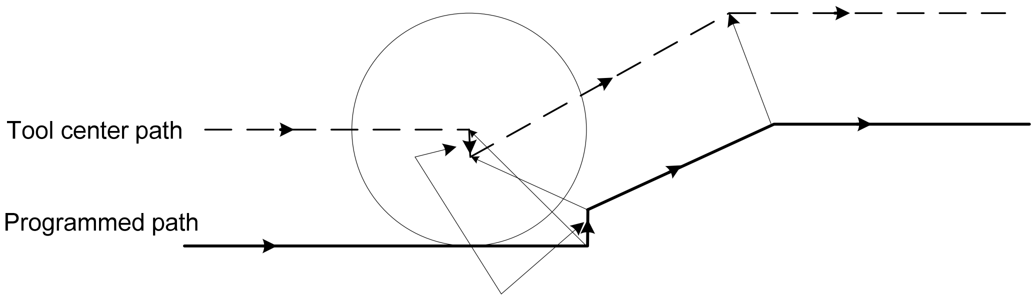


Fig. 8-2-5-1 tool compensation interference example ①



Difference of two paths’　direction is great（180°）

Fig. 8-2-5-2 tool compensation interference example ②

(b) Interference example

(1) One shallow depth, less than the compensation value.

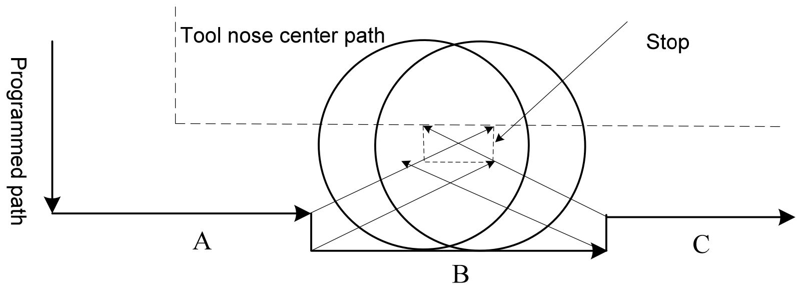


Fig. 8-2-5-3 tool compensation interference example ①

A program :

N1 T0101； (R<=10)

N2 G0 X0 Z30；

N3 G42 G01 X50 Z0 F500；

N4 U50；

N5 W20；

N6 U10；

N7 W20；

N8 U-10；

N9 W20；

N10 G40 G0 X0 Z30；

N11 M30；

In the above program, tool nose radius compensation value of No. 1 tool **R<=10**, the tool compensation is executed normally; **R>10**: the interference alarm occurs because the block B direction is reverse to the tool radius compensation's path.

(2) Grooving depth less than compensation value

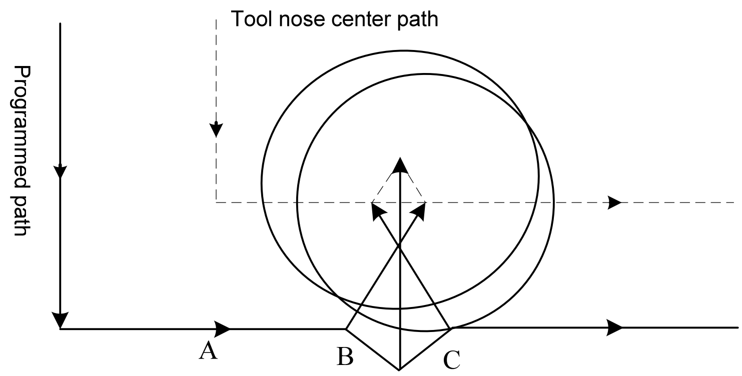


Fig. 8-2-5-4 tool compensation interference example ②

A program:

N1 T0101； (R<=25)

N2 G0 X0 Z30；

N3 G42 G01 X50 Z0 F500；

N4 U50；

N5 W20；

N6 U10 W10；

N7 U-10 W10；

N8 W20；

N9 G40 G0 X0 Z30；

N10 M30；

In the above program, tool nose radius compensation value of No. 1 tool **R<=25**, the tool compensation is executed normally; **R>25**: the interference alarm occurs because the block c direction is reverse to the tool radius compensation's path.

8.2.6 Tool Nose Radius Compensation in G90/G94

1. For all paths of cycle, the tool nose center path parallel with the programmed path.

① **G90**

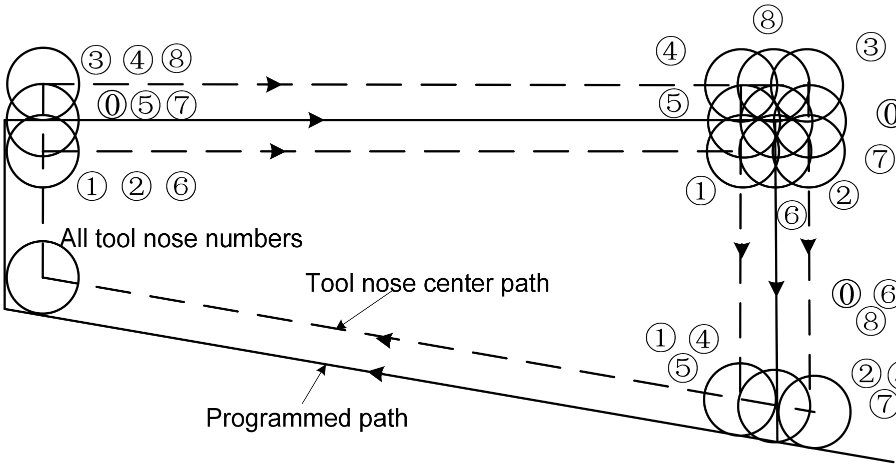


Fig. 8-2-6-1 **G90** tool nose radius compensation

② **G94**

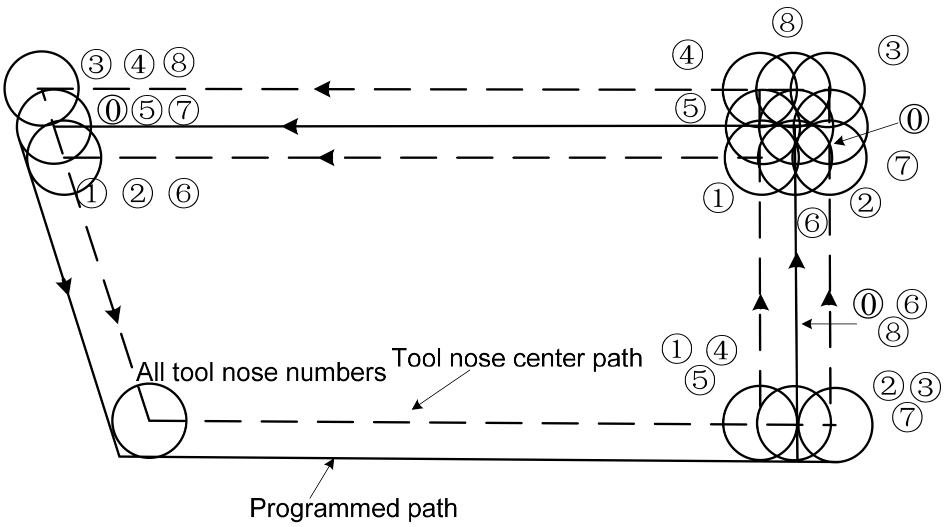


Fig. 8-2-6-2 G94 tool nose radius compensation

(b) In G41 or G42, the offset direction is shown below:

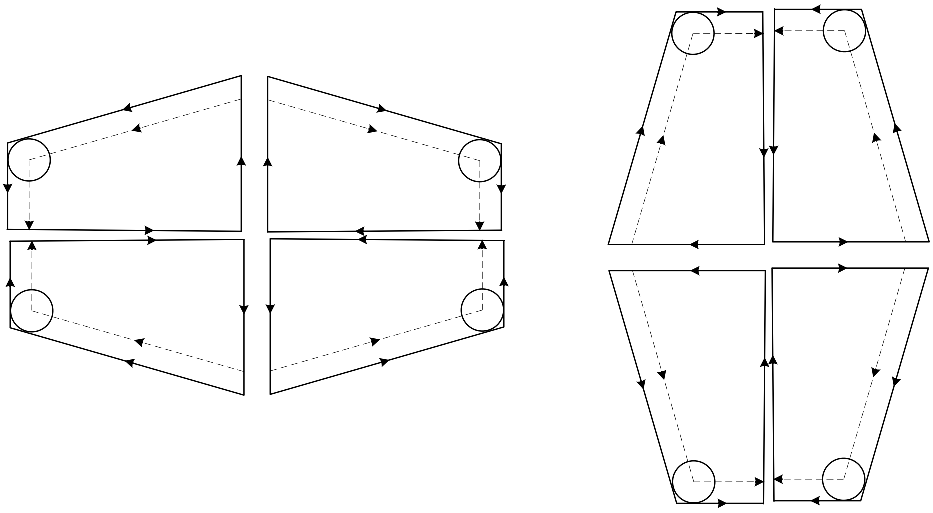


Fig. 8-2-6-3 G90/G94 tool nose radius compensation

8.2.7 Tool Nose Radius Compensation in G70

In G70 (finishing cycle), the tool nose radius compensation can be executed, and the tool center path automatically offsets one compensation value along finishing path. When G70 tool nose radius compensation is executed, G70 and G41/G42 are in the same block or G70 specifies G41/G42 in finishing cycle block.

8.3 Notes of Tool Compensation C

1. The system alarms (NO.39) when 30 non-movement blocks or more are specified continuously in the course of compensation:

N1 M05 ；.. M code output

N2 S21 ；... S code output

N3 G04 X10 ；.......................................….. Dwell

……

N29 G01 U0 ；.. Movement distance zero

N30 G98 ；.. only G code.

2. In MDI mode, blocks are executed but the tool nose radius compensation are not done.
3. After the system is turned on or when M30 is executed, the system enters into cancel mode. The program must end in cancel mode, otherwise, the tool cannot position in the end point, stops a position which is a vector from the end point.
4. Establishment or cancel of tool nose radius compensation only uses G00 or G01 instead of G02 or G03. When G02 or G03 is specified, the system alarms (NO.34).

5. Before the subprogram is called (i.e. before M98 is executed), the system must enters the compensation cancel mode. After the system enters subprograms, the offset is activated, but before the system returns to main programs (i.e. M99 is executed), it must be in the compensation cancel mode, otherwise, the system alarms. (No.36).
6. When compensation value (R) is negative, G4, G42 is exchanged each other. When the tool center traverses outside along the workpiece, it does inside, and vice versa. Because the compensation sign is changed, the tool nose offset direction also changes, but the imaginary tool nose direction does not change, so the compensation value sign should be changed freely.
7. When the mode is cancelled or the tool change is executed, the compensation value is changed. When the compensation value is changed in compensation mode, the new compensation is value after the tool change is executed.
8. When the tool compensation programs are execute, errors or alarms occur, but G code remains unchanged, at the moment, pressing reset key directly cancel the tool compensation C state.
9. G33/G34 does not execute the tool nose radius compensation.

8.4 Tool Compensation C Example

Tool compensation C example ①：

Machine the workpiece shown in Fig. 8-4-1, the tool nose radius R=1 and the tool is the first.

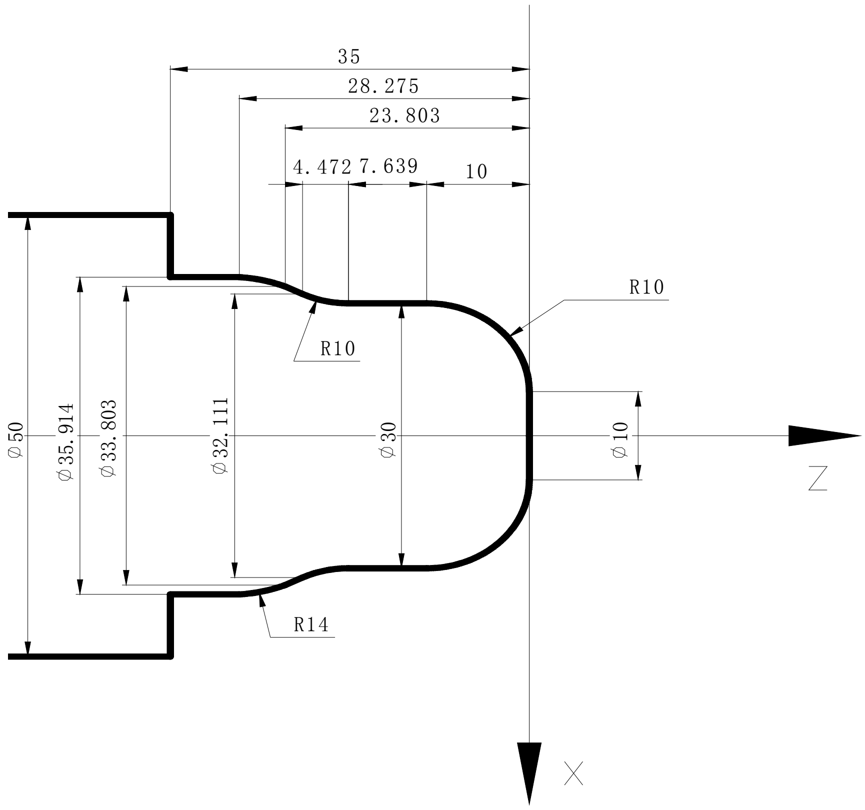


Fig. 8-4-1 tool compensation C example ①

A program：

```
O0001；
N010 G0 X100.0 Z100.0；         （Positioning to safety position）
N020 M3 S300；                  （Spindle rotation CCW, speed: 300r/min）
N030 M8；                       （Cooling ON）
N040 T0101；                    （Change No. 1 tool, execute its tool compensation）
N050 G00 X10.0 Z10.0；          （rapid positioning, approach workpiece）
N060 G42 G1 Z0 F80；            （Start executing tool nose radius compensation）
N070 G3 X30 Z-10 R10；
N080 G1 Z-17.639；
N090 G2 X32.111 Z-22.111 R10；
N100 G1 X33.803 Z-23.803；
N110 G3 X35.914 Z-28.275 R14；
N120 G1 Z-35；
N130 X50；
N140 G40 G0 X80 Z80；                （Cancel tool nose radius compensation）
N150 M09；                           （Cooling OFF）
N160 G00 X100.0 Z100.0 T0200；       （Rapid return to safety postiion, change reference tool,
                                      clear tool offset）
N170 M30；                           （End of program）
```

Tool compensation C example ②：

Machine the workpiece shown in Fig. 8-4-2, the tool nose radius R=1 and the tool is the first.

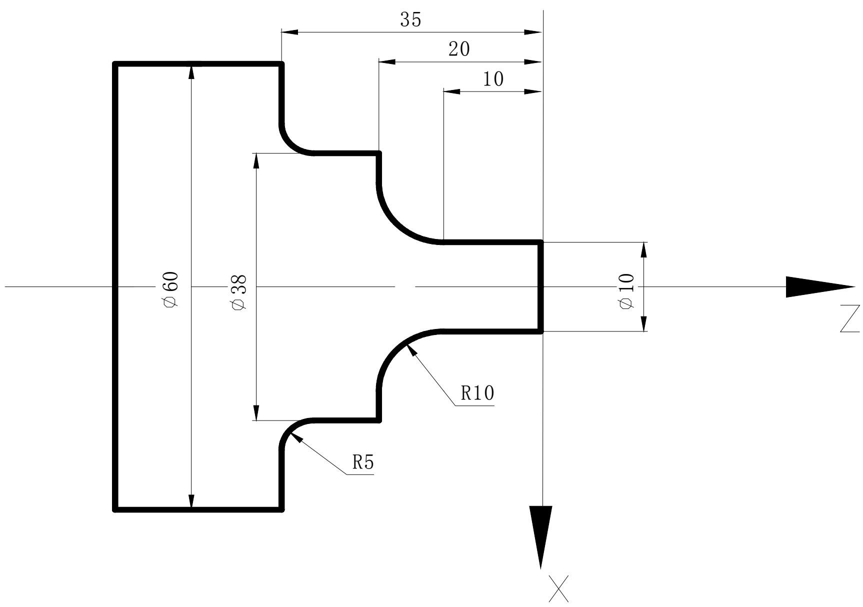


Fig. 8-4-2 tool compensation C example ②

```
O0002
NI G0 X100 Z100；
N2 M3 S800；
N3 M8；
N4 T0202；
N5 G0 X70 Z10；
N6 G71 U3 R1；
N7 G71 P8 Q14 U0 W0 F120；
N8 G0 X10；
N9 G1 Z-10 F80；
N10 G02 X30 W-10 R10；
N11 G1 X38；
N12 Z-30；
N13 G02 X48 W-5 R5；
N14 G1 X60；
N15 G0 X100 Z80；
N16 M3 S300；
N17 T0101；
N18 G0 X70 Z10；
N19 G42 G70 P8 Q14；      (G42 and G70 are in the same block to execute the tool nose radius
                           compensation)
N20 G40 G0 X80 Z50；       (Cancel tool nose radius compensation)
N21 G0 X100 Z100 T0200；
N22 M30；
```

Tool compensation C example ③：

Machine the workpiece shown in Fig. 8-4-3, the tool nose radius R=1 and the tool is the first.

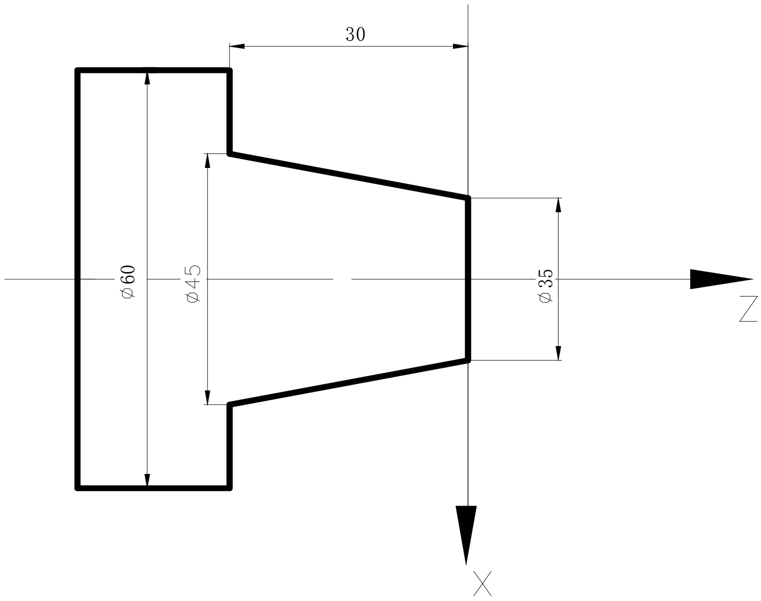


Fig. 8-4-3　tool compensation C example ③

O0003

N1 G0 X100 Z100；

N2 M3 S800；

N3 M8；

N4 T0101；

N5 G42 G0 X70 Z10；　　（Start executing tool nose radius compensation）

N6 G90 X45 Z-30 R-5 F80；

N7 G40 G0 X80 Z80；　　(Cancel tool nose radius compensation)

N8 G0 X100 Z100 T0100；

N9 M30；

PART THREE　OPERATION

CHAPTER ONE OPERATION PANEL

1.1 Panel division

GSK980TA3 systems use integrated panel including LCD (liquid crystal display), Edit keypad and machine control display sections as shown in the figure bellow:

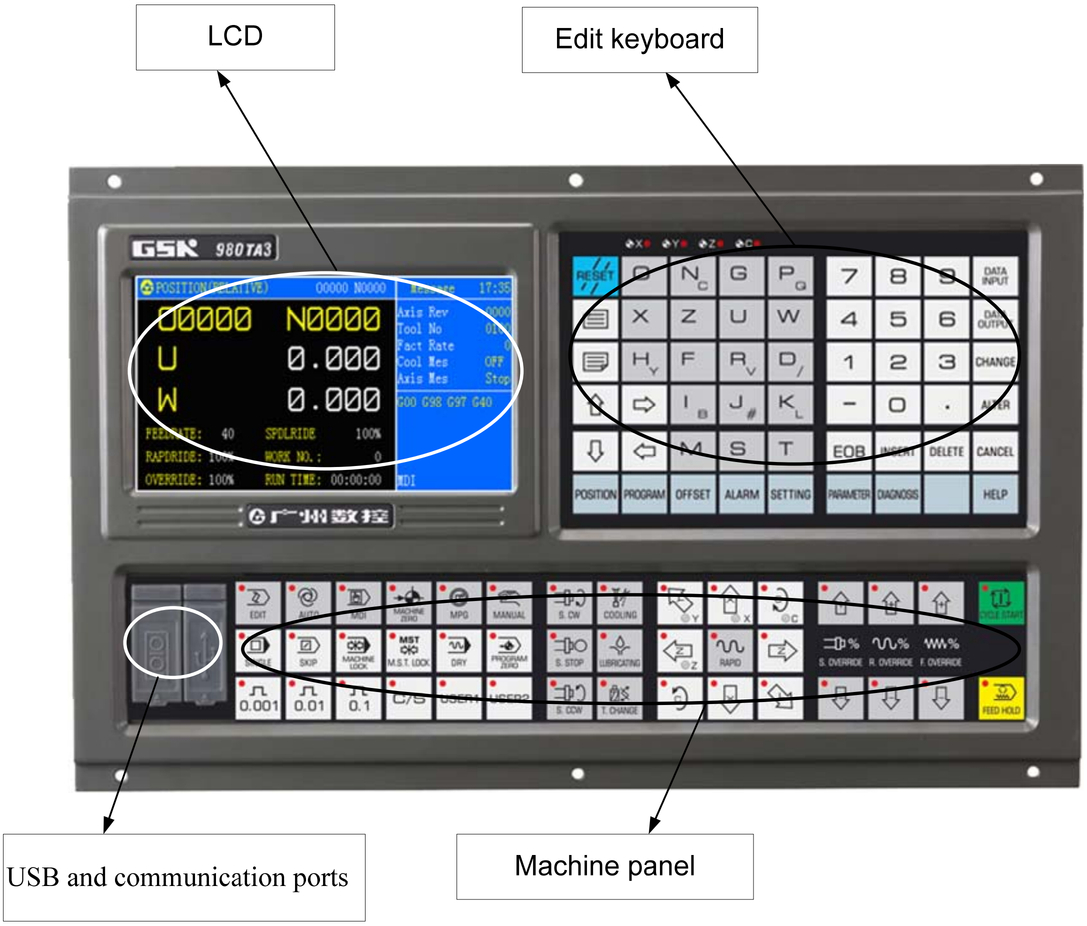


Fig.1-1-1 GSK980TA3

1.2 Instructions for Panel Functions

1.2.1 LCD (liquid crystal) display area

The display area of the system adopts color 7-inch liquid crystal display.

1.2.2 Edit Keypad Area

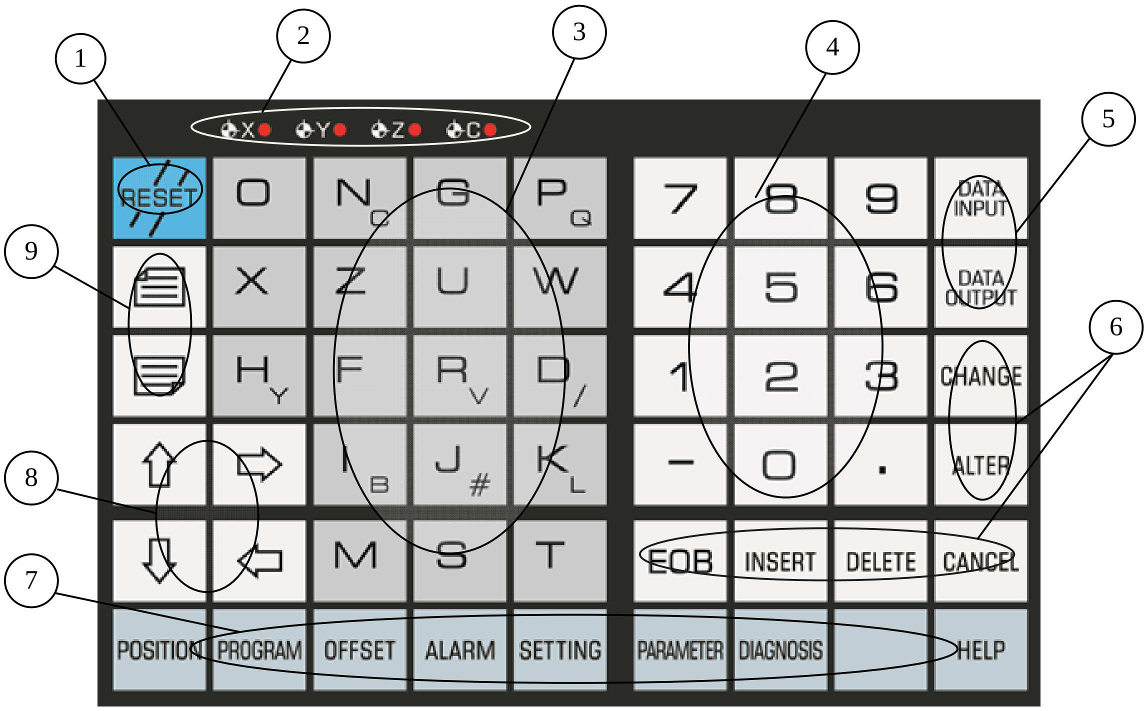


Fig. 1-2-2-1 GSK980TA3

The edit keypad area is further divided into 9 parts. Their detailed usages are as follows:

No.	Name	Function explanation
1	Reset key	The CNC reset, feed and output stop etc.
2	Indicator	Function indicator
3	Address key	Address input
4	Numerical key	Digit input
5	I/O key	For parameter, offset input. The files input and output by RS232 interface.
6	Functional key	Program insertion, alteration and deletion in program edit

7	Screen operation key	Press any key to enter corresponding interface (details are shown below)
8	Cursor moving key	Cursor moving up, down, left or right
9	Page key	For page turning in an interface

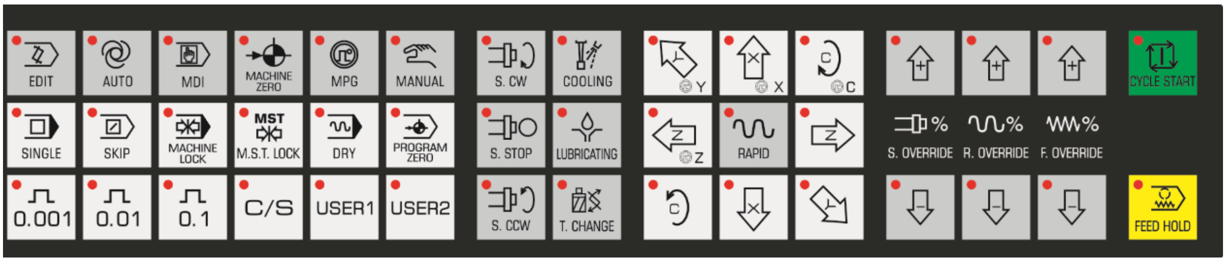
1.2.3　Introduction for Screen Operation Keys

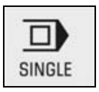
There are 8 page display keys on the operation panel of the system 980TA3, see the following figure:

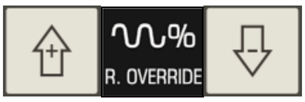
Name	Function Explanation	Remarks
Position	To enter the position interface	The relative coordinate, absolute coordinate, integrated coordinate and the position/program interfaces of the current point can be switched by pressing the page key.
Program	To enter the program interface	The program, MDI and catalogue interfaces can be switched by pressing the page key.
Offset	To enter the offset interface	The contents of number 001~064 and 101~164 can be switched by pressing this key continuously
Alarm	To enter the alarm interface	The interfaces of the alarm message and the external message can be switched by the page key.
Setting	To enter the setting interface	The setting and graph interfaces can be switched by pressing this key continuously
Parameter	To enter the parameter interface	The parameter and screw-pitch compensation interfaces can be switched by pressing this key continuously
Diagnosis	To enter the diagnosis interface	The diagnosis, machine panel and macro variable interfaces can be switched by pressing this key continuously

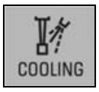
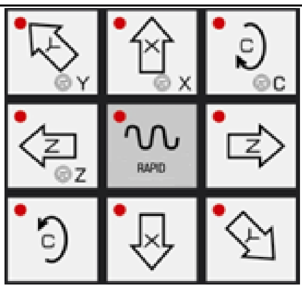
1.2.4　Machine Control Area

1.2.4.1　Machine Control Area of the 980TA3



Key	Name	Function explanations	Remarks and operation instructions
	Edit mode key	To enter Edit mode	Switch to Edit mode from Auto mode. CNC decelerates to stop after present block is finished.
	Auto mode key	To enter Auto mode	In Auto mode, CNC selects internal memory program
	MDI mode key	To enter MDI mode	Switch to MID mode from Auto mode. CNC decelerates to stop after present block is finished
	Machine zero mode	To enter Machine zero mode	After Auto mode is switched to Machine zero mode, the system immediately decelerates to stop
	MPG	To enter MPG mode	After Auto mode is switched to MPG mode, the system immediately decelerates to stop
	Manual	Manual mode key	After Auto mode is switched to, the system immediately decelerates to stop
	Single block	For switching of block/ blocks. Single block indicator lights up if Single mode is active	This mode is active in Auto mode and MDI mode

	Block Skip key	For skipping of block headed with“/”sign. If the indicator lights up, the block is skipped	This mode is active in Auto mode and MDI mode
	Dry run	If dry run is active, the indicator lights up	This mode is active in Auto mode and MDI mode
	M.S.T Lock key	If the M.S.T. lock is set for ON, the indicator lights up. and M,S,T function output is inactive	This mode is active in Auto mode and MDI mode
	Machine lock key	If the Machine lock is set for ON, the indicator lights up. and axis output is inactive	This mode is active in Auto mode, MDI, Machine zero, MPG, Manual mode
	Spindle control keys	Spindle CW Spindle stop Spindle CCW	In Manual mode, they are active when the spindle is in analog control mode
	Spindle override key	For spindle speed adjustment (It is active in spindle analog control)	Any mode
	Rapid override	Rapid override key	This mode is active in Auto mode, MDI, Machine zero, MPG, Manual mode
	Feedrate override key	For adjustment of the federate	This mode is active in Auto mode, MDI, Machine zero, MPG, Manual mode
	Manual step, MPG override selection key	For selection of manual step, MPG override.	This mode is active in MPG and Manual mode
	MPG axis selection key	Corresponding axis selection for manual pulse	It is active in MPG mode

		generator (MPG)	
	Lubricating on/off key	For lubricating on/off	This mode is active in Auto mode, MDI, MPG, Manual mode
	Cooling on/off key	For cooling on/off	This mode is active in Auto mode, MDI, MPG, Manual mode
	Work light on/off key	For work light on/off	This mode is active in MPG, Manual mode
	Manual feed key Rapid traverse	For positive/negative moving of X, Y, Z axes in manual mode.Rapid traverse on/off	This mode is active in machine zero, manual mode, MPG Any mode
	Feed hold key	Auto operation of the system is stopped by pressing this key	This mode is active in Auto mode, MDI mode
	Cycle start key	The system operates automatically by pressing this key	This mode is active in Auto mode, MDI mode

CHAPTER TWO SYSTEM POWER ON/OFF & SAFETY OPERATION

2.1 System Power-on

Before this GSK980TA3 is powered on, confirm that:

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

The current position (Absolute POS) is displayed after the self-check and initialization are finished.

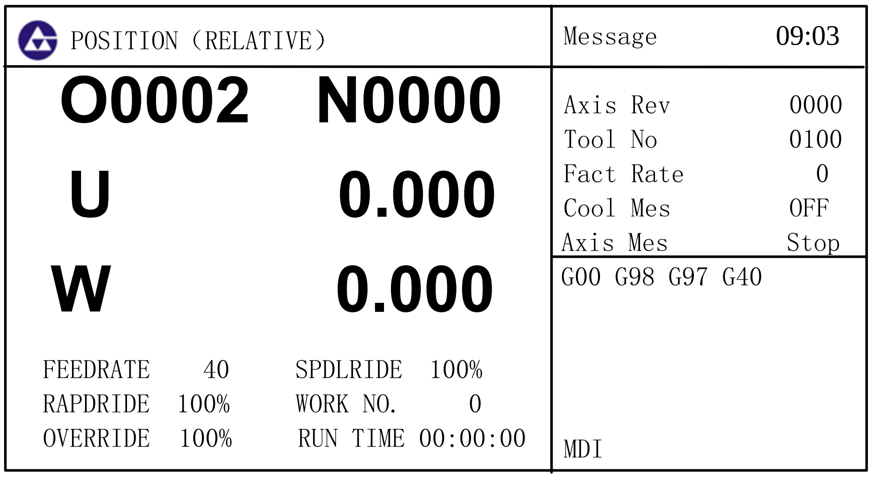


Fig.2-1-1

2.2 System Power-off

Before power is off, confirm that:

1. X, Z axes of the CNC are stopped;
2. Miscellaneous functions (spindle, pump etc.) are off;
3. Cut off CNC power prior to machine power cutting off.

Before power is off, check that:

1. LED indicating cycle start of the panel is in a halted state;
2. All movable parts on CNC machine are in a halted state;
3. CNC power is cut off by pressing POWER OFF button.

Emergency Power-off

Under the emergencies during the machine operation, the machine power should be cut off immediately to avoid the incidents. However, it should be noted that there may be an error between the system displayed coordinate and the actual position. Therefore, the machine zero and tool setting operations should be performed again.

Note: Please see the machine builder's manual for the machine power-off operation.

2.3 Safety Operation

2.3.1 Reset

Press 

key to the reset CNC if there are abnormal output and axis action about it:

1．All axes movement stops

2．Auto operation ends, modal function and state are held on.

3．By modifying parameter **N0:02#6**, it is able to set offset vector is unchanged/ deleted during reset.

4．By modifying parameter **N0:12#0**, it is able to set whether M03, M04, lubricating and cooling is off or not after reset.

5．By modifying parameter **N0:13#1**, it is able to set whether the macro common variable is deleted/ unchanged during reset.

6．By modifying parameter **N0:13#3**，it is able to set whether the expansion output IO port is off or not during reset.

2.3.2 Emergency Stop

By pressing the emergency stop button in machine operation, the movement of the machine such as spindle rotation, cooling, etc. are cut off immediately. Release the emergency stop button to leave the emergency stop state.

Note 1: Make sure that the troubles are removed before the emergency button is eliminated.

Note 2: After the button is released, reference return operation should be done to ensure the correctness of the coordinate position.

Note 3: Electric shock can be reduced by pressing down the emergency stop button before the system power ON/OFF. Emergency stop signal is a normally-closed contact signal. When the contact is disconnected, the system enters emergency stop state, and the machine stops.

2.3.3 Feed Hold

key can be pressed to make the running pause during the machine operation. It calls for special notice that the feed hold state can be entered only when the next block is not thread cutting or cycle command operation after the current command is performed in these operations.

2.3.4 Power-off

Under the emergencies during the machine operation, the machine power should be cut off immediately to avoid the accidents. However, it should be noted that there may be an error between the system displayed coordinate and the actual position. Therefore, the tool setting operation should be performed again.

2.4 Cycle Start and Feed Hold

Cycle start and feed hold keys on panel are used for program start and pause operation in Auto and MDI mode.

The additional panel can connect to the cycle start and feed hold keys.

By modifying parameter **NO:14#0, NO:14#1,** it is able to set whether the external start and pause are shielded.

2.5 Overtravel Protection

Overtravel protection should be employed to prevent the damage to the machine due to overtravel of the feed axis.

2.5.1 Hardware Overtravel Protection

The stroke limit switches are fixed at the positive and negative maximum stroke position of X and Z axes respectively. If the overtravel occurs, the running axis decelerates to eventually stop when it contacts with the stroke switch, and the emergency alarm is issued.

Detailed explanations

Once the workbench presses the limit switch during the operation, the system decelerates immediately to stop all operation of axes. The control signals of spindle, lubricating and cooling are off and overtravel alarm occurs.

Steps for eliminating the hardware overtravel alarm

1. Press and hold the overtravel release button, now the system alarm is eliminated.
2. Move the carriage at the opposite side (move out negatively for positive overtravel, vice versa) to leave the limit switch.
3. Release the overtravel limit button.

2.5.2 Software Overtravel Protection

Software stroke range is set by parameter P46~ P49 on the data parameter (appendix 1). Set coordinate value of the machine as reference value. If the machine position (machine coordinate) exceeds software stroke range, overtravel alarm will occur. When the overtravel alarm occurs, move the axis reversely in the manual mode and then press the reset key to eliminate the alarm.

CHAPTER THREE INTERFACE DISPLAY & DATA MODIFICATION & SETTING

3.1 Position Display

3.1.1 Four Ways for Interface Display

Press 

key to enter Position Interface. There are four modes in this interface such as 【REL.】,【ABS】,【COM.】,【POSC】. They can be viewed by pressing  or  key. Detailed information of each interface is as follows:

1) Relative POS.: It displays the current tool position in relative coordinate system (see Fig. 3-1-1-1)

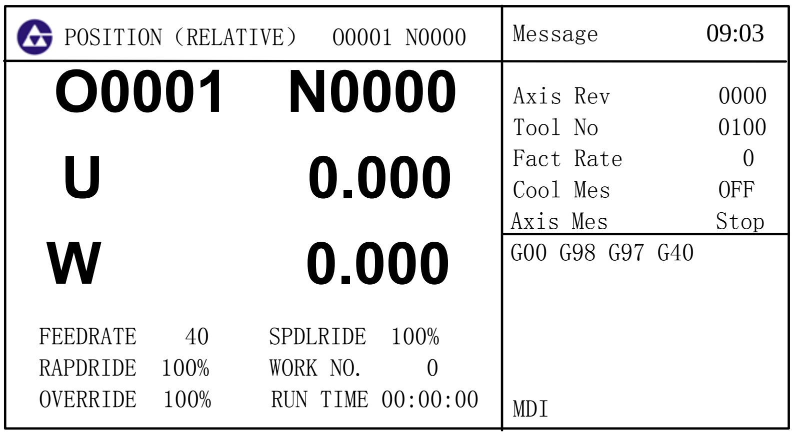


Fig. 3-1-1-1

Steps for clearing the relative coordinate system: Coordinate system U axis is blinking by pressing 

key, and then press 

key to cancel the coordinate system. The operation of the W axis is the same as that of the U axis.

Note: The relative POS. will be cleared automatically at power off.

2) Absolute POS.: It displays the current tool position in absolute coordinate system (see Fig. 3-1-1-2)

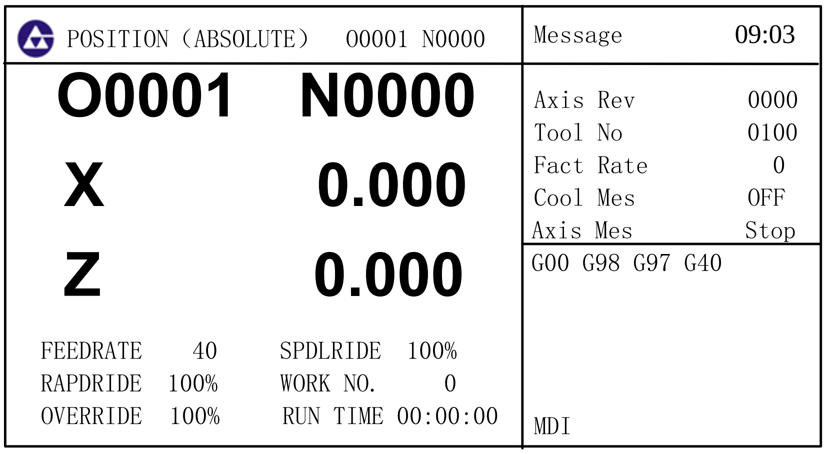


Fig. 3-1-1-2

3）Integrated interface: The following coordinate position values are displayed:

(A) Position on relative coordinate system

(B) Position on absolute coordinate system

(C) Position on machine coordinate system

(D) Distance-to-go (Display in Auto and MDI mode)

Display page is as follows (Fig.3-1-1-3):

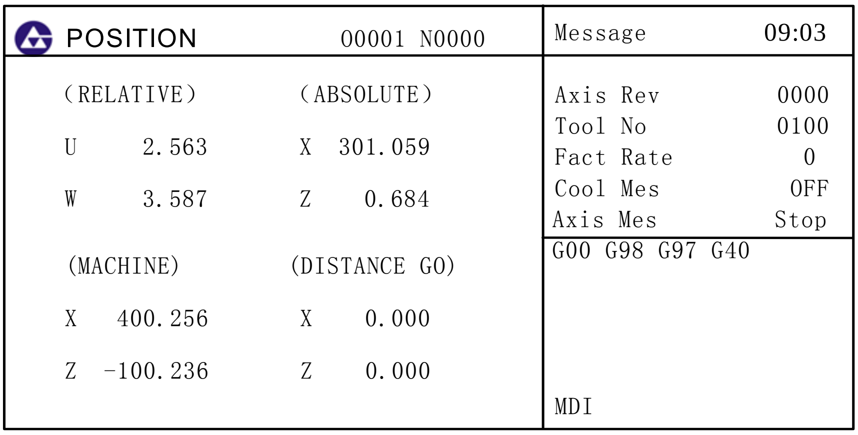


Fig. 3-1-1-3

4） Program monitoring mode: In this interface, relative coordinate and absolute coordinate of the current position and the current block can be displayed at the same time.

```
POSITION                 O0001 N0000  | Message          09:04
                                      |
  (RELATIVE)        (ABSOLUTE)        | Axis Rev          0000
                                      | Tool No           0100
  U     2.563       X   301.059       | Fact Rate            0
                                      | Cool Mes           OFF
  W     3.587       Z     0.684       | Axis Mes          Stop
 O0001;                               | G00 G98 G97 G40
G50 X250. Z450.;                      |
T0101;                                |
G00 X100. Z200.;                      |
G90 U-10. W-50. R-1.5  F500.;         |
                                      | MDI
```

Fig. 3-1-1-4

3.1.2 Display of Machining Time, Parts Number, Programming Speed, Override and Actual Speed

In absolute and relative interfaces of the POS., programming rate, rapid override, federate override, spindle override, parts number, cutting time can be displayed. Spindle speed, tool number, cooling state, spindle state, G function code, operation mode, the current date and time etc. can be displayed in state message column. See the Fig. 3-1-2-1.

```
POSITION (RELATIVE)                   | Message          09:03
  O0002      N0000                    | Axis Rev          0000
                                      | Tool No           0100
  U              0.000                | Fact Rate            0
                                      | Cool Mes           OFF
  W              0.000                | Axis Mes          Stop
                                      | G00 G98 G97 G40
FEEDRATE     40    SPDLRIDE  100%     |
RAPDRIDE   100%    WORK NO.     0     |
OVERRIDE   100%    RUN TIME 00:00:00  | MDI
```

Fig. 3-1-2-1

Detailed significances are as follows:

Programming rate: the rate specified by F code.

Rapid override: current rapid override of the system.

Feedrate override: current federate override of the system.

Spindle override: Spindle speed override for adjusting spindle speed.

Parts number: add 1 when M30 (or M99 in the main program) is executed.

Cutting time: It begins timing when auto operation is started. Unit: hour, minute, second.

Spindle speed: actual speed from the spindle encoder.

Tool number: tool number and tool offset number of the current tool.

Actual speed: actual processing speed after the magnification is multiplied.

Cooling state: cooling on/off state.

Spindle state: spindle CCW/STOP/CW.

G function code: modal G code of the current system.

3.1.3 Clearing Methods of Parts Number and Cutting Time

1）Switch to the position interface.

2）By pressing T key, parts number and cutting time on the interface are blinking. Then press CANCEL key, parts number and cutting time are cancelled. If user presses other keys while they are blinking, the value is not change.

Note 1: The encoder shall be installed on the spindle for displaying the actual spindle speed.

Note 2: Actual rate = F value of the programming rate × magnification. In G00, speed of each axis is set by data parameter P21～P22. It can be adjusted by the rapid override. The upper cutting speed is set by data parameter P27. If it exceeds the upper value, the system will alarm.

Note 3: By setting the parameter NO:3#6, it is able to set whether the power-off memory is active for the parts number and the cutting time.

3.2 Program Display

Press PROGRAM key to enter program display interface. There are three modes in this interface such as【PROG.】,【MDI input】,【PRG content】. They can be viewed by pressing or key. Detailed information of each interface is as follows:

1）Program display: the current program block in memory unit is displayed in this interface (see Fig. 3-2-1)

PROGRAM 00001 N0000	Message 09:04
00001;	
G50 X250. Z450.;	Axis Rev 0000
T0101;	Tool No 0100
G00 X100. Z200.;	Fact Rate 0
G90 U-10. W-50. R-1.5 F500.;	Cool Mes OFF
G90 U-50. W-10. R-3. F350.;	Axis Mes Stop
G00 X120. ;	G00 G98 G97 G40
T0202;	
M03 S01;	
G92 X80. W-65. R5.1 F0.5;	
ADRS Row 2	EDIT

Fig. 3-2-1

The detailed program edit, please see the chapter 10 operation of editing on operation manual. Only in program display mode of the edit mode, all contents of the current program can be viewed and edited by pressing  or  key.

2） **MDI** input display: multi-block can be edited and executed in the edit mode. Its program format is the same as that of the edit mode.

MDI operation applies to simple testing program (see fig. 3-2-2).

PROGRAM			00001 N0000	Message 10:04
X				Axis Rev 0000
Z	G94	F	1000.000	Tool No 0100
U	G97	M		Fact Rate 0
W	G98	S	0	Cool Mes OFF
R		T	0100	Axis Mes Stop
F				G00 G98 G97 G40
M	G21			
S	G40			
T				
P				
Q		SACT	0000	
ADRS				MDI

Fig. 3-2-2

3) The program (catalogue) displays catalogue display interface. See fig. 3-2-3 for its contents.

(a) System version number: current version of the system used.

(b) Number of the stored programs: number of the stored programs (including subprogram)

Available space: the space can be used.

(c) Used storage capacity: the space occupied by the stored programs (displayed by character number).

Available space: the space can be used.

(d) Program catalogue: the numbers of the stored programs are displayed in order. The programs are displayed in the order of the size of the program name.

```
PROGRAM                           O0001 | Message           10:04
N0000                                   | Axis Rev           0000
  SYSTEM EDIT:GSK-980TA3(08) V3.74.083  | Tool No            0100
  PRG.NO.USED:     13      FREE:687     | Fact Rate             0
  MEMORY USED:     704     FREE: 23840  | Cool Mes            OFF
  PRG.LIBRARY:                          | Axis Mes           Stop
 O0000 O0001 O0002 O0003 O0004 O0005    | G00 G98 G97 G40
 O0006 O0007 O0008 O0009 O0010 O0011    |
 O0045                                  |
                                        | MDI
 ADRS
```

Fig. 3-2-3

3.3 Display, Modification and Setting of the Tool Offset

3.3.1 Tool Offset Display

Press OFFSET key to enter offset Interface. There are numbers 001~064 and101~164 two parts in this interface. They can be viewed by pressing OFFSET key continuously. Detailed information is as follows:

1）Offset interface   The offset interface is as follows (see fig. 3-3-1-1):

```
OFFSET（mm）              O0001 N0000 | Message           10:09
  No.     X       Z       R      T    | Axis Rev           0000
  000   0.000   0.000   0.000    0    | Tool No            0100
  001   0.000   0.000   0.000    0    | Fact Rate             0
  002   0.000   0.000   0.000    0    | Cool Mes            OFF
  003   0.000   0.000   0.000    0    | Axis Mes           Stop
  004   0.000   0.000   0.000    0    | G00 G98 G97 G40
  005   0.000   0.000   0.000    0    |
  006   0.000   0.000   0.000    0    |
  007   0.000   0.000   0.000    0    |
ABSO   X  100.000       Z  100.000    |
RELA   U  -169.380      W  122.337    |
ADRS                                  | MDI
```

Fig. 3-3-1-1

There are numbers 001~064 and101~164 two parts in tool offset interface. Number 001~064 are used to modify tool offset value, while number 101~164 are used to set tool offset value (see Tool Setting for detailed functions). Number 000 and 100 are used to move work piece coordinate in parallel direction. X and Z are tool offset value from the tool post center to the tool nose along the direction of X, Z axis respectively. R is the radius compensation value of the imaginary tool nose. T is the number of the imaginary tool nose (See C Tool Offset Function Instructions for detailed functions).

3.3.2 Tool Offset Modification and Setting

In the offset interface, methods of tool offset setting are as follows:

1) Press OFFSET to enter the offset interface.
2) Move the cursor to the position where the offset number will be input.

Method 1: Press the page key to enter the page where the offset value to be modified locates. By pressing the direction keys, the cursor is moved to the position where the offset to be modified locates.

Method 2: Press P Q key to input the number to be altered. Then press DATA INPUT key, the cursor moves to the input number directly. Finally press CANCEL key to return to the state of the data input.

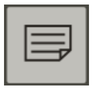
3) Press X or Z key to select the tool offset axis, an input the offset value. After pressing DATA INPUT key, the system will input the offset value automatically, and it will be displayed on the LCD screen.

Note 1: The offset value can not be set in auto operation.

Note 2: When you adjust the tool offset value in the range 001～064, use U, W to input the adjusting value. When tool setting is performed in the range 101～164, input actual measurement value of the X, Z respectively. If the add and subtraction of the current offset value is required, press U key for the X axis, and press W key for Y axis, then, input the value to be added or subtracted, finally press DATA INPUT key. For example, the offset value of Z axis is 3, then input W5 to get Z axis offset value 3+5=8.

3.4 Alarm Display

Press 

key to enter alarm Interface. There are two modes in this interface such as 【alarm signal】,【external information】. They can be viewed by pressing  or  key. Detailed information of each interface is as follows:

1. For the alarm interface, see Fig. 3-4-1:

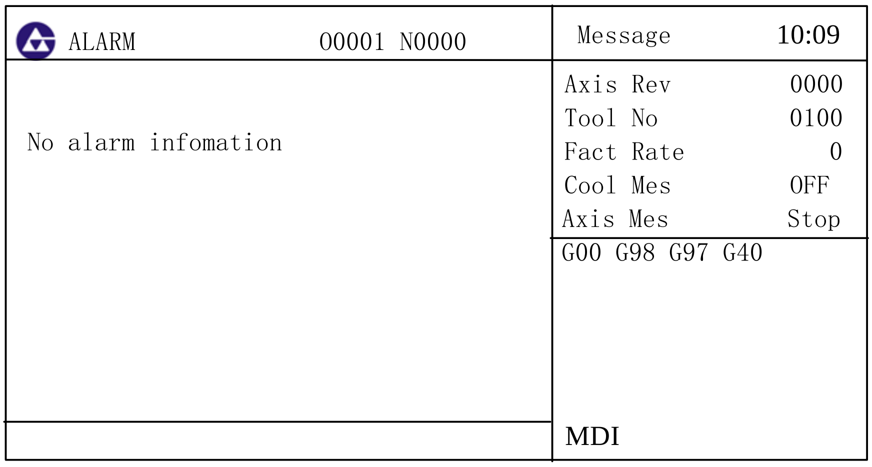


Fig. 3-4-1

The system will enter the alarm interface automatically when troubles occur. The current P/S alarm number and detailed alarm contents are displayed in the alarm interface, see appendix 2 for the contents.

2. For the external information interface, see Fig. 3-4-2:

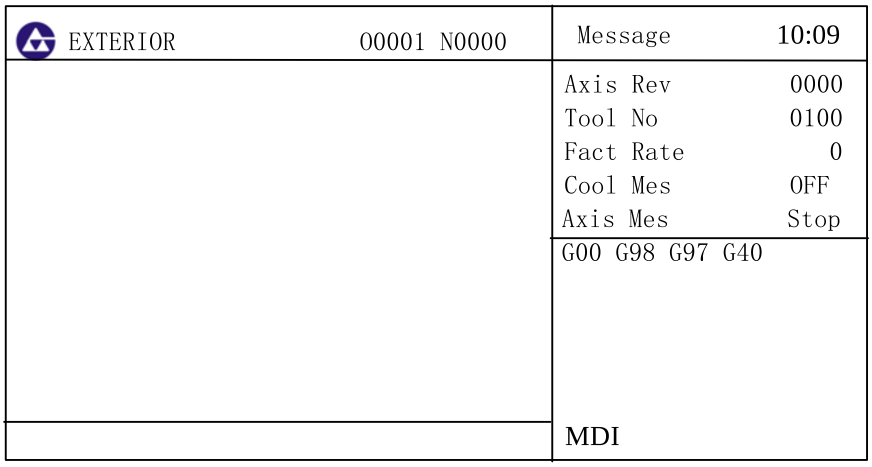
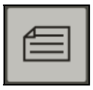


Fig. 3-4-2

3.5 Setting Display

Press  key to enter the setting interface that includes 【Setting】,【Graph】two types of display modes. They can be viewed and modified by pressing  key repeatedly.

3.5.1 Instruction for 【Setting】Interface Operation

【Setting】interface includes two interfaces 【Setting】 and 【User Password】, which can be viewed by pressing  or  key.

1. See the following figure for the setting interface (see 3-5-1-1):

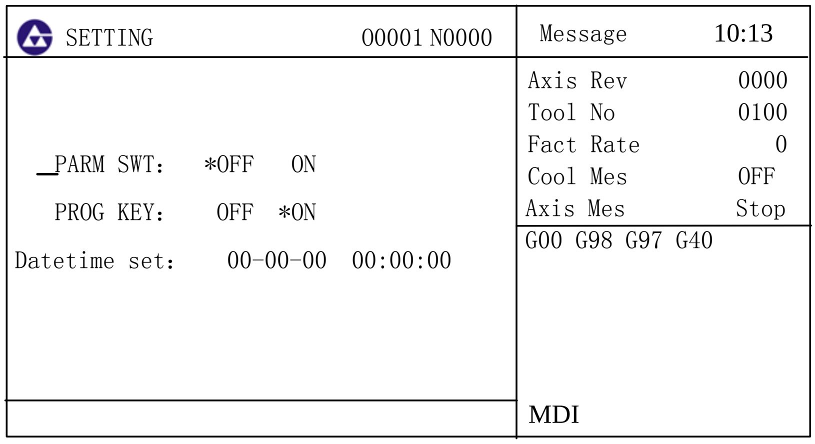
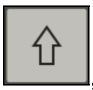
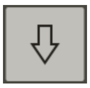


Fig. 3-5-1-1

Operating instructions:

On the above figure, the parameter switch and the program switch ON/OFF and time can be set. The detailed operation methods and steps are as follows:

（a） Move the cursor to the position of parameter switch, program switch or time setting by pressing ,  key.

（b） Press  key to set parameter switch or program switch to ON. At this time, parameter modification, setting and program editing can be performed. The system displays number P/S100 alarm when the parameter switch is set to ON. Meanwhile, input

Part Three Operation

parameter and then return to the setting interface. Press W key to set parameter switch or program switch to OFF, and the alarm is eliminated automatically after the parameter switch is off.

（c） In <MDI mode>, move the cursor to time setting, and key in data, time by sequence. Then press DATA INPUT key to confirm it, if input data is wrong, input it again or press CANCEL key to cancel it word by word from the back.

2. See the following figure for the user password interface (see fig3-5-1-2)

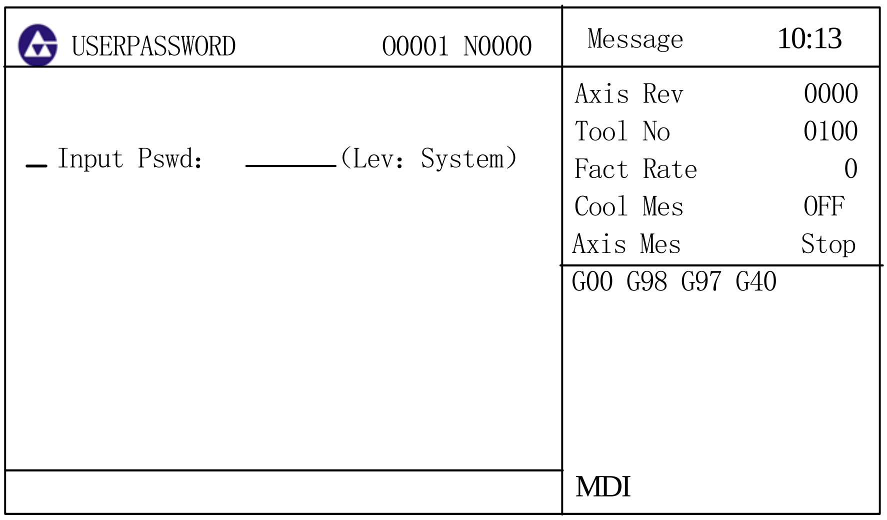


Fig. 3-5-1-2

Operating instructions:

（a）Enter this interface in MDI mode, press CHANGE key to select the password level. There are 4 levels, which are user, system, manufacturer and LMT.

（b）After selecting the level, input level password and press DATA INPUT key to do confirmation. If the operation is right, it enters to password modification interface and the password can be modified. The password consists of 1~6 numbers. If the current level is User or LMT, the system prompts: password is correct. you can modify data parameter and screw-pitch compensation data. Otherwise, it prompts: the input password is wrong. When the current level is manufacturer or system, the system prompts: password is correct. You can modify data parameter and screw-pitch compensation data. Otherwise, it prompts: the input password is wrong.

Note: The password is not always numbers, it includes “-” and “.”.

（c）After modification, input the data again if wrong password is input or original input password is inactive.

3.5.2 Operating Instructions for 【Graph】Interface

【Graph】interface includes two interfaces 【Graph】 and 【Drafting】, which can be viewed by pressing  or  key.

1. See the following figure for the 【Graph】 interface (see 3-5-2-1):

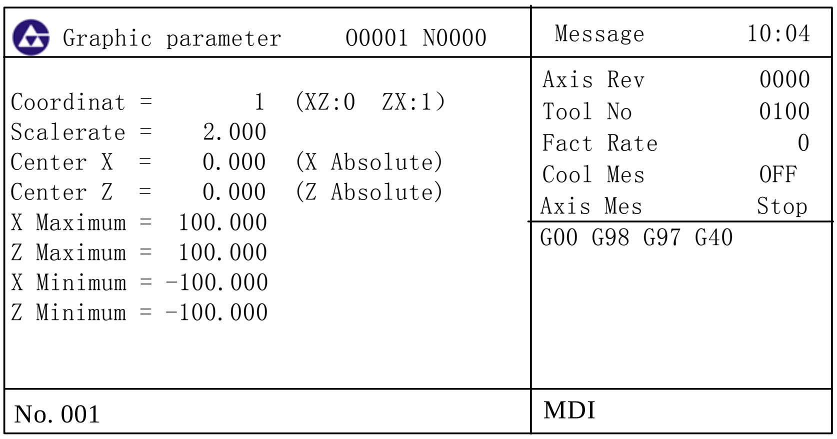


Fig. 3-5-2-1

Operation instructions:

1）. Graph (parameter) interface, see Fig. 3-5-2-1.

A. Significances of Graph (parameter)

Coordinate selection: There are 2 options for setting drafting plane, see the figure:

Set 0 for XZ coordinate, and set 1 for ZX coordinate, see figure 3-5-2-2.

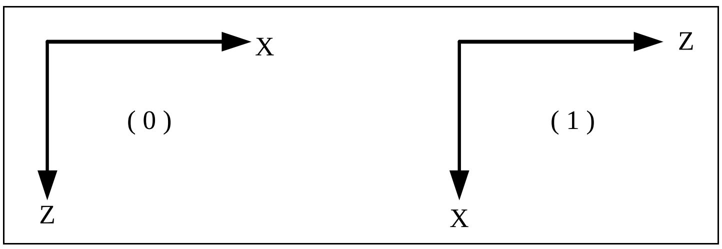


Fig. 3-5-2-2

Scaling ratio: To set drafting scale.

Graph center: To set the workpiece coordinate value to the LCD center in workpiece coordinate system.

Maximum and minimum value: After maximum and minimum values for the displayed axis are set, CNC system scales automatically and center value of the graph is set automatically.

X max.：X axial max. value in graph display（unit： mm）

X min.：X axial min. value in graph display（unit： mm）

Z max.：Z axial max. value in graph display（unit： mm）

Z min.：Z axial min. value in graph display（unit： mm）

B. Setting methods for graph (parameter)

a. In <MDI mode>, move the cursor to the parameter to be set.

b. Key in corresponding value according to the actual requirements.

c. Press 

key to confirm it.

2）. See the following figure for the 【Drafting】 interface (see 3-5-2-3):

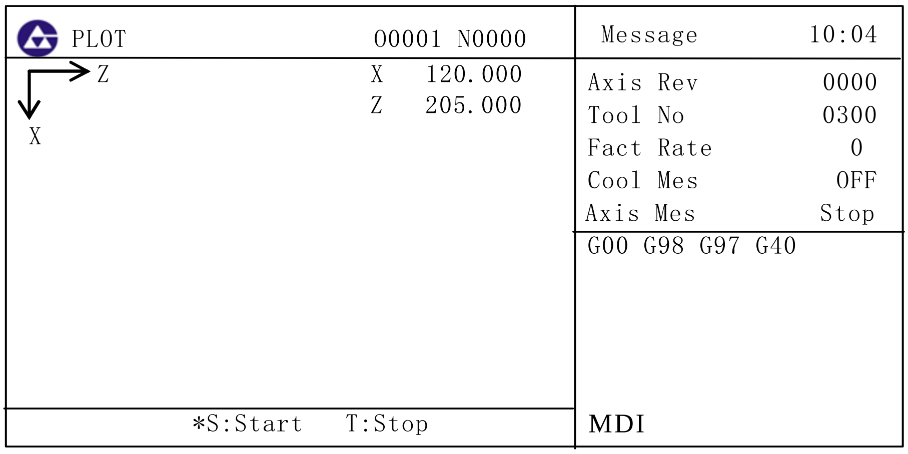


Fig. 3-5-2-3

In drafting interface, processing path of the program can be monitored.

A. Press 

key to enter the state of drafting and mark '*' moves to **S: drafting start**. In the operation of Auto/MDI/Manual mode, absolute coordinate value is changed and the corresponding motion path is drafted.

B. Press 

key to enter the state of drafting stop and mark '*' moves to **T: drafting stop**.

C. Press 

key to cancel the drafted graph.

D. Press up, down, left, right keys to move the whole graph to the appropriate position.

3.6 Display, Modification and Setting of the Parameter

3.6.1 Parameter Display

Press PARAMETER key to enter parameter Interface. There are two kinds of display modes, which are 【Parameter】,【Screw-pitch compensation parameter】. They can be viewed by pressing PARAMETER key continuously.

1）. See the figure for parameter interface (see 3-6-1-1):

```
PARAMETER                    O0001  N0000   | Message              10:04
   No.      DATA        No.      DATA       | Axis Rev              0000
  _001    00000000      009    00001100     | Tool No               0100
   002    01000001      010    00000000     | Fact Rate                0
   003    01000100      011    00000001     | Cool Mes               OFF
   004    00100000      012    00000000     | Axis Mes              Stop
   005    01000000      013    00000000     | G00 G98 G97 G40
   006    00000000      014    10001111     |
   007    00000000      015    00000000     |
   008    00000000      016    00000000     |
LAN SCW ***  MDSP  WHLA  RAD  ***  ***      |
 Bit0: Unused                               |
 No. 001                                    | MDI
```

Fig. 3-6-1-1

Detailed explanations for the parameter where the cursor locates are displayed in the lower part of the screen. See appendix 1 Parameter Explanation for definitions of each parameter.

2）. See the following figure for the screw pitch compensation interface (See 3-6-1-2):

```
WORMCOMPENSATE (mm/1000)  O0001 N0000  | Message              10:04
   No.          X (um)       Z(um)     | Axis Rev              0000
  _000          0            0         | Tool No               0100
   001          0            0         | Fact Rate                0
   002          0            0         | Cool Mes               OFF
   003          0            0         | Axis Mes              Stop
   004          0            0         | G00 G98 G97 G40
   005          0            0         |
   006          0            0         |
   007          0            0         |
   008          0            0         |
   009          0            0         |
 ADRS                                  | MDI
```

Fig. 3-6-1-2

Function settings for the screw pitch compensation number, please refer to Machine Debugging-Screw Pitch Compensation.

3.6.2 Modification and Setting for Parameter and Screw Pitch Compensation Value

There are four methods for modifying and setting parameter and screw pitch compensation value.

1. Modify it in the page:

1) Select <**MDI**> operation mode.

2) Enter <Setting> page, and turn on the parameter switch in 【Setting】interface. 【Password】 is required to be input for some parameter modification, See 3.5.1 for detailed operation.

3) Press 

key to enter the parameter display page.

4) Move the cursor to the position where the parameter to be modified locates.

(1). Press the page key to display the page where the parameter to be altered. Press the direction key (use left and right keys in bit parameter) to select the parameter to be altered.

(2). Press 

key and input the sequence number of the parameter to be modified. Then press 

key to confirm it, and the cursor will move to the number directly.

5) Input new parameter value by numerical keys.

6) Press 

key, the parameter value is input and displayed.

7) Turn off the parameter switch after parameter setting and confirmation is finished.

2. Loading and saving with backup disk

1) Loading parameter value from backup (see Fig.3-6-2-1)

The system has parameter and screw pitch compensation backup function. Take parameter value as an example bellow. The backup function of screw pitch compensation value is similar to that of the parameter value. Therefore, there is no further explanations. In parameter value I/O interface, select parameter input and output function by moving operation keys up and down. When parameter output function is selected, the parameter being used (including bit parameter and data parameter) can be backed up by the system. Back up files according to selected A, B, C OR D (selected by left and right operation keys). A and B are read-only disks, while C and D are read-write disks. Parameters in disk A and B are set in factory (A is default value for servo drive and B is default value for step drive). Disk A, B, C or D can be processed during loading, while only disk C, D can be processed during saving. When selecting parameter loading function, select A, B, C or D by left and right operation keys, namely, electronic disk number of the parameter file is selected. Then press input key to finish the loading, the parameter that CNC is using becomes the newly input parameter. Detailed operation process is as follows:

① Set the parameter switch as ON.

② Select <MDI> mode.

③ Press PARAMETER key to enter parameter display interface

④ Press CHANGE key to enter parameter value I/O page.

⑤ Move the cursor up and down to "Parameter loading" position. Select the disk to be loaded by left and right key.

⑥ By pressing DATA INPUT key, the system will open corresponding electronic disk from the storage and load the data. After that, "Loading completion" is displayed in the lower left part of LCD.

2) Store the parameter to the disk (See the Fig. 3-6-2-1)

①～④ are the same as that above.

⑤Move the cursor up and down to "Parameter storing" position. Select the disk to be loaded by left and right key.

⑥ By pressing DATA OUTPUT key, the system will write the parameter value into the user selected electronic disk. After that, "input completion" is displayed in the lower left part of LCD.

If "The electronic disk", "Parameter" and "Backup" already exist in user partition, new parameter will cover the original parameter.

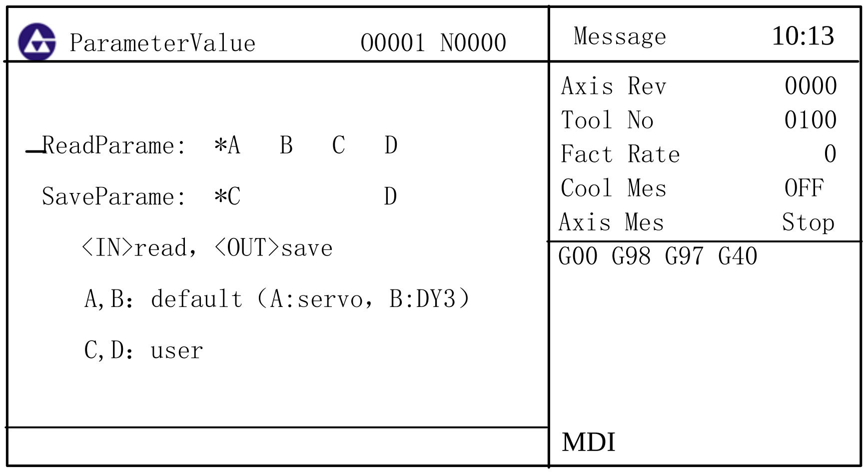


Fig. 3-6-2-1

The disk loading and storage of the screw pitch error compensation parameter are similar to that of the system parameter. In screw pitch parameter interface, press 

key to enter screw compensation electronic disk operation panel.

3. From CNC to CNC

The parameter and screw pitch error compensation can communicate with each by two different systems. For detailed operation, see sectioon10.5 Communication between CNC and CNC.

4. Transmission with software

Connect PC and CNC with communication cable, turn on the CNC program switch, and open user communication software in PC. See the chapter ten Serial port communication for detailed operations.

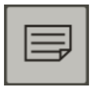
3.7　Diagonosis Display

DI/DO signal state between CNC and the machine, system interface signal, keys state of MDI panel, CNC input/output state, system internal CNC state signal, etc. are displayed on the diagnosis interface. Please refer to appendix three for corresponding significances of each diagnosis number.

This diagnosis is used for detecting the operation state between CNC interface signal and internal signal, which can not be modified.

3.7.1 Diagnosis Data Display

Press DIAGNOSIS key to enter diagnosis interface, which consists of【Diagnosis】,【Machine Panel】,【Macro Variable】three display modes. They can be viewed and modified by pressing DIAGNOSIS key repeatedly.

1. 【Diagnosis】interface can be viewed by pressing  or  key. See figure 3-7-1-1.

```
DIAGNOSTIC                      O0001 N0000  | Message          10:09
No.    DATA        No.    DATA               | Axis Rev         0000
001    00000000    009    00001100           | Tool No          0100
002    01000001    010    00000000           | Fact Rate           0
003    01000100    011    00000001           | Cool Mes          OFF
004    00100000    012    00000000           | Axis Mes         Stop
005    01000000    013    00000000           | G00 G98 G97 G40
006    00000000    014    10001111           |
007    00000000    015    00000000           |
008    00000000    016    00000000           |
Machine tool side input signal(IOboard)      |
Bit0:TCP DIQP DECX DITW *SP ST DECZ *ESP     |
No.  001                                     | MDI
```

Fig. 3-7-1-1

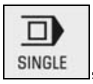
In diagnosis display interface, there are three diagnosis contents display columns in the lower part of the LCD. They display the contents of the diagnosis number where the cursor locates. See the *appendix III* for corresponding detailed significances of each diagnosis number.

2. See figure 3-7-1-2 for the contents of 【Machine Panel】 interface.

```
OPERATOR                        O0001 N0000  | Message          10:13
                                             | Axis Rev         0000
   MLK (KEY1) :   *OFF   ON                  | Tool No          0100
   AFL (KEY2) :   *OFF   ON                  | Fact Rate           0
   SBK (KEY3) :   *OFF   ON                  | Cool Mes          OFF
   DRN (KEY4) :   *OFF   ON                  | Axis Mes         Stop
   BDT (KEY5) :   *OFF   ON                  | G00 G98 G97 G40
                                             |
   POSITION (RELATIVE)                       |
      U     0.000                            |
      W     0.000                            |
                                             | MDI
```

Fig. 3-7-1-2

Part Three Operation

When the machine panel displays pages, use "1. 2. 3. 4. 5" keys to switch Machine lock, M.S.T Lock, Single Block, Dry run, Block skip. Their functions are the same as that of the , , , ,  keys on the panel.

3. See figure 3-7-1-3 for the contents of 【Macro variable】 interface.

MACRO		O0001 N0000	
No.	DATA	No.	DATA
200	0.000	208	0.000
201	0.000	209	0.000
202	0.000	210	0.000
203	0.000	211	0.000
204	0.000	212	0.000
205	0.000	213	0.000
206	0.000	214	0.000
207	0.000	215	0.000
POSITION(RELATIVE)			
U -169.380		W 122.337	
No. 200			

Message	10:09
Axis Rev	0000
Tool No	0100
Fact Rate	0
Cool Mes	OFF
Axis Mes	Stop
G00 G98 G97 G40	
MDI	

Fig. 3-7-1-3

There are two parts in macro variable interface, which are #200～#231 power-off clearing and #500～#515 power-off memory. See variable instructions in ***User Macro Variable*** for detailed functions.

In macro variable interface, macro variable setting and modification are the same as that of the tool compensation. See variable instructions in ***User Macro Variable*** for detailed functions.

Part Three Operation

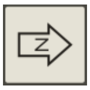
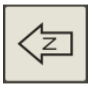
CHAPTER FOUR MANUAL OPERATION

Press the  to enter manual operation mode, which mainly includes the manual feed, the spindle control and the machine panel control etc.

4.1 Coordinate Axis Movement

In the mode of the manual operation, X and Z axes can be operated at the manual feedrate or the manual rapid traverse rate.

4.1.1 Manual Feed

Press the feed axis  or  key, and arrow indicates the direction here. The axis movement is stopped after releasing the key. In this case, the feed override can be adjusted to change the feedrate and the operation is similar to X-axis.

4.1.2 Manual Rapid Traverse

Press  key and enter the manual rapid traverse state after the indicator  on the upper part of the edit panel is lighted up, and then press the key of feed axis, each axis moves at the rapid traverse rate.

Note 1: The feedrate performed in manual rapid traverse, the time constant and the acceleration/ deceleration modes are same as rapid traverse rate specified by program commands (G00 positioning).

Note 2：When rapid traverse is active (Rapid traverse indicator  lights up), NO.005 Bit6 determines whether the rapid move speed is manual feedrate or rapid traverse speed before the reference position returns after power on.

Note3: In Edit/MPG mode, the key is inactive and the indicator is off. Rapid feed can be selected in other modes and it is cancelled after the other mode is shifted.

4.1.3　Manual Feed and Manual Rapid Traverse Rate Selection

When the manual feed operation is performed, federate override adjustment keys 

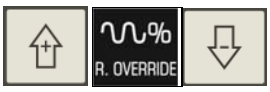
on the panel can be used to select the manual federate override (16 in total, 0%～150%).

See the table 4-1-3-1:

Table 4-1-3-1

Feedrate override (%)	Feedrate (mm/min)
0	0
10	1.5
20	3.2
30	5.0
40	7.9
50	12.6
60	15
70	32
80	50
90	79
100	126
110	200
120	320
130	500
140	790
150	1260

Note: The table has an error margin of 3%.

When the manual rapid traverse is performed, the manual rapid traverse rate can be selected by pressing rapid traverse adjustment keys 

. Five gears rapid override are available: Fo, 25％, 50％, 75％ and 100％ (Fo is set by the data parameter P32).

Note 1: The rapid override selection is active for the following traverse speed

(1) G00 rapid feed

(2) Rapid feed in canned cycle

(3) Rapid feed in the command G28

(4) Manual rapid feed

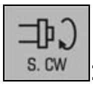
(5) Rapid feed in the manual reference return

Example: when the rapid feedrate is 8 m/min, and if the override is 50%, the speed is 4 m/min.

Note 2: The rapid override is inactive when the axis is moving.

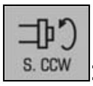
4.2 Spindle Control

4.2.1 Spindle CW (Negative rotation)



: S rotation speed can be specified in MDI mode, the spindle rotates CW by pressing this key in the mode of manual/MPG/single step.

4.2.2 Spindle CCW (Positive rotation)



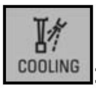
: S rotation speed can be specified in MDI mode, the spindle rotates CCW by pressing this key in the mode of manual/MPG/single step.

4.2.3 Spindle Stop

: The spindle stops in the mode of manual/MPG/single step by pressing this key.

4.3 Other Manual Operations

4.3.1 Cooling Control



: It is a compound key. The cooling is shifted between on and off. Its ON/OFF state can be viewed on LCD screen.

4.3.2 Lubricating Control

： It is a compound key. The lubricating function is shifted between on and off.

4.3.3　Manual Tool Change

：In Manual/MPG/Single step mode, by pressing this key, tool post rotates to change for another tool. Total tool number can be set by the data parameter P81. See the machine tool manufacturer’s manual.

4.4　Tool Setting

Several tools are needed for processing a part. Because the tool installation error and tool error exist, tool nose positions do not entirely coincide when the tools rotate to the cutting position. In order to facilitate user programming without caring about the deviation between tools, the system is designed with tool offset automatic generating method. After tool setting, user makes programs according to part drawing and machining techniques without having to concern the deviation between tools. User only have to call corresponding tool compensation value in programs. The system is designed with many modes such as fixed-point tool setting, tool setting by test cutting and tool setting by machine zero return.

4.4.1　Fixed-point Tool Setting

When setting offset in 

interface, if press 

key after inputting the address key U (or W), the current relative coordinate value is set as offset value corresponding to this address.

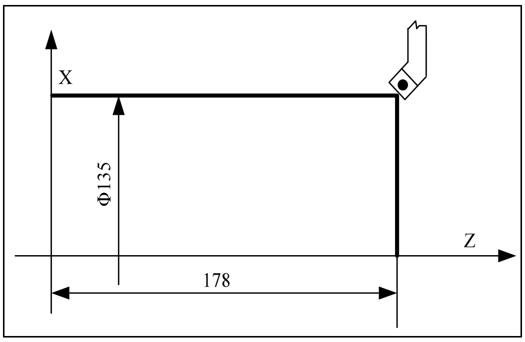


Fig. A

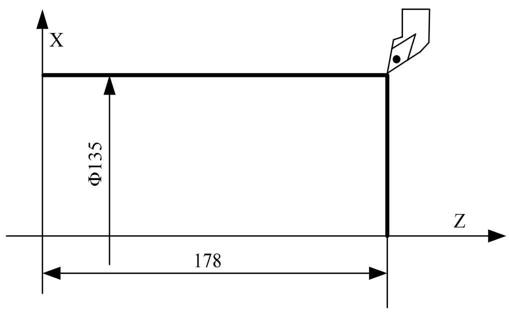


Fig. B

Fig. 4-4-1-1

Establish a workpiece coordinate system before the fixed-point tool setting.

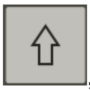
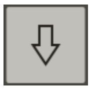
Steps:

1）Select a tool without tool offset at random (usually the 1st tool, this tool will be used as the reference tool)

2）Position the tool nose of the reference tool to a point (tool setting point), as shown in Fig. A

3）Clear the relative coordinate (U, W). See section 3.1.1 for detailed operations

4）Press 

key and move the cursor by ,  keys to select a offset and take it as the offset of the reference tool.

First confirm whether the X and Z axial tool offset values are zero or not. If it is not zero, clear it as the following method.

Select number 001～064 tool offset by moving the cursor, input X0, Z0 to clear it.

5）After the tool is moved to a safe position, select another tool and move it to the setting point. As shown in Fig. B.

6）Press 

key and select a offset in 001～064 as the offset of the tool. Then confirm whether the X and Z axial tool offset values are zero or not. If it is not zero, clear it as above step 4）introduced.

7）Press address key 

, then press 

key to input the tool offset value of X axis into the corresponding offset number of that.

8）Press address key 

, hen press 

key to input the tool offset value of Z axis into the corresponding offset number of that.

9）Repeat the steps from 5 to 8 to set the tool offset for other tools.

4.4.2 Tool Setting by Test Cutting

When the workpiece coordinate system is not changed, perform tool setting by test cutting. Operation steps are as follows (Establish coordinate system on the part end surface)：

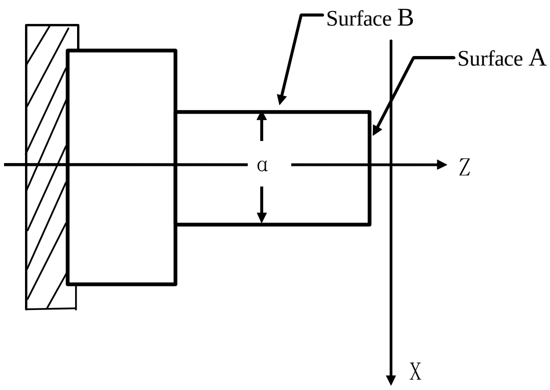


Fig. 4-4-2-1

1. Select a tool without tool offset at random (supposing it is the 1st tool)
2. In manual mode, it cuts along surface A in negative direction of X axis.
3. Retract the tool along the positive direction of the X axis when Z axis is not moving and stop the spindle.
4. Press OFFSET key to enter the Offset interface, and move the cursor by ⇧, ⇩ keys to select offset 101 and take it as the offset of the tool.
5. Key in by sequence of the address key “Z”, numerical key “0” and DATA INPUT key.
6. In manual mode, it cuts along surface B in negative direction of Z axis.
7. Retract the tool along the positive direction of the Z axis when X axis is not moving and stop the spindle.
8. Measure diameter “a” (Supposing α’=30)
9. Press OFFSET key to enter the Offset interface, and move the cursor by ⇧, ⇩ keys to select offset 101.
10. Key in by sequence of the address key “X”, numerical key“3”, “0” and DATA INPUT key.
11. Move the tool to the safe position for tool changing.
12. Change another tool (supposing it is the 2nd tool) without tool offset.

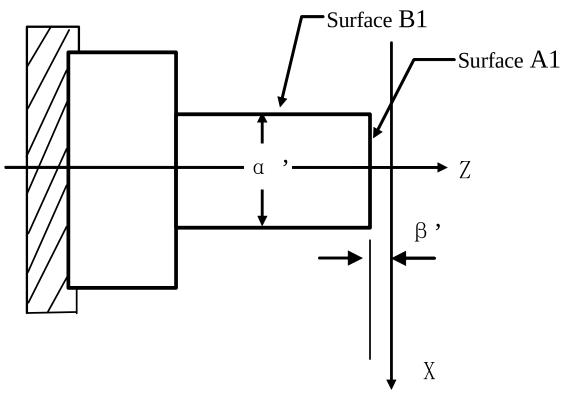


Fig. 4-4-2-2

13. In manual mode, it cuts along surface A1 in negative direction of X axis.
14. Retract the tool along the positive direction of the X axis when Z axis is not moving and stop the spindle.
15. Measure the distance “β” from surface A1 to the workpiece coordinate system (Supposing β=1).
16. Press OFFSET key to enter the Offset interface, and move the cursor by ⇧, ⇩ keys to select offset of 102.

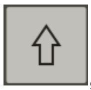
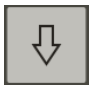
Part Three Operation

17.Key in by sequence of the address key “Z”, symbol key“-”, numerical key“1” and  key.

18.In manual mode, it cuts along surface B1 in negative direction of Z axis.

19.Retract the tool along the positive direction of the Z axis when X axis is not moving and stop the spindle.

20.Measure the diameter "α’" (Supposing α’=28)

21.Press  key to enter the Offset interface, and move the cursor by ,  keys to select offset of 102.

22.Key in by sequence of the address key “X”, numerical key“2”, “8” and  key.

23.Repeat the steps from 11 to 22 to perform tool setting operation for other tools.

4.4.3 Tool Setting by Machine Zero Return

There is no reference tool in this tool setting method. When the tool is worn or to be adjusted, it only needs to be set again. Machine zero return should be done before the tool setting (See section 8.2 in this manual for machine zero return). The machining can be continued by performing machine zero return once after reset, which is convenient for the operation.

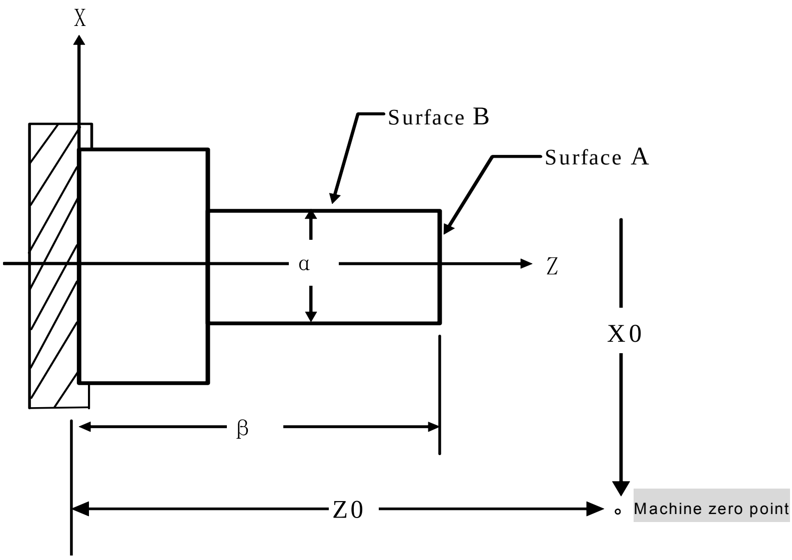


Fig.4-4-3-1

Operation steps are as follows:

1） Press  key to enter Machine Zero Return mode. Press positive direction keys of the X and Z axes to make the two axes return to machine zero point.

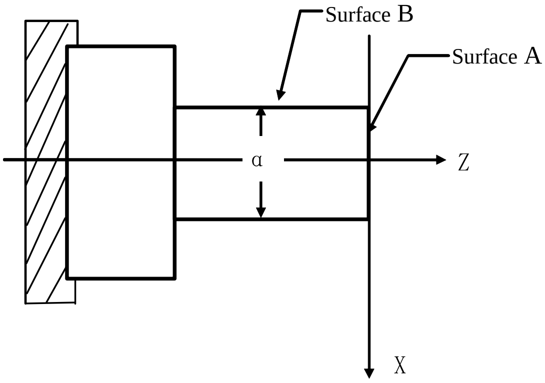


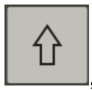
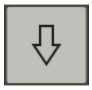
Fig. 4-4-3-2

2） Select a tool without tool offset at random (supposing it is the 1st tool)

3） In manual mode, it cuts along surface A in negative direction of X axis.

4） Retract the tool along the positive direction of the X axis when Z axis is not moving and stop the spindle.

5） Press 

key to enter the Offset interface, and move the cursor by ,  keys to select offset 101 as the offset number of the tool.

6） Key in by sequence of the address key “Z”, numerical key“0” and 

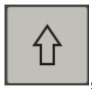
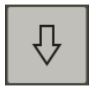
key.

7） In manual mode, it cuts along surface B in negative direction of Z axis.

8） Retract the tool along the positive direction of the X axis when Z axis is not moving and stop the spindle.

9） Measure the diameter "α’" (Supposing α’=30)

10） Press 

key to enter the Offset interface, and move the cursor by ,  keys to select offset of 101.

11） Key in by sequence of the address key “X”, numerical key“3”, “0” and 

key.

12） Move the tool to the safe position for tool changing.

13） Change another tool (supposing it is the 2nd tool) without tool offset.

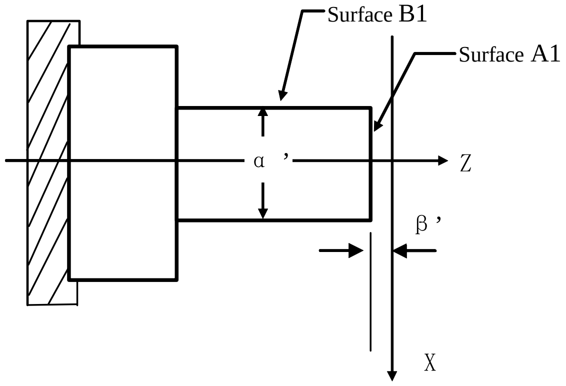


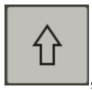
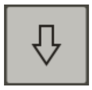
Fig. 4-4-3-3

14） In manual mode, it cuts along surface A1 in negative direction of X axis.

15） Retract the tool along the positive direction of the X axis when Z axis is not moving and stop the spindle.

16） Measure the distance “β” from surface A1 to the workpiece coordinate system (Supposing β=1).

17） Press 

key to enter the Offset interface, and move the cursor by ,  keys to select offset of number 102.

18） Key in by sequence of the address key “Z”, symbol key “-”, numerical key “1” and 

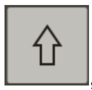
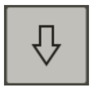
key.

19） In manual mode, it cuts along surface B1 in negative direction of Z axis.

20） Retract the tool along the positive direction of the Z axis when X axis is not moving and stop the spindle.

21） Measure diameter “a” (Supposing α’=28)

22） Press 

key to enter the Offset interface, and move the cursor by ,  keys to select offset of number 102.

23） Key in by sequence of the address key “X”, numerical key“2”, “8”and 

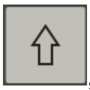
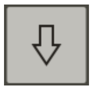
key.

24） Repeat the steps from 12 to 23 to perform tool setting operation for other tools.

4.5　Offset Alteration

Offset can be input by U, W.

Example: If 0.010mm is need to be added to the offset value of the Z axis, press 

key to enter the Offset interface, and move the cursor by ,  keys to select a offset number of 001, then input U0.010 and press 

key, and 0.010mm is added to the original offset value automatically by the system.

CHAPTER FIVE MPG/SINGLE-STEP OPERATION

Press 

key to enter MPG/ Single step mode, which can be selected by bit parameter NO:1#3. In the MPG/single step feed mode, the machine moves according to the defined step length in the system each time.

5.1 Step Feed

Set parameter **NO:1#3** to 0, and press 

to enter Step operation mode. The move increment can be selected by pressing any key of 

, 

, 

, and it is displayed on the interface. The step increment 0.100 is displayed on <Position> interface by pressing 

key, （See figure 5-1-1）:

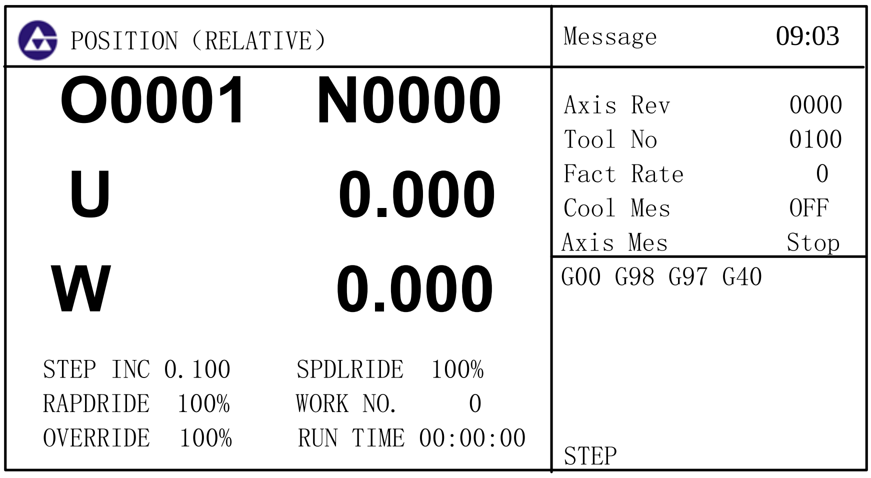


Fig. 5-1-1

Press X- or X+ key, it can move the X axis negatively or positively by a step increment. Press Z- or Z+ key, it can move the Z axis negatively or positively by a step increment.

Note 1: The speed of single step movement and that of the manual feed are the same.
Note 2: Rapid override and federate can be altered in Single Step mode.

5.2 MPG (Manual pulse generator) Feed

Set parameter **NO:1#3** to 1, and press MPG to enter MPG operation mode. The move increment can be selected by pressing any key of 0.001, 0.01, 0.1, and it is displayed on the interface. The MPG increment 0.100 is displayed on <Position> interface by pressing 0.1 key, （See figure 5-2-1）:

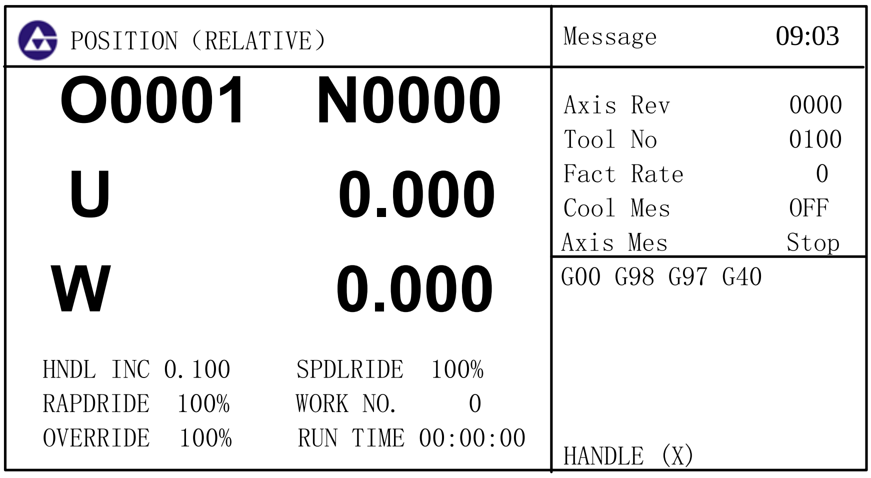
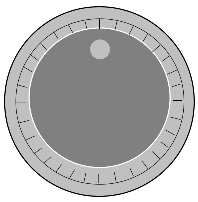


Fig. 5-2-1

THE MPG EXPERIENCE IS AS THE FIGURE BELLOW：



Manual pulse generator（MPG）

5.2.1 Moving Axis and Direction Selection

In MPG mode, press X, Y or Z key, the corresponding axis is selected and it will be controlled by MPG. For example, X axis can be moved by moving MPG after X key is pressed (See figure 5-2-1-1).

Generally, MPG CW is for positive feed, and CCW for negative feed.

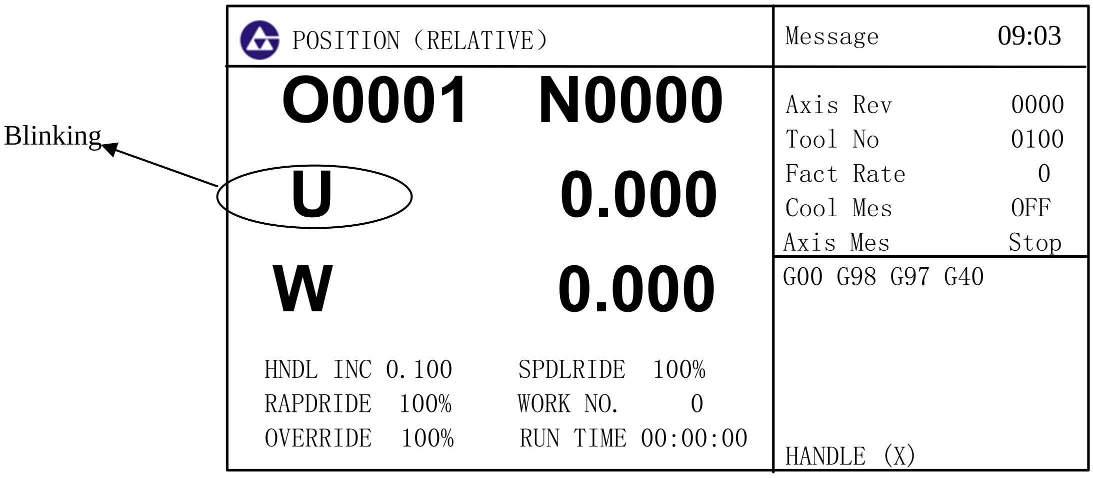


Fig. 5-2-1-1

The MPG (Manual pulse generator) feed direction is defined by its rotation direction. See the manufacturer's instructions for details. Generally, MPG CW is for positive feed and CCW is for negative feed.

5.2.2 Instructions for MPG Feed

1. Relationship between MPG scale and machine movement amount is shown in the table bellow:

Table 5-2-2-1

	Movement amount of each MPG scale		
MPG increment (mm)	0.001	0.01	0.1
Machine movement amount (mm)	0.001	0.01	0.1

2. Values in the above table vary with mechanical transmission. See manufacturer's instructions for details.
3. The rotation speed of the MPG should be less than 5 r/s. If it is over 5 r/s, the scale may not coincide with the movement amount.

4. The 

 and 

 keys are only active in MPG/Step mode.

5.3 The Miscellaneous Control in MPG/Step Operation

It is the same as the manual operation mode. Refer to the section **4.2** and **4.3** of this operation manual for details.

CHAPTER SIX　MDI OPERATION

The MDI operation function is added besides data input, parameter and offset alteration functions. Commands that are input directly can be executed with this function. There are detailed instructions for MDI, parameter and offset alteration in *interface display and data alteration and setting*, chapter 3. MDI operation function is introduced in this chapter.

6.1　MDI Blocks Input

Program input in 【MDI】mode is the same as that of the Edit mode, *see edit operations*, chapter 7. Input steps in 【MDI】 mode are introduced bellow.

Example: Steps for inputting a block G00 X50　Z50 by 【MDI】operation interface are as follows:

1. Press 

key to enter MDI mode

2. Press 

key to enter 【MDI】operation interface, see section 3.2 for detailed operations. (See figure 6-1-1)

3. After the address key G00 is input, press 

key to confirm the operation and G00 is displayed on LCD screen. Input X50 and press 

key to confirm. Then input Z50 and press 

key to confirm. At this time, programs are input and can be viewed in the interface, see the figure bellow:

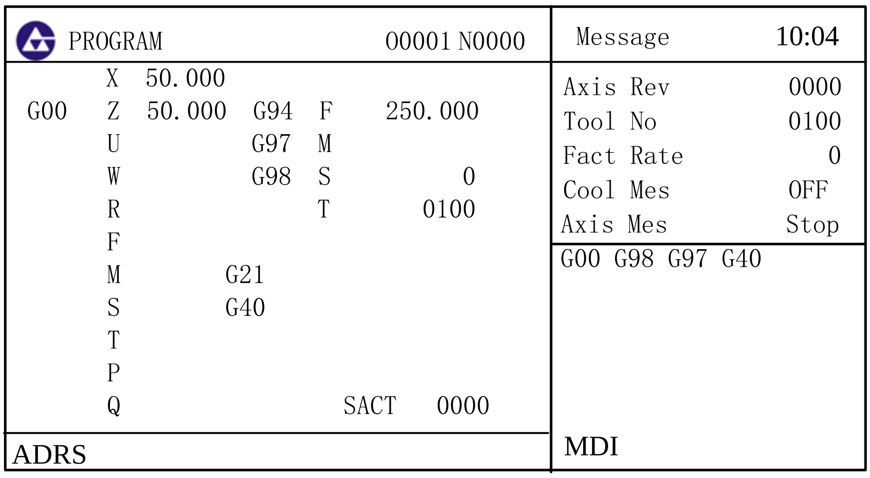
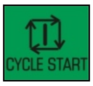


Fig. 6-1-1

6.2　MDI Block Operation and Stop

The MDI can be performed by pressing  key after the command is input as the steps in section 6.1. The block operation can be stopped by pressing  or pressing  key during operation.

Note:　MDI operation must be performed in the MDI mode!

6.3　MDI Block Words Alteration and Clearing

If there are errors during words input, press  key to cancel it word by word. If the errors are found after it is finished, input right one to take the place of the wrong one or press  key to clear all contents, and then input it again.

6.4　Conversion of Operation Modes

The MDI mode operation may stop immediately after shifting to the other modes. Specific as follows:

1）After shifting to Edit, AUTO mode, the machine is stopped after the current block is performed.

2）After shifting to Manual, MPG, Step mode, the machine is stopped immediately.

3）After shifting to Machine zero return interface, the machine decelerates to stop.

CHAPTER SEVEN EDIT OPERATION

7.1 Program Edit

To prevent the program from being modified or deleted by people at random, the system is designed with a program switch. The switch must be turned on before program editing. Please refer to switch setting in section 3.5.1.

Workpiece program editing should be performed in edit mode. Press 

key to enter edit mode.

Press 

key to enter program display interface, and the program can be edited and altered (See Fig. 7-1-1).

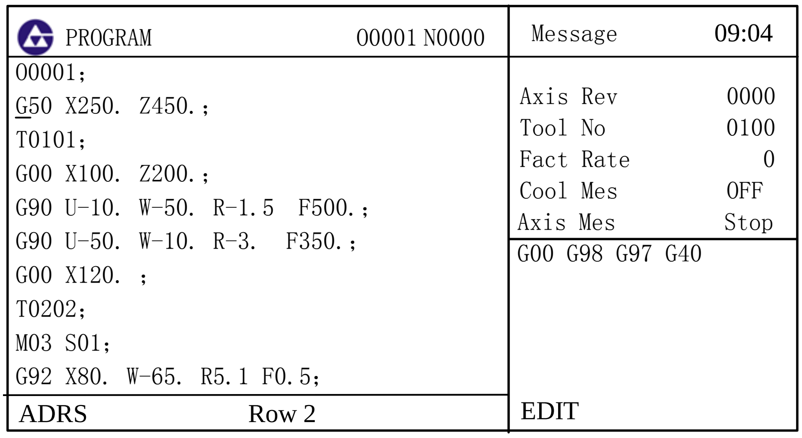


Fig. 7-1-1

7.1.1 Program Creation

7.1.1.1 Creation of the Program Number

By setting the parameter NO:2#7, it is able to set whether the block number is active or not. If the parameter is set to 1, the block number is inserted automatically during program editing, increment of the block number is set by the data parameter P50.

7.1.1.2 Input and Storage of the Program

1. Press **EDIT** key to enter the Edit mode.

2. Press **PROGRAM** key to enter the Program interface (See figure 7-1-1-2-1)

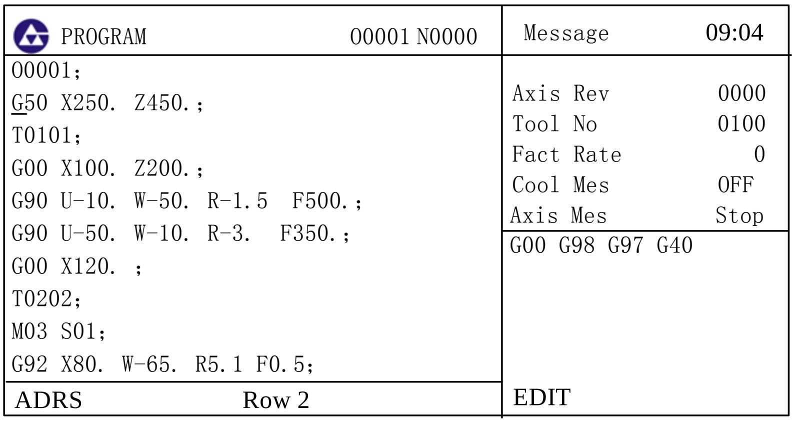


Fig. 7-1-1-2-1

3. Press address key **O**, and key in numerical keys by sequence of **3**, **3**, **3**, **3** (Program O3333 creation for an example). O3333 is displayed behind the numerical column. See the figure bellow (Fig. 7-1-1-2-2).

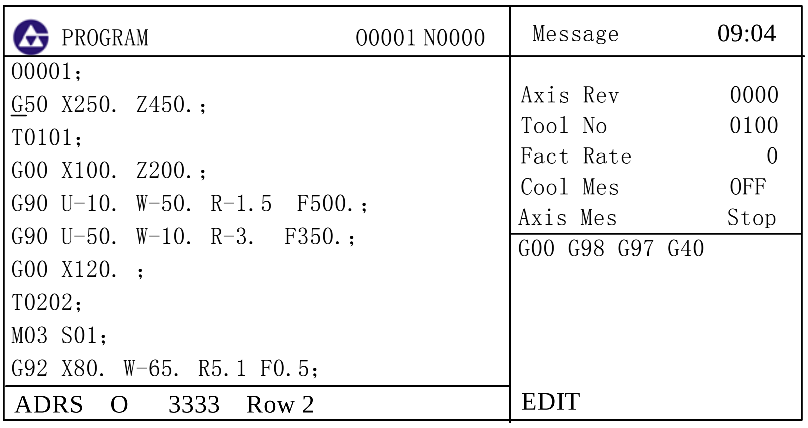


Fig. 7-1-1-2-2

4. Press **EOB** key to create a new program, as shown in the following figure (Fig. 7-1-1-2-3).

PROGRAM 03333 N0010	Message 09:04
03333; N10 %	Axis Rev 0000 Tool No 0100 Fact Rate 0 Cool Mes OFF Axis Mes Stop
	G00 G98 G97 G40
ADRS Row 2	EDIT

Fig. 7-1-1-2-3

5. Input the edited program word by word, and press 

key after a block is finished. Switch to other interface (e.g. 

interface) after a program is finished, or switch to other working mode and at the same time, the program is saved automatically.

Note 1: In Edit mode, only a complete word can be input. Single character or number can not be input.

Note 2: If there are errors during block inputting, press 

key to cancel the whole block.

7.1.1.3 Search of the Sequence Number, Character and Line Number

Sequence number retrieval, which means searching a sequence number of a program, is used to start program execution or edit from the retrieved sequence number. The block skipped for searching has no effect on the state of the CNC. (The coordinate value, M, S, T code and G code of the skipped block have no effect on the coordinate value and mode value of the CNC).

If execution starts from a block of the retrieved program, the machine state and CNC state should be checked. In addition, they should be consistent with corresponding M, S, T code and the coordinate system setting (it can be set by MDI mode) before execution.

Character search is used to search the specified address or numerical characters in program, which is mainly used in edit program.

Steps of searching sequence number, characters in program:

1) Press 

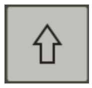
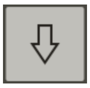
key to enter Edit mode.

2) Open the target program in 

interface. See 6.1 for detailed operations.

There are two methods for searching characters:

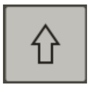
1）Scanning: find the page where the target locates by the page key, and then move the cursor to the target position.

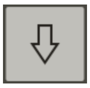
2）Search method: input the character or sequence number to be searched, and press  or  key to search it upward or downward. The same one downward can be searched, the alarm occurs if it is not searched.

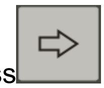
7.1.1.4　Cursor Positioning

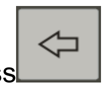
In edit mode, select 

key to enter program page.

1） Press  key to move up the cursor to the last line. When the line which the cursor is in is bigger than the end row of the last line, the cursor can be moved up to the end of the last line.

2） Press  key to move down the cursor to the next line. When the line which the cursor is in is bigger than the end row of the last line, the cursor can be moved down to the end of the next line longer

3） Press  to move right the cursor to one row. When the cursor is in the end of the line, it can be moved to the home of the next line.

4） Press  to move left the cursor to one row. When the cursor is in the home of the line, it can be moved to the end of the last line

5） Press  to page up the screen and the cursor moves to the same position of the last screen.

6） Press  to page down the screen and the cursor moves to the first row of the first line in the next screen.

7） Press RESET key, the cursor moves to the head of the program.

7.1.1.5 Character Insertion, Deletion and Alteration

By the methods in **7.1.1.3** and **7.1 .1.4,** move the cursor to the position to be altered.

1. Insertion of the character

After inputting the data, press INSERT key, the input contents will be inserted at the right side of the cursor.

2. Deletion of a character

Move the cursor to the character to be deleted, and the character where cursor locates will be deleted by pressing DELETE key. If the DELETE key is pressed continuously, the contents at the right side of the cursor will be deleted one by one.

3. Alteration of the character

Move the cursor to the character to be altered, then input the contents and press ALTER key, the contents is input to take the place of the character where the cursor locates.

7.1.2 Deletion of a program

Steps for deleting a program in the memory are as follows:

1) Select <Edit> mode.

2) Press PROGRAM key to enter the program display interface.

3) Press address key O.

4) Input the name of the program to be deleted

5) Press DELETE key, and the corresponding program in the memory will be deleted.

Note: If the program does not exist, system alarms (P/S alarm: 014 the program does not exist)

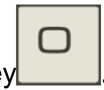
7.1.3　Deletion of All Programs

Steps for deleting all programs in the memory are as follows:

1) Select <Edit> mode.

2) Press 

key to enter the program display interface.

3) Press address key  .

4) Input -9999 and press 

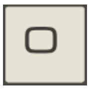
key, all the programs will be deleted.

7.1.4　Duplication of the Program

Copy the current program and save it with a new program name.

1） Select <Edit> mode.

2） Enter the program display interface, and open the program to be copied.

3） Press address key  and input new program number.

4） Press 

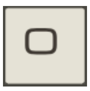
key to complete the file duplication and enter a new program edit interface.

7.1.5　Rename of the Program

Change the current name to a new one:

1） Select <Edit> mode.

2） Enter program display interface.

3） Key in address  and input new program name.

4） Press 

key to complete the performance.

Note 1: In program creation, if the input program name exists, this program will be opened.

Note 2: In program duplication or rename program, the name of the target program should not

be the same as any one existed in the memory. Otherwise, the alarm occurs.

7.2 Program Management

7.2.1 Search of Program List

Press PROGRAM key to enter the Program List page (See 7-2-1-1):

```
PROGRAM                             O0001 | Message          10:04
N0000                                     | Axis Rev          0000
  SYSTEM EDIT:GSK-980TA3(08) V3.74.083    | Tool No           0100
  PRG.NO.USED:    13     FREE:687         | Fact Rate            0
  MEMORY USED:    704    FREE: 23840      | Cool Mes           OFF
  PRG.LIBRARY:                            | Axis Mes          Stop
 O0000 O0001 O0002 O0003 O0004 O0005      | G00 G98 G97 G40
 O0006 O0007 O0008 O0009 O0010 O0011      |
 O0045                                    |
                                          | MDI
ADRS
```

Fig. 7-2-1-1

In this page, it displays all programs names saved in the memory in the form of program list. If one page can not display all programs, press CHANGE key to the next page.

7.2.2 Quantity of the Saved Programs

The memory of this system can save 500 programs. For quantity of saved programs, please refer to program list page in the above 7.2.1.

7.2.3 Memory Capacity

For detailed storage state, please refer to program list page in the above 7.2.1.

7.2.4　Program Lock

To prevent the program to be altered or deleted by people at random, the system is designed with a program switch. It must be turned off after program editing. Please refer to section 3.5.1 for details.

7.3　Other Operations Available in Edit Mode

In the Edit mode, the following operations are available:

1. Spindle override adjustment.
2. Rapid override adjustment.

CHAPTER EIGHT AUTO OPERATION

8.1 Selection of the Program To Be Run

1. Program loading in Edit or Auto mode

(a) Select the Edit or Auto mode

(b) Press PROGRAM key to enter the PRG display page

(c) Press the address key O to input the name of the program to be performed and then press ⇩ key to search the program (Or after pressing O key once, press ⇧ or ⇩ key directly to search the program existed)

(d) If the program name is searched, the program searched will be shown on the LCD screen. If the

program does not exist, the alarm will be issued.

2. Load programs in MDI mode

(a) In MDI mode.

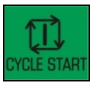
(b) Press PROGRAM key to enter the PRG display page, and move the cursor to the program to be performed.

(c) Press EOB key to confirm.

(d) Select the Edit or Auto mode, and press PROGRAM key to enter the PRG display page, then the selected contents can be viewed on the LCD screen.

8.2 Start of the Auto Run

After the programs are selected by the two methods introduced in 8.1. In <AUTO> mode, press CYCLE START key to start the program. <**Position**>, <**Program**> and <**Graph**> interfaces can be shifted to observe the program operation state.

Since the program execution begins from the block where the cursor locates, before pressing the  key, make a check whether the cursor is located at the block to be executed and whether each mode is correct. If begins from the start line, move the cursor to the start line by pressing  key in Edit mode, and press  key to execute the program from the beginning .

Note: If the cycle start key is pressed continuously, the system issues alarm that ERR191 overtime. The parameter P159 can be altered to adjust the key alarm time.

8.3 Auto Run Stop

In Auto run, there are five ways to stop the programs:

1. Stop by  key

In Auto run, by pressing key  or external dwell key, the machine remains at the following state:

1) The machine feed decelerate to stop

2) During the execution of the dwell command (G04), it pauses after G04 command execution is finished.

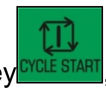
3) The modal function and state are saved

4）The program execution continues after pressing the  key

2. Stop by Reset key 

Please refer to section 2.3.1 operation manual.

3. Stop by code (M00)

After the block containing M00 is executed, the auto run is stopped. So the modal function and state are all reserved. Press the key , the program execution continues

4. Stop by Emergency stop button

Please refer to section 2.3.2 operation manual.

5. Stop by mode switching

When the programs are being performed in Auto, MDI interface of MDI mode, the program can be stopped by switching to other modes. See the following for details:

1） When the Auto mode is switched to Edit, MDI interface, the machine can be stopped after the current block is executed.

2）When the Auto mode is switched to Manual, MPG, Step interface, the machine is stopped immediately.

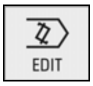
3）When the Auto mode is switched to the Machine zero, the machine decelerates to stop.

8.4 Auto Run From an Arbitrary Block

The system supports auto run from an arbitrary block. Detailed operation steps are as follows:

1. Press 

key to enter the Manual mode, and start the spindle and other miscellaneous functions.

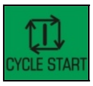
2. Press 

key to enter the Edit mode, open the programs to be executed. (See section 8.1 for detailed operations)

3. Move the cursor to the block to be executed;

4. Press 

key to enter the Auto mode;

5. Press 

key to start the program.

Note: When a block is selected, confirm the relationship between the selected block and the current absolute coordinate. If the program of tool change will be performed, ensure collision will not happen at the current position to avoid machine tool or people damage.

8.5 Dry Run

Before the program execution, in order to avoid the programming errors, the Dry run mode is selected to check the program. It is used with M.S.T lock and machine lock.

Method 1: Press 

key to enter Auto mode, and press 

(the indicator  on the upper part of the edit panel lights up, it means that the dry run function is selected).

Method 2: In Auto or MDI mode, press the 

key continuously to enter the tool panel page, press numerical key 4 to select Dry run ON/OFF.

In Dry run mode, no matter how the federate is defined by the programs, the system runs at the rates in the following table:

Table 8-5-1

	Program command		
	Rapid feed（see note 1）	Cutting feed（Except for screw feed）	Screw feed
Rapid traverse switch ON	Rapid feed	Max. JOG feed	Screw lead *500
Rapid traverse switch OFF	JOG federate or rapid feedrate（see note 2）	JOG feed	

Note 1: X, Z axis speed of the rapid feed are set by the data parameters P21, P22 in the parameter page.

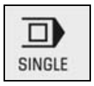
Note 2: By setting the parameter NO：2#1, it is able to set whether the rapid traverse command in Dry run is active or not.

Note 3: If the dry run function is executed in Auto or MDI mode, it is still active when the Auto mode or MDI mode are switched to the other modes (Manual, MPG, Edit, Machine zero return, Program zero), but the ON/OFF of the Dry run can not be set at this time.

8.6 Single Block Execution

If the state of single block execution is to be checked, select Single Block execution.

Method 1: Press 

key to enter Auto mode, and press 

key (If the indicator  on the upper part of the edit panel lights up, it means that the Single Block execution is selected). In single mode, when the current block execution is finished, the CNC stops. If the next block is to be executed, it needs to press the 

key, and then repeat this operation until the whole program is finished.

Method 2: In Auto or MDI mode, press the 

key continuously to enter the Tool panel page, press the numerical key 3 to select Single block ON/OFF.

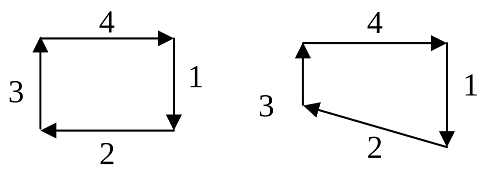
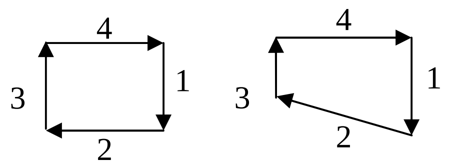
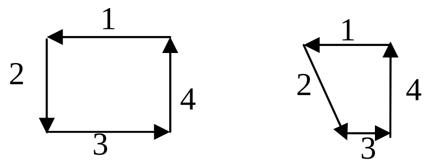
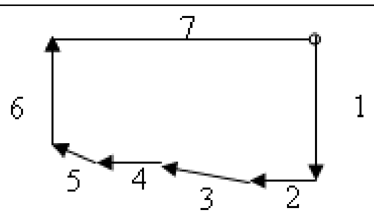
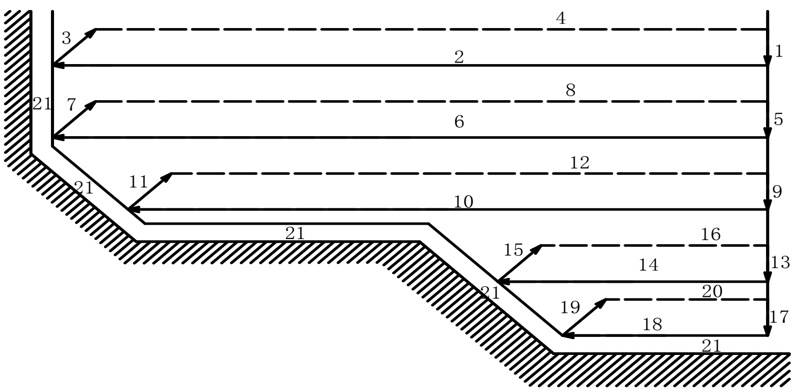
Note 1: Even at the mid point, the single block stops in G28.

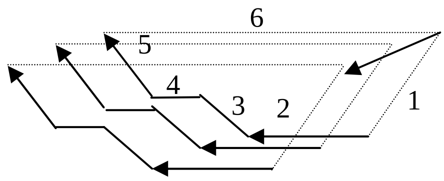
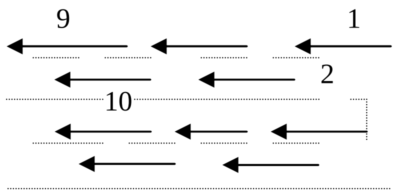
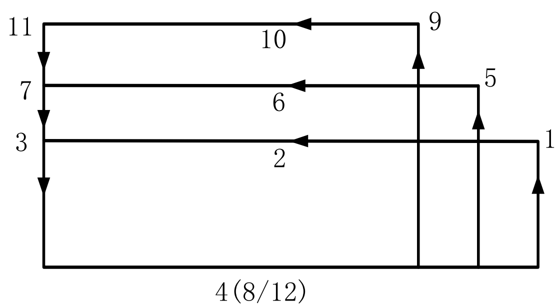
Note 2: When the single block is on, and the canned cycle G90，G92，G94，G70～G75 are being executed, see the table 8-6 for details.

Note 3:Single block stop is available in programs M98＿； M99；and G65. By setting the system parameter P13 BIT0, it is able to set G65 single block execution is active.

Note 4: If the single block function is executed in Auto or MDI mode, it is still active when the Auto mode or MDI mode are switched to the other modes (Manual, MPG, Edit, Machine zero return, Program zero), but the ON/OFF of the Single block function can not be set at this time.

Table 8-6-1

G code	Tool path	Explanation
G90		Action 1 to 4 is a complete circulation. It stops after action 4 is finished.
G92		Action 1 to 4 is a complete circulation. It stops after action 4 is finished.
G94		Action 1 to 4 is a complete circulation. It stops after action 4 is finished.
G70		Action 1 to 7 is a complete circulation. It stops after action 7 is finished.
G71，G72	 Note: This figure is for the condition of G71. Condition G72 is the same.	1～4，5～8，9～12，13～16，17～20 is a circulation, it stops after actions are finished.

G73		1 ～ 6 is a circulation, it stops after actions are finished.
G74，G75		1 ～ 10 is a circulation, it stops after actions are finished.
G76		1～4，5～8，9～12 is a circulation, it stops after actions are finished.

8.7　Machine Lock

Method 1: In Auto or MDI mode, press  key (the indicator  on the upper part of the edit panel lights up, it means that it has entered the machine lock state). Now each axis of the machine can not move, while the display of the coordinate is the same as that in machine operating, and M, S, T can be executed. This function is used for program check.

Method 2:　In Auto or MDI mode, press  key continuously to enter t the Tool Panel page. Press the numerical key 1 to select ON/OFF of the machine lock.

Note 1: If the machine lock function is executed in Auto or MDI mode, it is still active when the Auto mode or MDI mode are switched to the other modes (Manual, MPG, Edit, Machine zero return, Program zero), but the ON/OFF of the machine lock can not be set at this time.

8.8 M. S. T Lock

Method 1: In Auto or MDI mode, press  key (the indicator  on the upper part of the edit panel lights up, it means that it has entered the M.S.T Lock state). Now the machine carriage moves without the M, S, T code being executed. The machine lock and M, S, T lock are used together to check the program.

Method 2: In Auto or MDI mode, press  key continuously to enter the Tool Panel page. Press the numerical key 2 to select ON/OFF of the M.S.T lock.

Note 1: M00，M30，M98，M99 are executed according to the common practice.

Note 2: If the M.S.T Lock function is executed in Auto or MDI mode, it is still active when the Auto mode or MDI mode are switched to the other modes (Manual, MPG, Edit, Machine zero return, Program zero), but the ON/OFF of the M.S.T Lock can not be set at this time.

8.9 Block Skip

Method 1: In Auto or MDI mode, press  key before execution (the indicator  on the upper part of the edit panel lights up, it means that it has entered the Block Skip state). Now the program headed with “/” sign is skipped without execution.

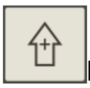
Method 2: In Auto or MDI mode, press  key continuously to enter t the Tool Panel page. Press the numerical key 5 to select ON/OFF of the Block skip.

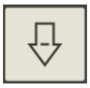
Note 1: If the block skip is executed in Auto or MDI mode, it is still active when the Auto mode or MDI mode are switched to the other modes (Manual, MPG, Edit, Machine zero return, Program zero), but the ON/OFF of the Block Skip can not be set at this time.

8.10 Ajustment of the Feedrate, Rapid Rate in Auto Mode

In Auto mode, the running speed can be altered by adjusting the feedrate override, rapid override.

In Auto mode, press  keys on the panel to select the federate override, 16-level real-time federate adjustment can be obtained.

Press the key each time, the feedrate override ascends a level (10% for each level) till 150%

Press the  key each time, the federate override descends a level (10% for each level) till 0.

Note 1: The value specified by F is adjusted by feedrate override

Actual federate = value specified by F× feedrate override

In Auto mode, press 

keys on the panel to select the federate override, which includes Fo，25％，50％，75％，100％ five gear levels. (Fo is set by the data parameter P32)

Note 2: Following is calculation form of the rapid traverse rate obtained by data parameter P21, P22 setting and rapid override adjustment.

X axis actual rapid traverse rate = value set by parameter P21*rapid override

When the rapid override is Fo, the actual rapid traverse rate is set by the data parameter P32 (Used for whole axis)

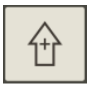
Calculation method for the actual rapid traverse rate of the Z axis is the same as above.

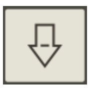
8.11 Spindle Speed Adjustment in Auto Mode

While the spindle speed is controlled by the analog voltage output in Auto mode, it can be adjusted by spindle override

In Auto mode, press 

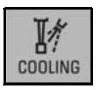
key to adjust the spindle override for spindle speed, it can realize 8-level real-time override adjustment between 50％～120％.

Press the  key each time, the spindle override ascends a level (10% for each level) till 120%.

Press the  key each time, the spindle override descends a level (10% for each level) till 50%.

The actual speed of the spindle=spindle speed of program command × spindle override

8.12 Cooling Control

Cooling ON/OFF can be shifted by pressing 

key on the panel, and its state can be viewed on the LCD screen.

CHAPTER NINE ZERO RETURN OPERATION

9.1 Concept of the Machine Zero Return

The machine coordinate system is an inherent coordinate system of the machine. The origin of the machine coordinate system is called **machine zero** (or mechanical reference point). Usually it is fixed on the positive max. Strokes of X, Z axes. The machine zero is a fixed point, which is determined after the machine is designed, manufactured, and debugged. Normally, the machine zero point can not be recognized till the CNC device is turned on, the machine zero point return is performed in Auto or by manual operation.

By setting Bit0, 1 of the parameter NO:6, it is able to set whether a machine zero point on X, Z axis according to whether there is a zero return switch fixed on the machine.

a) When the parameter NO:6 Bit0=0，Bit1=0, it is type A of zero return. The process of the reference return is as follows:

The zero return axis moves to the machine coordinate zero point (that is X0 or Z0) at the speed of G0. After reaching the zero point, the coordinate of the zero return axis is cleared, and machine zero return is finished.

b) When the parameter NO:6 **Bit0=1，Bit1=1，Bit4=0， Bit5=0,** it is **type B** of zero return. The process of the reference return is as follows:

① The zero return coordinate axis moves along with its positive direction at the rapid traverse speed. As the axis presses down the deceleration switch, the coordinate axis decelerates to the minimum speed set by the system [the speed specified by the parameter **P043** and it moves reversely at this speed until the zero block departs from the deceleration switch.

② The zero return axis moves at the minimum speed [the speed is specified by the parameter **P043** until the one-turn signal (or the signal of approaching switch zero point) of the motor encoder is detected, it continues to move a short distance forward then travels reversely at the speed specified by the system. When the one-turn signal of the motor encoder is detected again, it reaches the zero point of the coordinate axis to stop. The machine coordinate of the zero return axis is cleared and the zero return completed.

Only one block is needed as a deceleration signal in type B, zero signal is PC signal provided by drive unit.

c) When the parameter NO:6 **Bit0=1，Bit1=1，Bit4=1， Bit5=1,** it is **type C** of zero return. The process of the reference return is as follows:

① The zero return coordinate moves along with its positive direction at the rapid traverse speed. As the axis presses down the deceleration switch, the coordinate axis decelerates to the minimum speed set by the system [the speed specified by the parameter **P043** and it moves reversely at this speed.

② The system stops when the axis departs from the block, and this point is referred as the zero point. Then the machine coordinate of the zero return axis is cleared and the zero return completed. Only one block is needed as a deceleration and zero signal in type C.

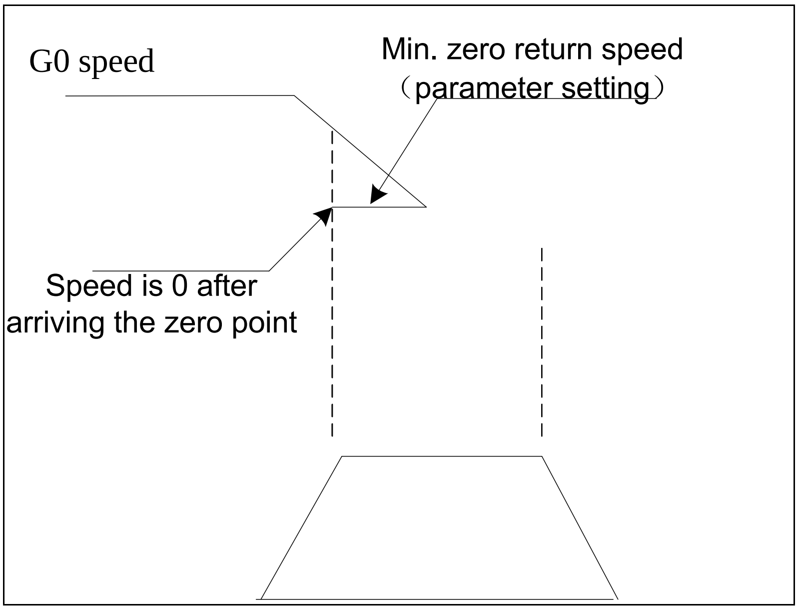
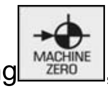


Fig.9-1-1

9.2 The Operation Procedures of Machine Zero Return

1. The procedures of type B, C

1. Enter the machine zero return mode by pressing , the “Machine Zero” is displayed on the LCD screen at this time.

2. The X, Y axis, which is to be returned the machine zero point, is selected

3. The machine moves along with its zero point direction, and it rapidly moves before decelerating. The machine moves to its zero point (it is also called as reference position) at the FL speed [The move speed is set by data parameter **P43** after the deceleration switch is pressed. The coordinate axis stops and zero return indicator lights up when the machine zero point is returned.

Note 1: The corresponding indicator lights up when the machine zero point return is completed.

Note 2: The indicator is power-off if operator moves out a corresponding axis from the machine zero point or press down the emergency button.

Note 3: Refer to the manual issued by the manufacturer for the machine zero point direction, then confirm it by Bit0 and Bit1 of the parameter N0:5. Before reference return, the tool post should be stopped in the negative position of the machine reference point, machine zero point (namely, reference point) direction. Please refer to the manual provided by the machine tool builder.

Note 4: Make sure Bit0 and Bit1 of the parameter N0:6 are set to 0 before the machine reference switch is

installed on the machine. Select machine zero return type A, but do not perform zero return. Otherwise, accidents will occur because the tool post moves rapidly for there is no deceleration switch.

2. The procedures of type A

1. Press 

key, press  or  key to enter 【**Integrated**】position page. By pressing “X” key, X axis coordinates on the screen will be highlighted, and then press the 

key. Machine zero return completes and indicator 

lights up.

Note: Zero return operations of the Z axis are the same as above.

9.3 Concept of Program Zero Point

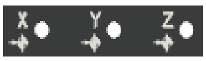
To facilitate programming, programming personnel use work piece coordinate system (also called part coordinate system). A point is selected as the program zero point (This point is specified by G50 code and it is called program zero point), and a workpiece coordinate system is setup.

9.4 The Operation Procedures of Program Zero Return

1. Press 

key to enter the Program Zero mode, the lower right corner of the screen displays “Machine Zero”.

2. Select X and Z axes that to be returned to program zero point.

3.The machine moves toward the program zero, and the axis stops with program zero indicator 

lighting up after the axis returns to the program zero.

Note: G50 should be executed after system is powered on, otherwise, ＃021 alarm occurs.

CHAPTER TEN SYSTEM COMMUNICATION

Serial terminal port software applies to **GSK980TA3 C**NC system and bi-directional data conversion between computers.

- Operating environment：Windows 98/ Windows ME / Windows 2000/ Windows XP
- Interface standard：RS232
- Max. transmission rate：115200 bit/s
- Optimum transmission rate：9600 bit/s

Version characteristics:

A computer can communicate with a NC system only.

Maximum transmission rate 115200bit/s (Corresponding setting for the system is needed)

Hardware connection:

Before the file transmission, connect PC machine serial port to the NC system communication port. Make sure connection is correct, and file transmission can be performed after the CNC operates normally.

10.1 Serial Port Communication

Serial port communication software is window interface, which is used for file transferring/receiving from PC end to CNC end. This software can be applied to Win98. WinMe. WinXP and Win2K.

10.1.1 Program Start

Communication software program is performed directly. Example: Comm980TA3.exe. The interface is displayed after the program starts, refer to the following figure:

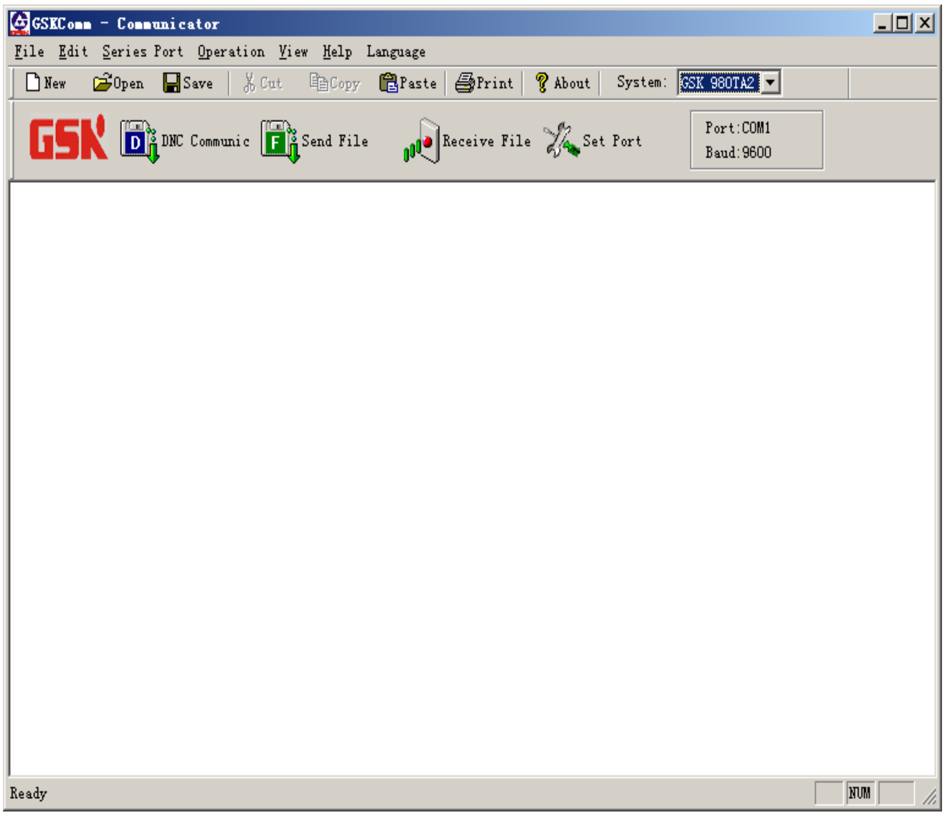


Fig. 10-1-1-1

10.1.2 Function Instruction

1. File menu

File menu includes Add Files, Open, Save List, Print and Print Setting, Recently opened file lists, etc. functions.

2. Edit menu

Edit menu includes Cut, Copy, Paste, Undo, Search, Replace, etc. functions.

3. Serial port menu

This is mainly used for opening and setting serial port.

4. Transmission mode menu

It includes DNC transmission mode (it is not used temporarily), File Sending Mode, File Receiving Mode.

5. View menu

Display and concealment of both toolbar and status bar.

6. Help menu

Version information for the software.

10.1.3 Software Usage

1. Methods of sending files

1) Open and set a serial port. Baud rate can be set to 9600, 57600 or 115200.

2) After "send file" menu of the transmission mode menu is opened or the button send file 

of the toolbar is pressed, the dialogue box will be popped up as below:

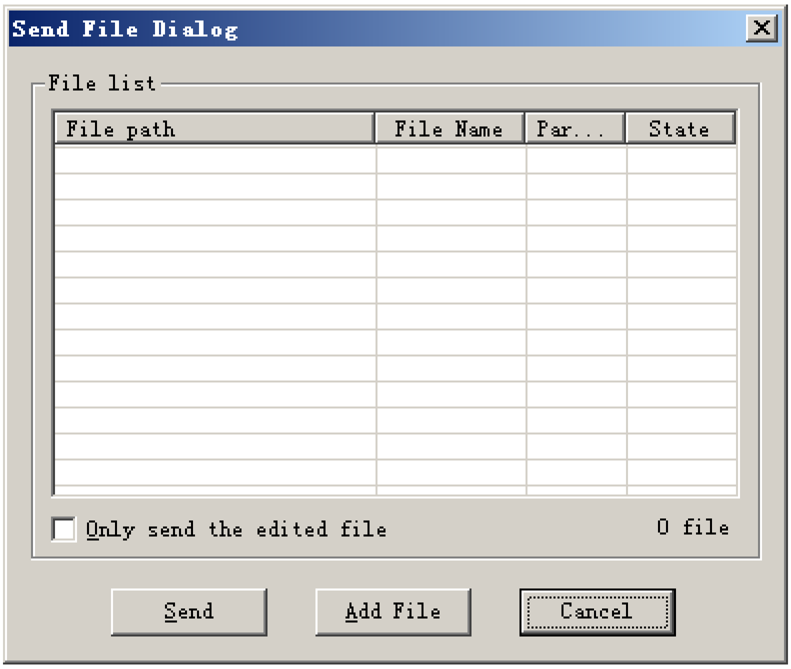


Fig. 10-1-3-1

3) Select "add files" button, the dialogue box "select zone" is displayed:

Fig. 10-1-3-2

Program files can be only sent to "user zone", while the system configuration files to the system zone. Otherwise, the files can not be identified by the system. System manufacturer of machine manufacturer authorities are necessary for sending system configuration files or backup files. The corresponding password can be input on "password set" of the Setting page on the CNC end.

4) The dialogue box appears after zone is selected, press Shift or Control key to select multiple files: maximum 399 files can be selected.

5) Click "open" button after the files are selected, then return to "send file" dialogue box.

6) Names of the programs to be sent to the user zone must be headed with the letter “O” and followed by four-digit number. Otherwise, the dialogue box will prompt the user to change the name.

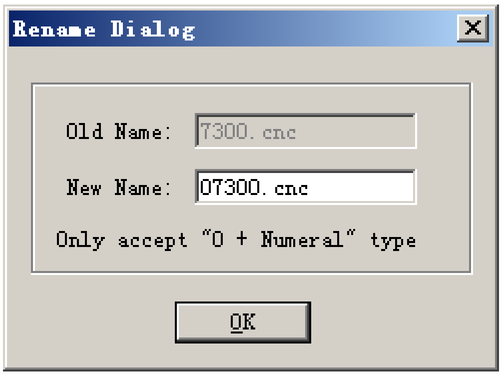


Fig. 10-1-3-3

7) Return to “send file” dialogue box, and click “send” key. The file is transferred and the dialogue box will be shown as bellow:

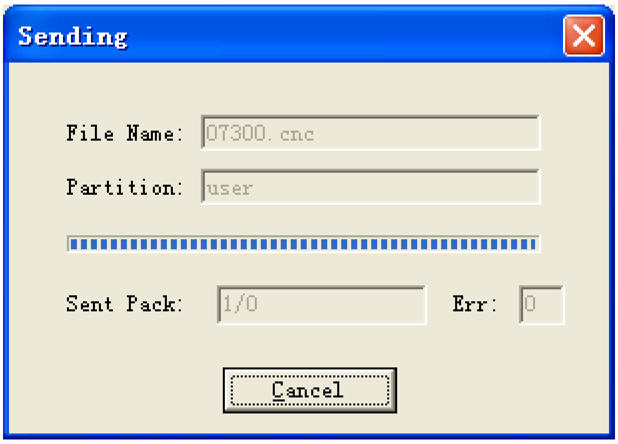


Fig. 10-1-3-4

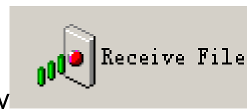
8) Transmission complete

Note: If the program transmission is performed when the system works, it will affect normal operation of the system.

2. Methods of receiving files

1) Enter into the data interface of the setting page, Move the contents to the table of “data output” by red mark of the direction keys.

2) Open and set the serial port, baud rate can be set to 9600, 57600 or 115200.

3) Open “receive file” of the transmission menu or press receive file key 

, the dialogue box will be popped up as below:

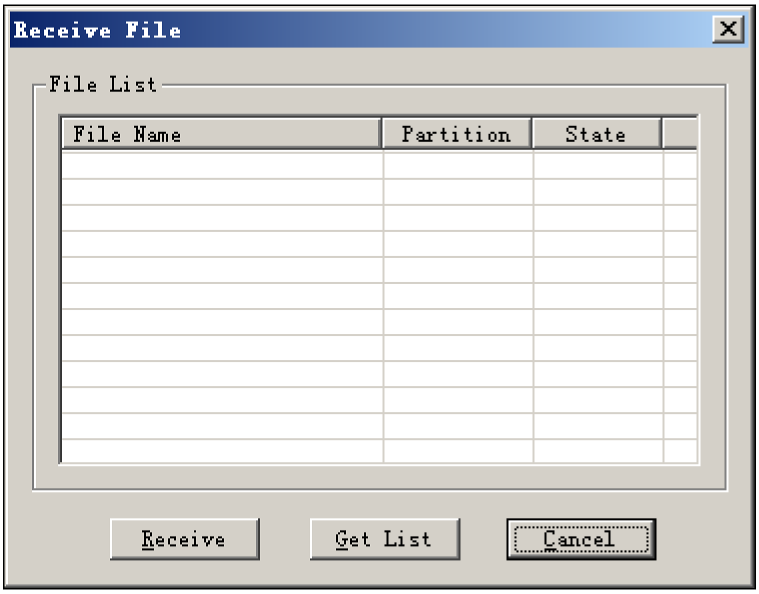


Fig. 10-1-3-5

4) Press “obtain list” key to get file list of the CNC.

5) Select the file to be transferred. Multiple files can be selected by pressing Shift or Ctrl key.

6) Click “start receiving” key, select file storage address and then the dialogue box is popped up as bellow:

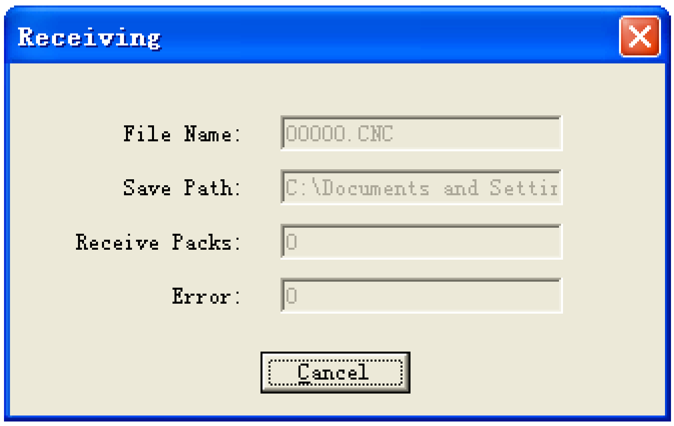


Fig. 10-1-3-6

7) Transmission completes.

Note: If the program transmission is performed when the system works, it will affect normal operation of the system.

10.1.4 Communication between CNC and CNC

Two CNC systems can communicate with each other. Take 980TA3 for an example as bellow.

1. Mutual program communication between two CNC systems.

(1) Connect two 980TA3 systems with communication line.

(2) After two 980TA3 systems are powered on, set identical baud rates for them, e.g. 9600.

(3) In the system 980TA3 to send programs, press edit key to enter the edit mode, select the program name (Supposing it is O8888), press the address key “O” and the numerical key, then wait for the system displaying communication success and the communication completes.

2. Mutual parameter communication between two CNC systems

Connect two GSKGSK980TA3 systems with communication line, and turn on parameter switch after the systems power on. The following operations are performed on the system to send parameters.

Select the Edit mode and enter the parameter interface. Press DATA OUTPUT key on the panel, and now “communicating” is displayed in the lower part of the screen (If the CNC systems are not connected well, alarm “the port is not connected” occurs in the lower part of the interface). Communication is finished after the characters disappear. If alarm “communication error” occurs, please check whether the baud rates of the two GSK980TA3 systems are consistent or lower the transmission baud rates of the ports. After communication, the parameters of the GSK980TA3 system that receives parameters are updated, now it is necessary to power on again. When the screw-pitch compensation parameters to be sent, enter the screw-pitch compensation parameter interface, and the operation steps are the same as that of the parameter transmission.

10.2 USB Communication

10.2.1 Overview and Cautions

1. Because the U-disk files is entirely separated from the CNC system, you had better not enter into the U-disk system in operation. Otherwise, the problem of work piece damage may occur.
2. The U-disk file system supports repeatedly hot plug and play. However, do not plug the U-disk before power-on to avoid failing to identify the U-disk. Plug it after entering the U-disk operation interface.
3. To avoid data in U-disk is not performed completely, pull out the U-disk when the indicator is not flash or wait a little while.
4. Only txt file with a name “O+ four digits” can be displayed, e.g. “O0001”.

10.2.2 Enter U-disk

1. After the CNC system is powered on, press down “setting” key, and press “D” to turn on the parameter switch, then the system enters the state of alarm. By pressing edit key, the system is shifted to the edit mode, and press the program key twice, the system enters to USB operation interface.
2. Plug in U-disk after the system enters the U-disk program.

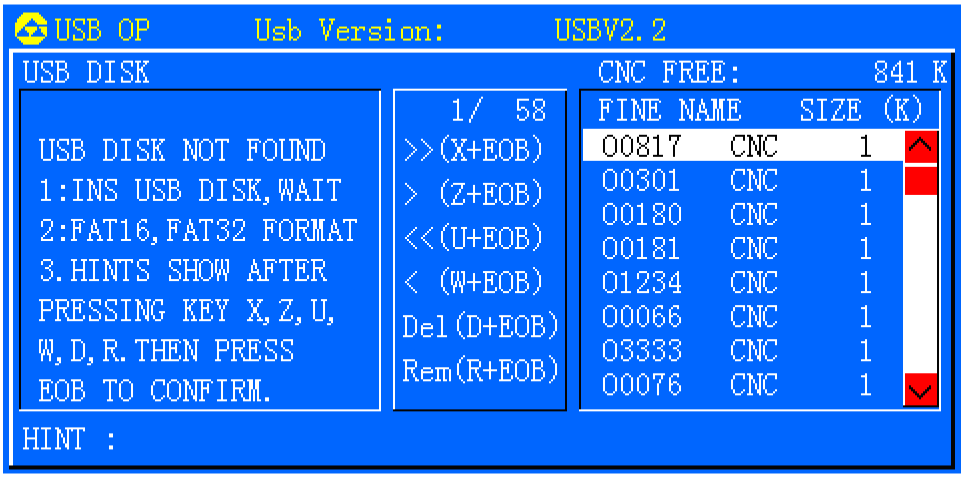


Fig. **10-1-2-1**

10.2.3 Procedures

Plug in U-disk, the system will automatically read U-disk, all O????.CNC programs on root directory of the USB disk and files: "PARA3.txt","PARA4.txt ","WORM2.txt ","WORM3.txt ","WORM4.txt ". By pressing left, right arrows, the cursor can be shifted between CNC memory and USB memory. And the file on the current memory can be selected by pressing up and down keys.

1. Copy CNC program from U-disk to the user disk

1.1 Copy all CNC programs from U-disk to the user disk

a. Press left and right arrows to display U-disk files.
b. After pressing “X” key, the interface prompts that: copy all files from USB to the CNC system. This operation can be cancelled by pressing left and right arrows. Press the <enter> key to start the duplication. If same file names exist, the system will prompt that: files have the same name, O???? CNC interchange（IN）,cancel（CAN）, all（ALL）. Select the one needed.
c. After duplication is finished, the system will prompt: copy completed.

1.2 Copy a CNC program from U-disk to the user disk

a. Press left and right arrows to display U-disk files.
b. Select files on the U-disk by pressing up and down keys.
c. After pressing “Z” key, the interface prompts that: copy the files selected from USB to the

CNC system. This operation can be cancelled by pressing left and right arrows. Press the <enter> key to start the duplication. If same file names exist, the system will prompt that: “files have the same name” O????　CNC interchange（IN）,cancel（CAN）, all（ALL）. Select the one needed.

d. After duplication is finished, the system will prompt: copy completed.

2. Delete files from the U-disk

2.1 Delete all files from U-disk

a. Press left and right arrows to display U-disk files.
b. After pressing “R” key, the interface prompts that: “delete all files from USB”. This operation can be cancelled by pressing left and right arrows. Press the <enter> key to delete the files.
c. There is no files in the U-disk after they are deleted.

2.2 Delete a file from the U-disk

a. Press left and right arrows to display U-disk files.
b. Select CNC files in the U-disk by pressing up and down keys.
c. After pressing “R” key, the interface prompts that: “delete the files selected in the USB”. This operation can be cancelled by pressing left and right arrows. Press the <enter> key to delete the files.
d. This file is not in the U-disk after it is deleted.

3. Copy CNC programs from the system user disk to the U-disk

3.1 Copy all CNC programs from the user disk to the U-disk

a. Press left and right keys to the lists of the CNC.
b. After pressing “U” key, the interface prompts that: copy all CNC programs from the user disk to the U-disk. This operation can be cancelled by pressing left and right arrows. Press the <enter> key to copy the files. If same file names exist, the system will prompt that: files have the same name, O????　CNC interchange（IN）,cancel（CAN）, all（ALL）. Select the one needed.
c. After duplication is finished, the system will prompt: copy completed.

3.2 Copy a CNC program from user disk to the U-disk

a. Press left and right keys to the lists of the CNC.
b. Press up and down keys to select CNC files in the CNC memory.
c. After pressing “W” key, the interface prompts that: copy the selected program from the CNC to the USB system. This operation can be cancelled by pressing left and right arrows. Press the <enter> key to copy the files. If same file names exist, the system will prompt that: files have the same name, O????　CNC interchange（IN）,cancel（CAN）, all（ALL）. Select the one needed.

d. After duplication is finished, the system will prompt: copy completed.

4. Delete files from the system user disk

4.1 Delete files from the system user disk

a. Press left and right keys to the lists of the CNC.
b. After pressing “R” key, the interface prompts that: delete all files from the CNC. This operation can be cancelled by pressing left and right arrows. Press the <enter> key to delete the files.
c. There is no files in the U-disk after they are deleted.

4.2 Delete a file from the user disk

a. Press left and right keys to the lists of the CNC.
b. Select CNC files in the CNC memory by pressing up and down keys.
c. After pressing “D” key, the interface prompts that: delete the selected file from the CNC. This operation can be cancelled by pressing left and right arrows. Press the <enter> key to delete the file.
d. This file is not in the CNC memory after it is deleted.

10.2.4 U-disk Removal

1. Pull out the U-disk until the indicator is not flash.
2. Press the screen operation keys to enter the system, or press reset key to cancel the alarm and enter the CNC system.

PART FOUR CONNECTION

CHAPTER ONE SYSTEM CONFIGURATION & INSTALLATION

1.1 System Configuration

GSK980TA3/TB3-V systems include many units as follows, see figure 1-1-1 for the system configuration.

（1）GSK980TA3 CNC system

（2）Additional operation panel

（3）Digital AV servo drive unit (Stepping motor drive unit)

（4）Servo motor (Stepping motor)

（5）Isolation transformer

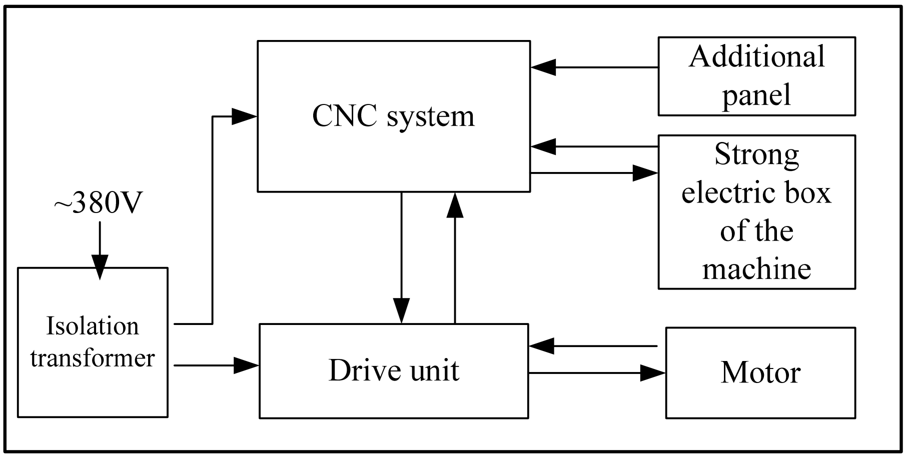


Fig. 1-1-1

1.2 System Installation

Check whether the units such as CNC system, drive unit, motor and photoelectric encoder etc. to be installed are complete, intact and the one you ordered.

The system must be installed securely and keep a certain space around it for heat dissipation. The CNC should be mounted at a place where is convenient for operation without chip and cooling.

Strong and weak current shall be separated. The isolation transformer provides power for the CNC and the drive unit. It should be separated from the strong current part. Each signal line shall keep as far away from the AC contactor to reduce interference. It is better to connect the photoelectric encoder, limit signal and emergency stop signal to the system directly, but not to strong electric box. Power supply must be strictly grounded.

Fix each connector and screw securely. It is not allowed to plug in or pull out the connector after system power on.

Prevent the panel from damaging by hard and sharp tools when installing the CNC system. When painting, remove the operation panel to avoid the paint stain it.

Ensure there is no strong current, strong magnetic interference source around the CNC system, and keep it far away from inflammable and explosive materials or other dangerous objects.

1.3 Mounting Dimension of the CNC System

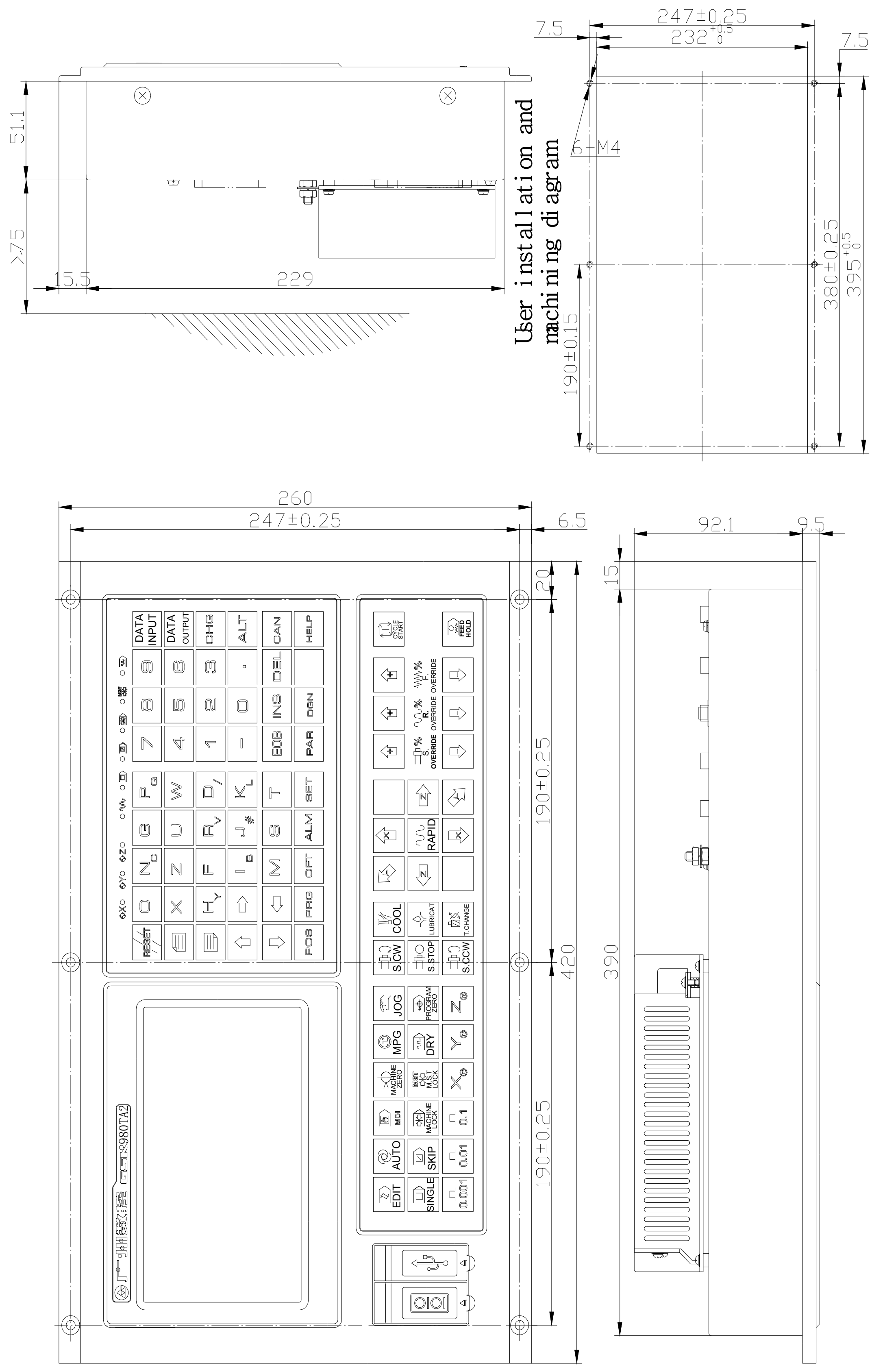


Fig. 1-3-1 Mounting dimension of the system GSK980TA3

1.4 Additional Panel

User can select additional panel for this system and define the functions such as emergency stop, program lock, system power on/off, feed hold, cycle start, MPG etc. for the expansion hole of the panel. The optional accessories for the system GSK980TA3 are as follows:

Communication elements: communication cable 5m×1, communication software installation CD ×1.

Electronic MPG: MPG60-T1-100B/05（match with AP01 additional panel）or MPG80-T1-100B/05（match with AP02 additional panel）

Additional panel: AP01（aluminum alloy 420mm×71mm）, it can be fixed bellow the panel

AP02（aluminum alloy 100mm×260mm）, it can be fixed at the side of the panel

Emergency stop button: LAY3-02ZS/1（GSK980TA3 have fixed the emergency button）

Non-self-lock button: KH-516-B11 (Green or red)

Self-lock button: KH-516-B21 (Green or red)

Note 1: Communication function is the standard function configuration, but communication elements are optional accessories.

Note 2: Optional accessories are supplied as product accessories (unset and unconnected). Please note on the order if installation and connection are necessary.

Additional panel AP01

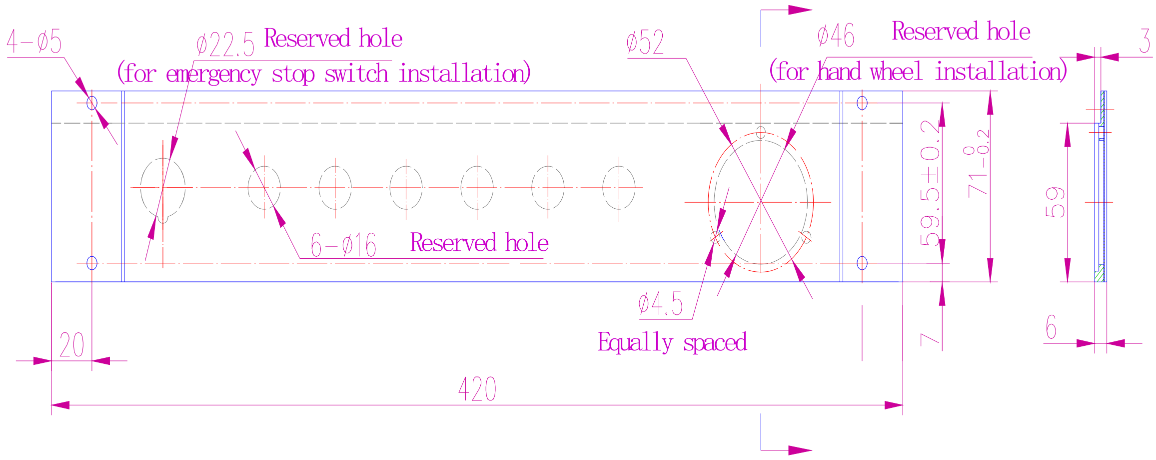


Fig. 1-4-1

Additional panel AP02

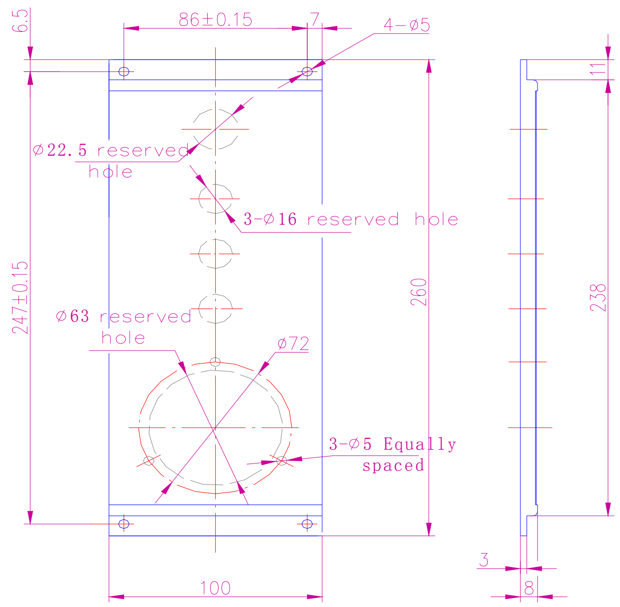


Fig. 1-4-2

CHAPTER TWO　EQUIPMENT CONNECTION

2.1　System Interfaces

2.1.1　Interface Layout

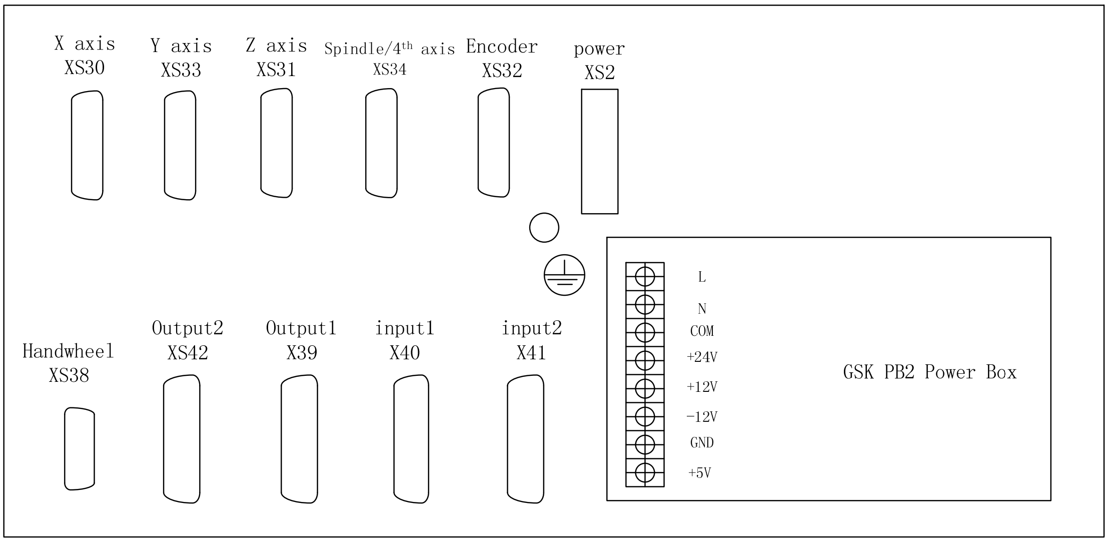


Fig.2-1-1-1　GSK980TA3 back view

2.1.2　Explanations for Interfaces

1）Power interface　（+5V. +24V. 0V）

5 terminals for the system and power interface.

2）XS30，XS31，X33 are drive interface of the X, Z, Y axis respectively.

15-core D type socket (female)

3）XS32 spindle encoder interface (incremental pulse encoder, 1024 or 1200 wires)

15-core D type socket (male)

4）XS36 general-purpose PC machine interface (RS232 port)

9-core D type socket (female)

5）XS34 spindle interface

26-core D type socket (female)

6）XS38 MPG interface

26-core D type socket (male)

7）XS39, XS40 input/output interface

XS39 is 25-core D type socket (female), XS40 is 25-core D type socket (male)

8）XS41 ，XS42 extended input /output interface

XS41 is 25-core D type socket (male), XS40 is 25-core D type socket (female)

2.1.3 Connection Diagram

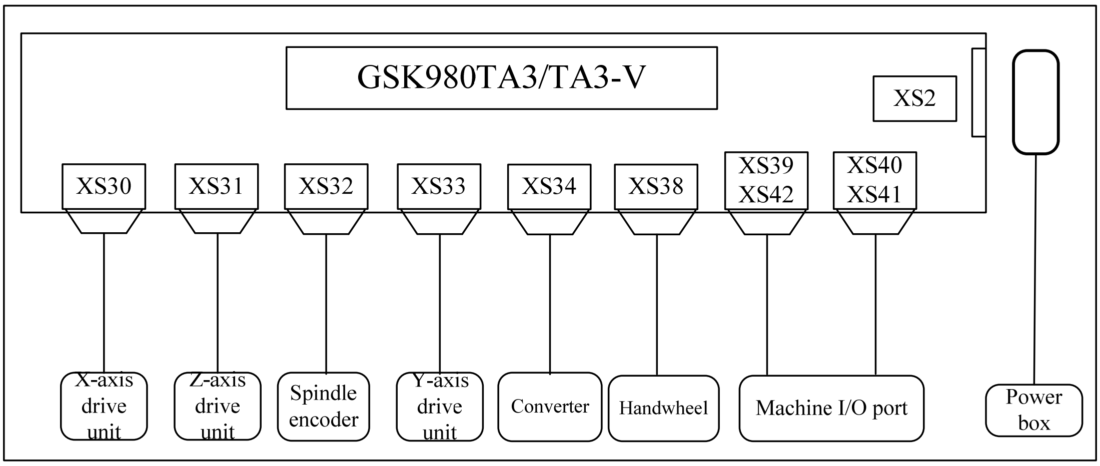


Fig. 2-1-3-1

2.2 Connection of the System to the Drive Unit

2.2.1 Connection to the Drive Unit

1) Interface diagram of PC side

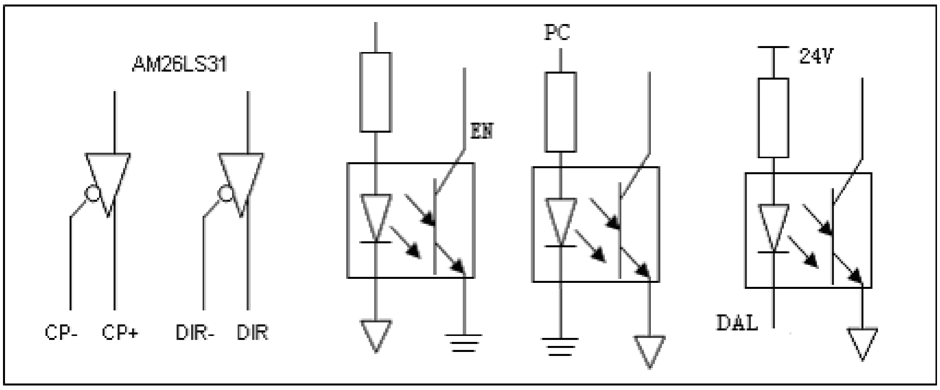


Fig. 2-2-1-1

2) Definition of the interface signal

Part Four Connection

XS30: DB15F (X-axis)

1	XCP+	9	XCP-
2	XDIR+	10	XDIR-
3	XPC	11	0V
4	+24	12	+5V
5	XDALM	13	+5V
6	*XSET	14	0V
7	XEN	15	0V
8	0V		

XS31: DB15F (Z-axis)

1	ZCP+	9	ZCP-
2	ZDIR+	10	ZDIR-
3	ZPC	11	0V
4	+24	12	+5V
5	ZDALM	13	+5V
6	*ZSET	14	0V
7	ZEN	15	0V
8	0V		

Signal	Description
nCP+. nCP-	Code pulse signal
nDIR+. nDIR-	Code direction signal
nPC	Zero signal
nDALM	Drive unit alarm signal
nEN	The system ready signal
nSET	Setting signal

Fig. 2-2-1-2

2.2.2 Signal Explanation

1. Mono-pulse movement commands signal

XCP+，XCP-，ZCP+，ZCP- are instruction pulse signal, XDIR+，XDIR-，ZDIR+，ZDIR- zre movement direction signal. Two groups of signals are differential output signals.

Connection diagram is as follows:

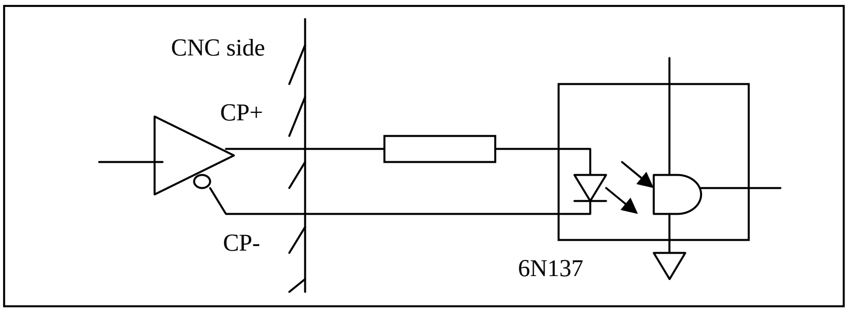


Fig. 2-2-2-1

2. Alarm signal of the drive unit (input)

The receive mode of the signal at the system side is as follows. By setting the parameter (008), it is able to set whether low level “0” or high level “1” is drive unit fault.

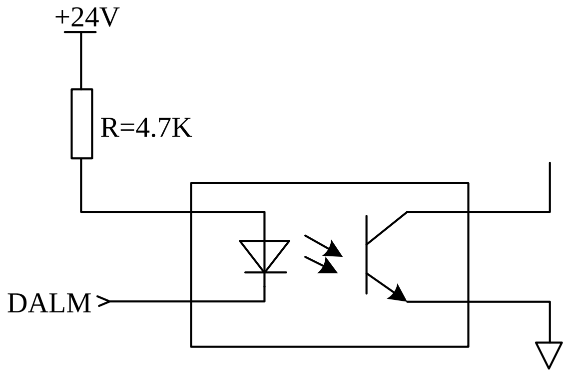


Fig. 2-2-2-2

3. CNC system ready signal EN (contact output)

Closed contact indicates the CNC system is ready. When the system alarm, the contact signal will be cut off.

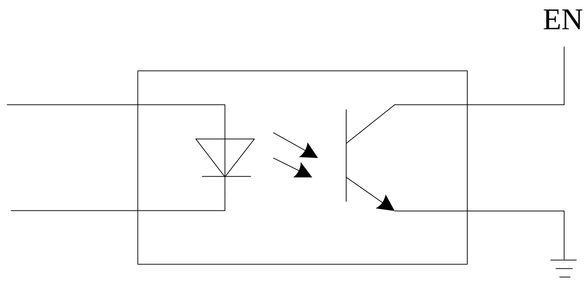


Fig. 2**-2-2-3**

4. Set signal *SET (output)

When there are movement commands output from the system, it is high level signal. When there is no movement command output from the system, it is low level signal.

5. Signal PC is used for returning to the reference point

The receive mode of the signal at the system side is as follows.

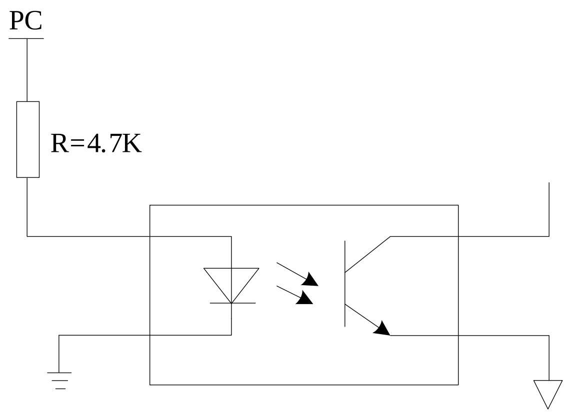


Fig. 2-2-2-4

Note 1: If PC signal output by the drive unit is 5V, wire jumper makes SR3028，SR3030，SR3032（510Ω）on. If PC signal output by the drive unit is 24VV, wire jumper makes SR3027，SR3029，SR3031（4.7KΩ）on.

Note 2: The user should provide the PC signal, which waveform is as follows.

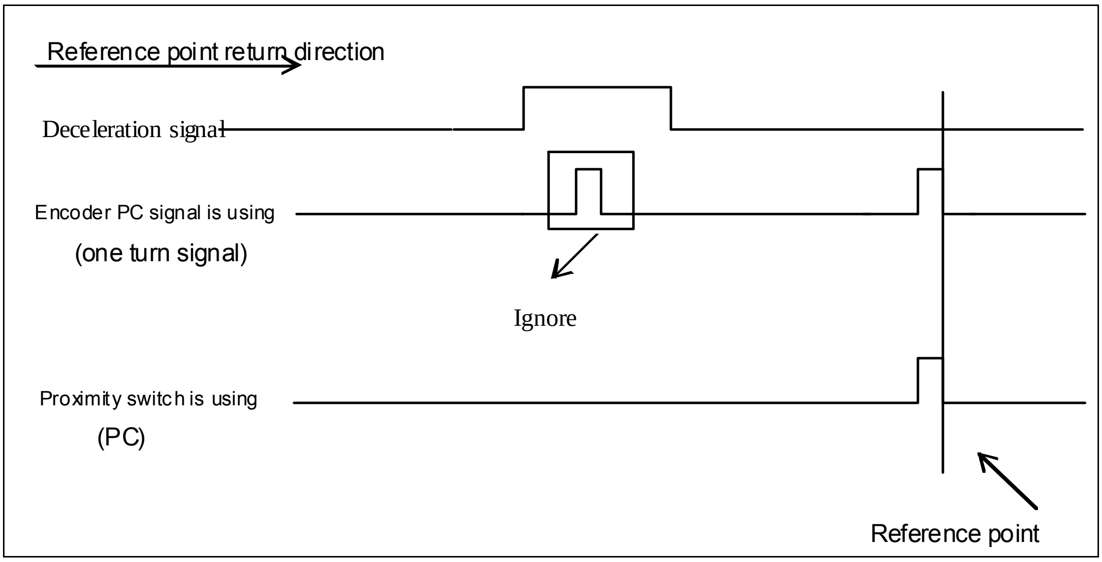


Fig. 2-2-2-5

6）Connection methods for a proximity switch that is taken as a deceleration switch and a zero return switch.

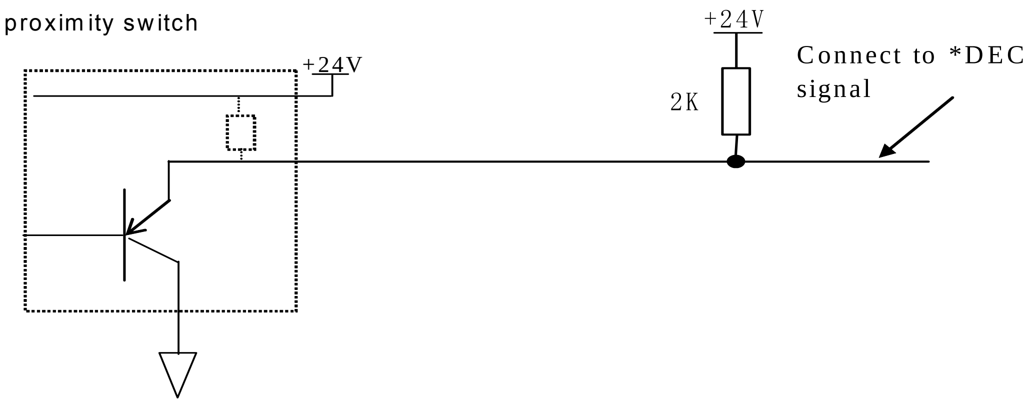


Fig. 2-2-2-6

Note: When this zero return mode is used, ZCX, ZCZ of the parameter should set to "1".

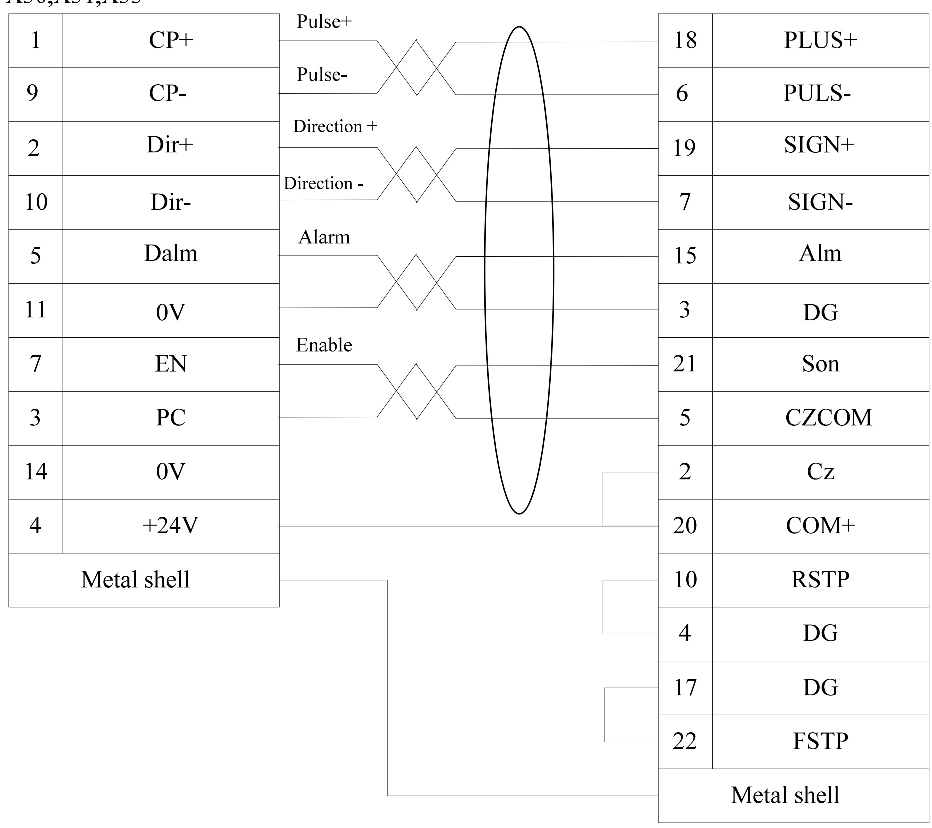
Part Four Connection

2.2.3 Connection Between the System and the Drive Unit

GSK 980TA3/TA3-V connect to DA98 drive unit

GSK 980TA3/TA3-V (X, Y,Z axis) X30,X31,X33

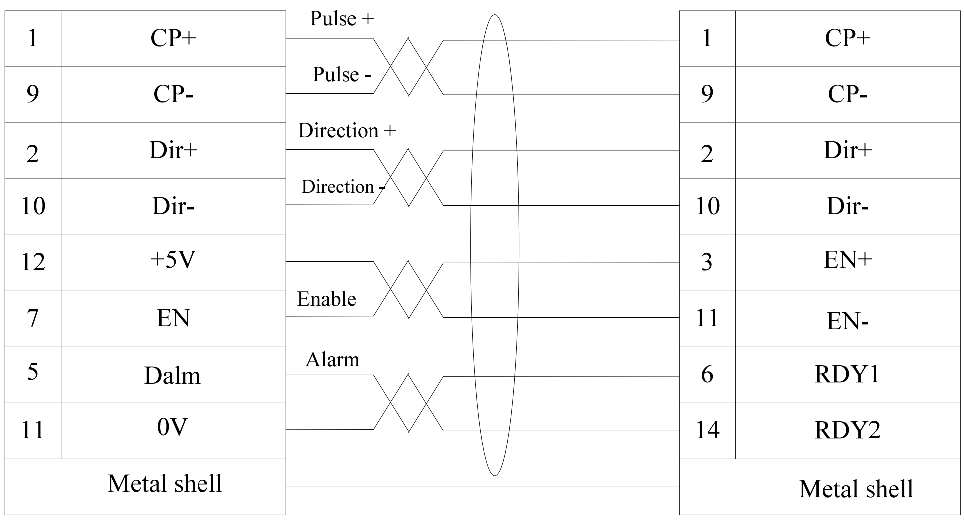
DA98 drive unit signal interface



GSK 980TA3\TA3-V connect to DY3 drive unit

GSK 980TA3\TA3-V (X, Y,Z axis) X30,X31,X33

DY3 drive unit signal interface



Part Four Connection

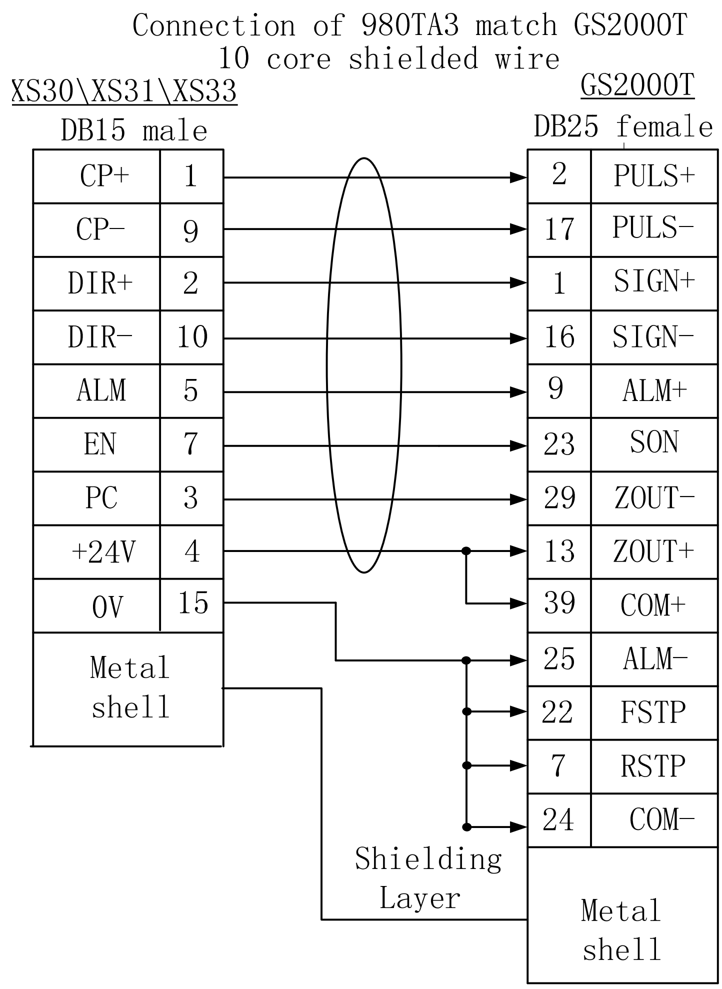


Fig. 2-2-3-1

2.3 Connection to the Spindle Encoder

1024-line or 1200-line incremental position encoder can be used with the system. *PCS，PCS，*PBS，PBS， *PAS，PAS bellow correspond to $\overline{Z}$，Z，$\overline{B}$，B，$\overline{A}$，A respectively.

XS32: DB15F (Spindle encoder)

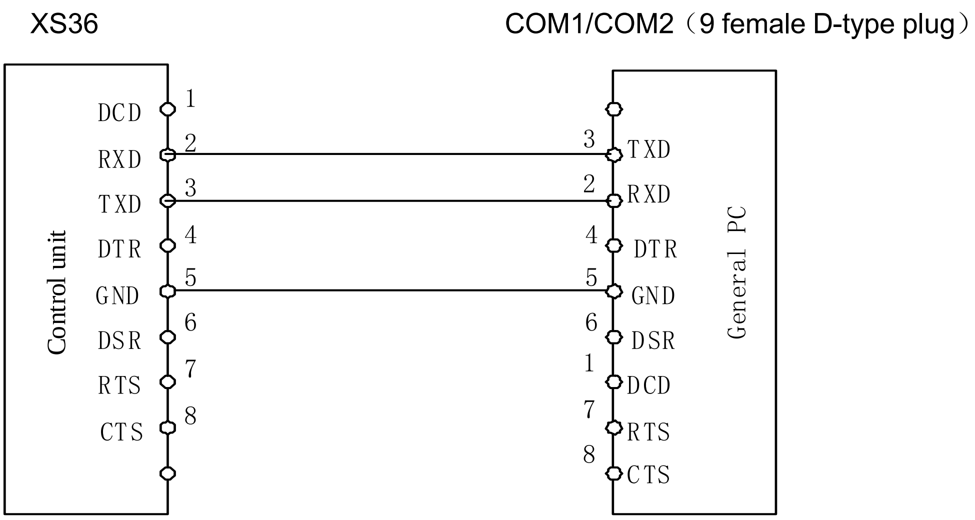
1		9	
2		10	
3	*PCS	11	0V
4	PCS	12	+5V
5	*PBS	13	+5V
6	PBS	14	0V
7	*PAS	15	0V
8	PAS		

Signal	Explanation
PAS, *PAS	Encoder A-phase pulse
PBS, *PBS	Encoder B-phase pulse
PCS, *PCS	Encoder Z-phase pulse

Fig. 2-3-1

2.4 RS232-C Standard Serial Port (Optional)

The CNC system can communicate with PC machine via RS232-C serial port (Special communication software shall be installed). Connection diagram is as follows.



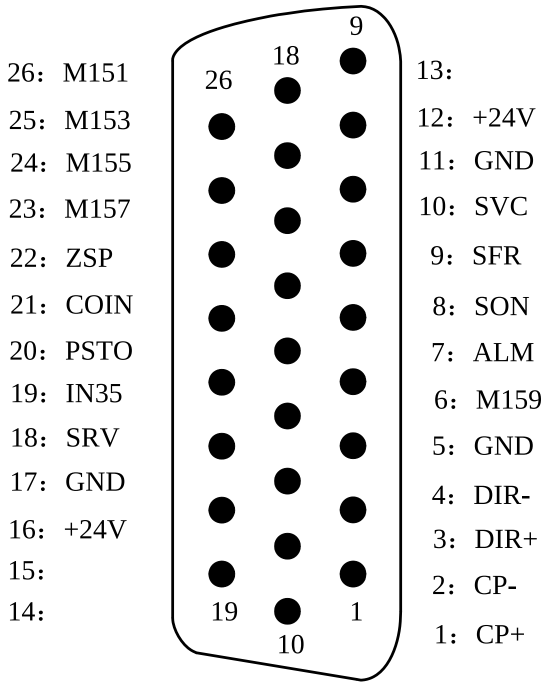
Signal	Explanation
RXD	Data receiving
TXD	Data sending

Fig. 2-4-1

Note: Weld the shielding layer to the metal of the plug.

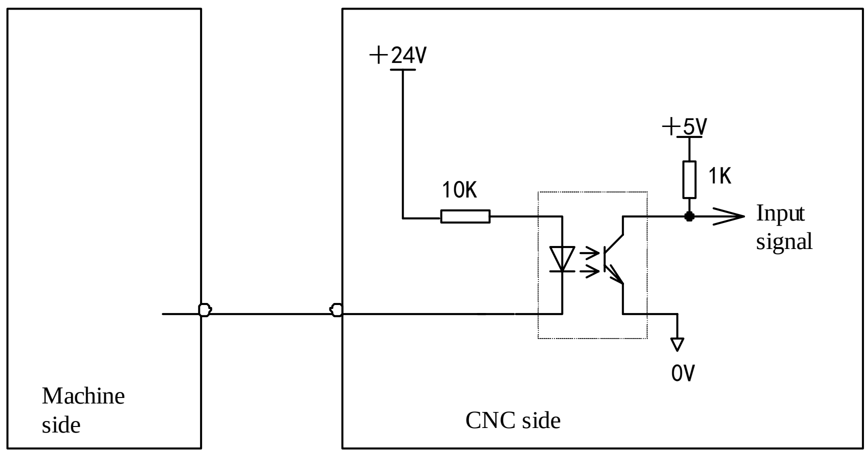
2.5 Spindle Interface

XS34 spindle/4th interface
DB26 female

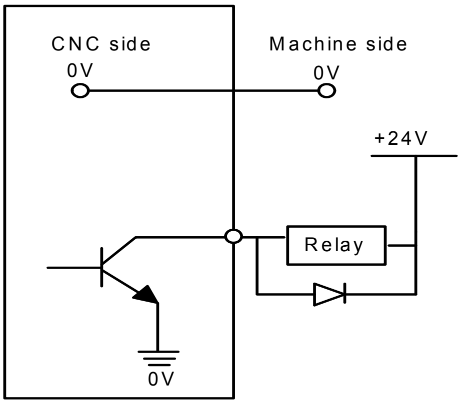


Signal	Explanation
CP4+、CP4-	Spindle pulse signal
DIR4+、DIR4-	Spindle direction signal
SVC	Analog voltage output
M151	Spindle orientation
M153	Zero speed clamp
M155	Speed/pos switch
M157	M157
M159	M159
SON	Spindle enable signal
SFR、SRV	Spindle CW/CCW
ZSP	Spindle zero speed check
COIN	Spindle orientation finish
PSTO	Spindle speed/pos switch finish
IN35	Spindle clamp input
ALM	4th axis/spindle abnormal alarm
+24V	+24V
GND	0V

Input signal cicuit of spindle interface (XS34) :



Onput signal cicuit of spindle interface (XS34):



2.5.1 980TA3 Connection Drawing & Normal Relay

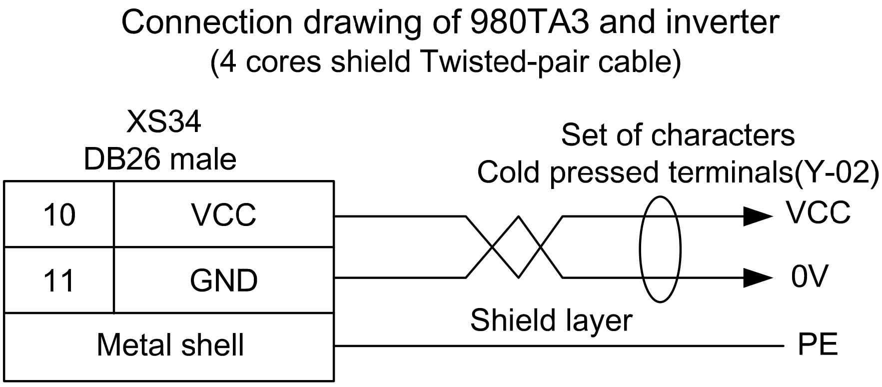


Fig.2-5-1

Note: **Weld the shielding layer to the metal of the plug.**

2.5.2 980TA3 Connection Drawing & Spindle Driver DAP03(Require Position & Speed Control Function)

980TA3 match DAP03 spindle signal wire (Dividing pos function)

DAP03 DB44 female

Connector	Signal	Pin	Direction	DAP03 Pin	DAP03 Signal	Description
XS32 DB15 female	A+	8	←	16	PA+	Signal of A phase
	A-	7	←	1	PA-	
	B+	6	←	17	PB+	Signal of B phase
	B-	5	←	2	PB-	
	Z+	4	←	18	PZ+	Signal of Z phase
	Z-	3	←	3	PZ-	
XS34 DB26 male	GND	11	→	15	VCMD-	Analog voltage output
	SVC	10	→	14	VCMD+	
	SON	8	→	24	SON	Spindle enable signal
	CP+	1	→	42	PULS+	Pulse signal
	CP-	2	→	28	PULS-	
	DIR+	3	→	33	SIGN+	Direction signal
	DIR-	4	→	34	SIGN-	
	ALM	7	←	7	ALM	Spindle abnormal alarm
	SFR	9	→	9	SFR	Spindle CW
	SRV	18	→	25	SRV	Spindle CCW
	+24V	16	→	37	COM+	
	0V	17	→	35	COM-	
	M151	26	→	11	STAO	Spindle orientation
	M153	25	→	27	ZSL	Zero speed clamp
	M155	24	→	12	VP	Speed/pos switch
	ZSP	22	←	20	ZSP	Spindle zero speed check
	COIN	21	←	21	COIN	Spindle orien. finish
	PSTO	20	←	44	VPO	Speed/pos switch finish
	Metal shell		Shield layer	Metal shell		

Fig.2-5-2

2.5.3 980TA3 Connection Drawing & Spindle Driver GS3000Y-N (Require Position & Speed Control Function)

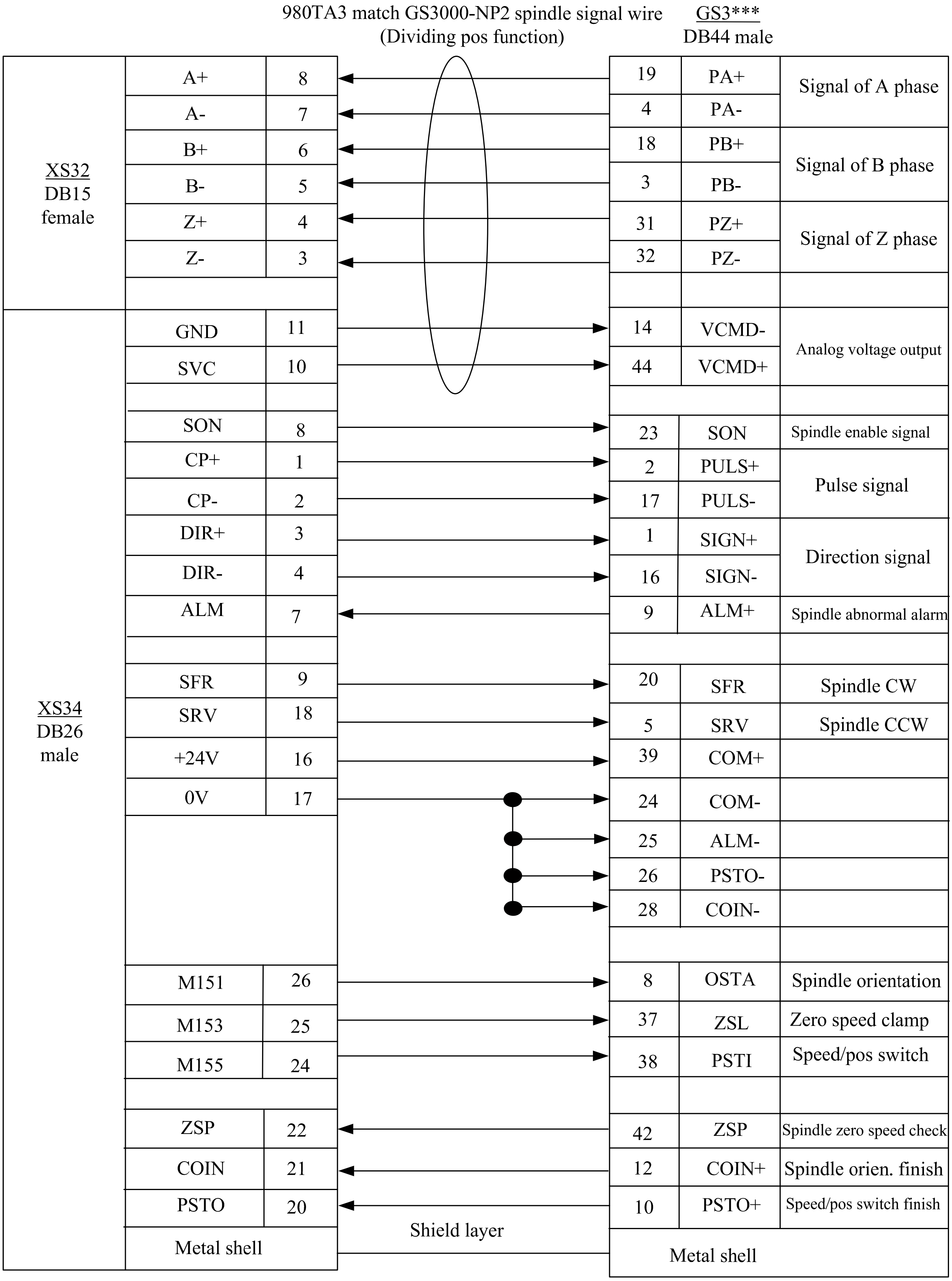
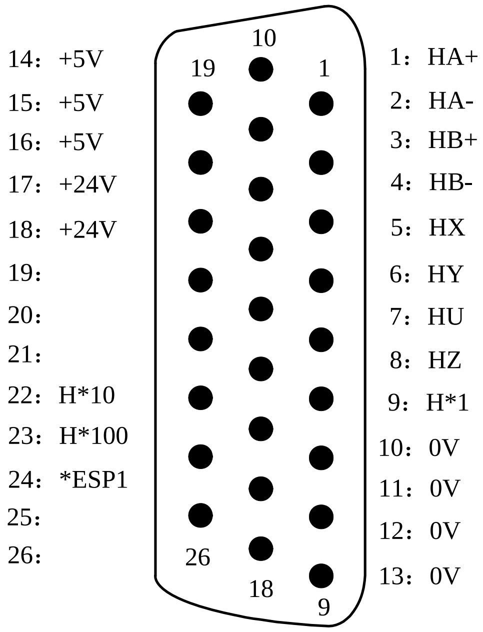


Fig.2-5-3

2.6 MPG Interface

XS38 handwheel/MPG interface
DB26 male



Signal	Explanation
HA+、HA-	Signal of MPG A phase
HB+、HB-	Signal of MPG B phase
HX	X handwheel axial
HY	Y handwheel axial
HU	U handwheel axial
HZ	X handwheel axial
H*1	Increment*1
H*10	Increment*10
H*100	Increment100
*ESP1	Emergency stop signal
+24V	+24V
+5V	+5V
0V	0V

2.6.1 Normal MPG Connection Drawing

Connection drawing of GSK 980TA3 and single MPG

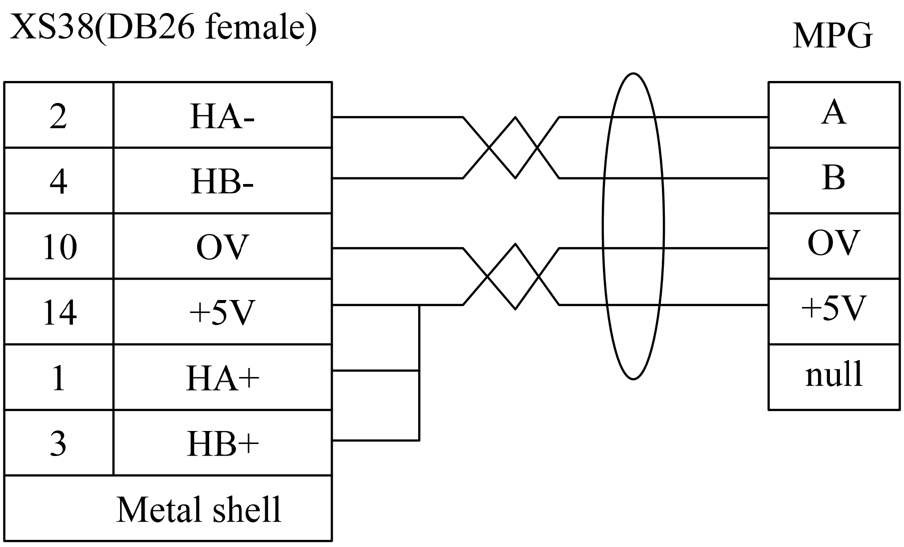
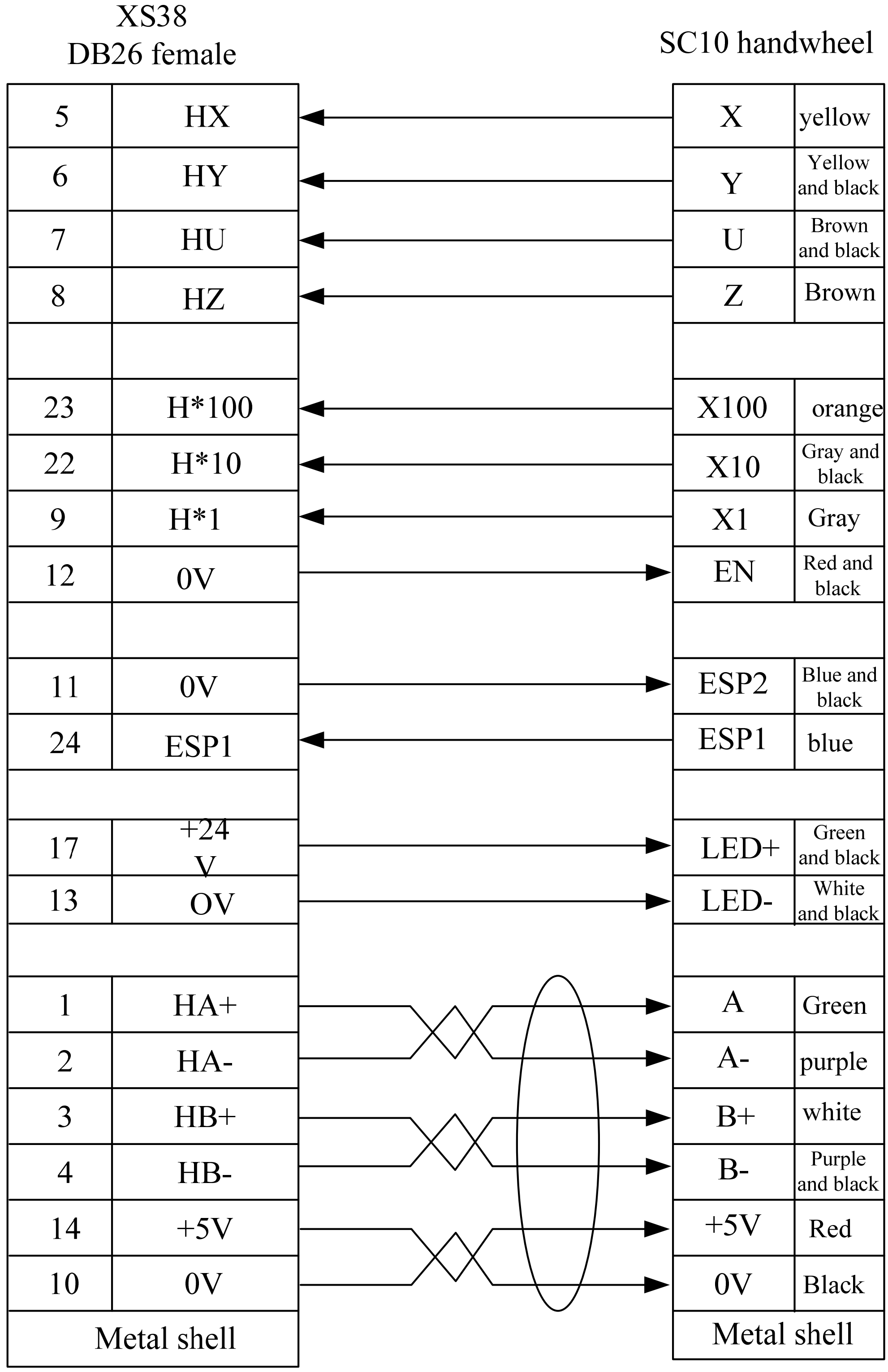


Fig.2-6-1

2.6.2 Connetion drawing of handwheel (match MPG GSKSC10)

Connection drawing of GSK980TA3 and handwheel GSKSC10



Handwheel parameter set:

1. Set No.1 sys parameter BIT3 to be 1, select MPG mode.
2. Set No.7 sys parameter BIT6 to be 1, select external MPG mode.
3. When match row tool post, it is need to set No.81 to be 1.

2.7 Connection of the Power Interface

GSKPB2 power is applied in the systems GSKGSK980TA3, which involves three groups of input voltage:

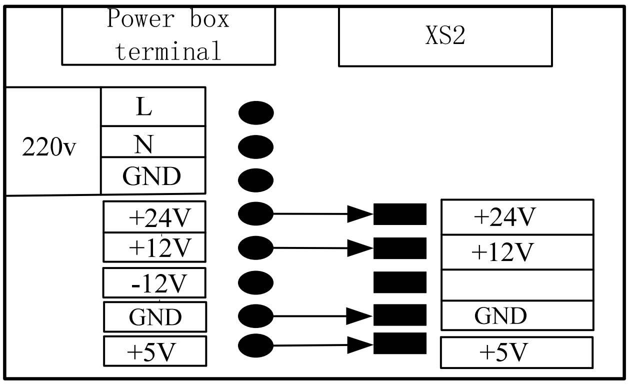


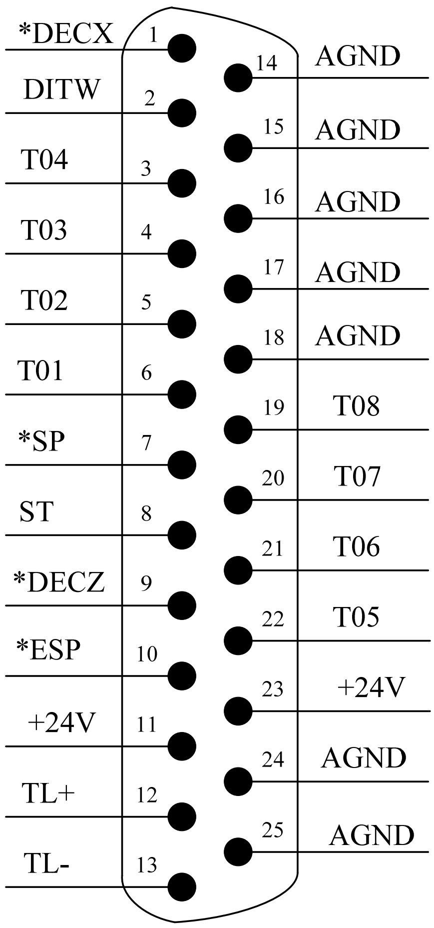
Fig. 2-7-1

CHAPTER THREE I/O INTERFACE OF MACHINE TOOL

3.1 List of the Machine Tool I/O Interface

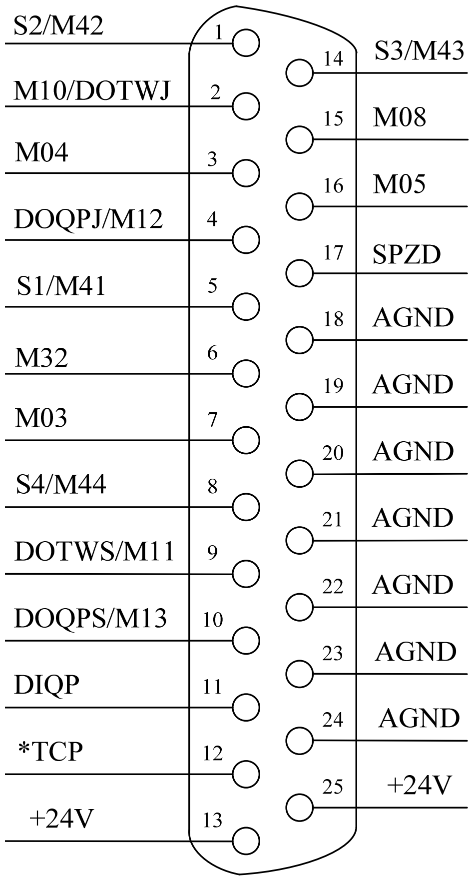
3.1.1 Standard I/O Interface of GSK980TA3

XS40 Input signal interface
(male)



Signal	Explanation
*DECX、*DECZ	X, Z axis reference return DEC. signal
DITW	Tail stock control signal
T01~T08	Tool number signal
*SP	Feed hold signal
ST	Cycle start signal
*ESP	Emergency stop signal
TL+、TL-	Tool post CCW, CW signal
+24V	+24V
AGND	0V/ common terminal

Fig. 3-1-1-1

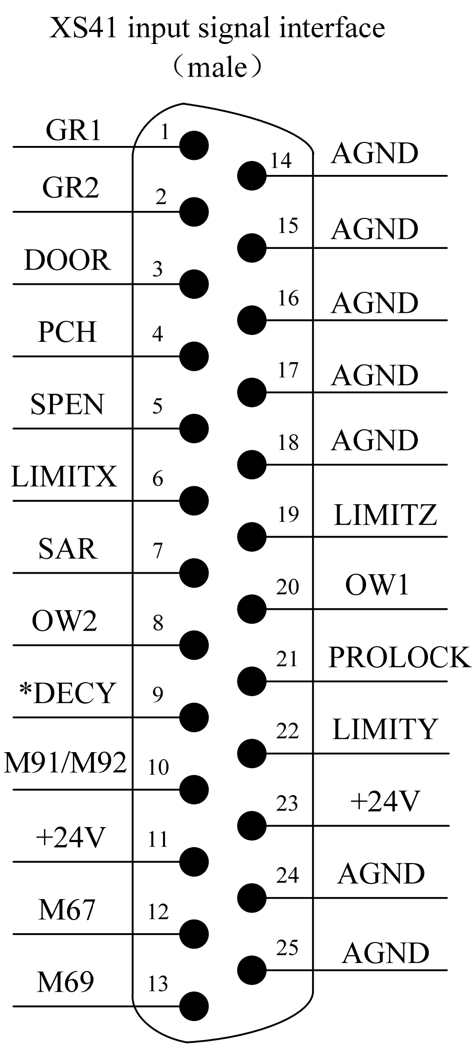


Signal	Explanation
S1~S4	Spindle gear signal
M41~M44	Spindle auto gearing signal
M03、M04、M05	Spindle CCW, CW, stop signal
M08	Coolant ON
M32	Lubricant ON
M10、M11	Tail stock forward/ retreat
M12、M13	Clamp/ release signal
SPZD	Spindle break signal
*TCP	Tool post clamp in-position signal
DIQP	Clamp control signal
+24V	+24V
AGND	0V/ common terminal

Fig. 3-1-1-2

Part Four Connection

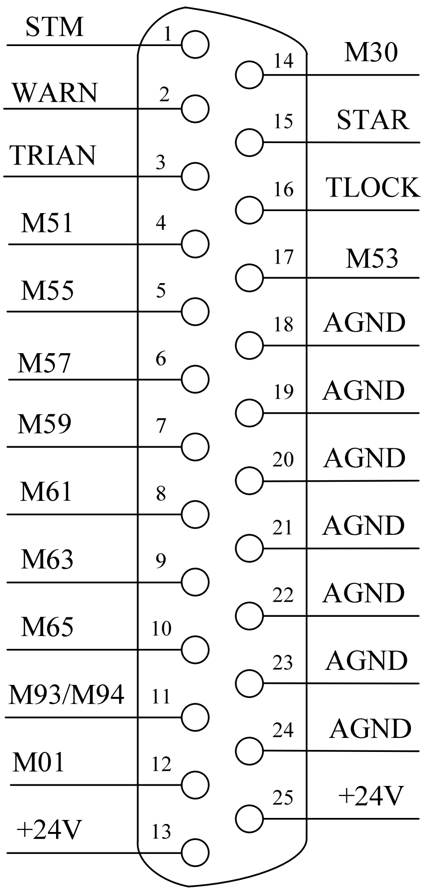
3.1.2 Expansion I/O Interface of GSK980TA3



Signal	Explanatiion
GR1、GR2	Gearing in-position signal
DOOR	Safety door detection signal
PCH	Pressure detection signal
SPEN	Feed enable signal
SAR	Spindle rotation enable signal
LIMITX、LIMITZ、LIMITY	Hardware limit signal
OW1、OW2	External alarm signal
PROLOCK	Program lock signal
*DECY	Y-axis reference return DEC. signal
M91/M92	Expansion user input signal
M67~M69	Expansion M code signal
+24V	+24V
AGND	0V/ common terminal

Fig. 3-1-2-1

XS42 output signal interface (female)



Signal	Explanation
STM	Auto running signal
M30	Output alarm signal
WARN	Machining completion signal
STAR、TRIAN	Star triangle start signal
TLOCK	Tool post clamping signal (match with LIO SHING tool post)
M51~M65	Expansion M code signal
M93/M94	Expansion user input signal
M01	Expansion user input signal
+24V	+24V
AGND	0V/ common terminal

Fig. 3-1-2-2

Note 1: Various functions can be defined to some of the input and output interfaces, and they are indicated by "/" sign in the above table.

Note 2: If the output signal is through on to COM, the output function is active. In the contrary situation, the output function is inactive.

Note 3: If the input signal is through on to +24V, the input function is active. In the contrary situation, the input function is inactive. When the input signal with "*" sign is through on to +24V, the input function is inactive. In the contrary situation, the input function is active.

Note 4: The effectiveness of +24V, COM terminals are equivalent to the CNC power box terminals that have the same names.

3.2 Input Signal

1) DC input signal A

DC input signal A is the signal from keys, limit switch, contacts of the relay (including DECX，DECZ，ESP，TCP，ST，SP，DITW，X16, etc.) at the machine side.

a) The contact point at the machine side should meet the following requirements:

Contact capacity: DC30V, 16mA above

Leakage current between contacts in open circuit: 1mA bellow (voltage 26.4V)

Voltage drop between contacts in closed circuit: 24V bellow (current 8.5mA, including cable voltage drop)

b) Signal loop of this kind of signal is shown in Fig. 3-2-1

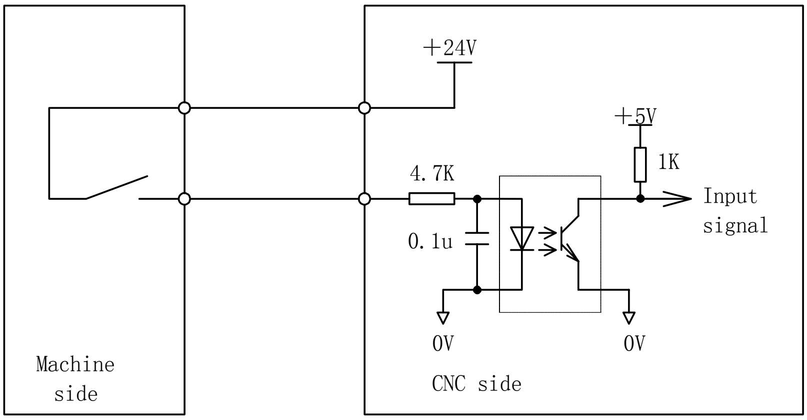


Fig. 3-2-1

2) DC input signal B

DC input signal B is the signal from the machine tool to the CNC, which is used at high speed (T1~T8)

a) The contact point at the machine side should meet the following requirements:

Capacity of the contact: DC30V, 16mA above

Leakage current between contacts in open circuit: 1mA bellow (voltage 26.4V)

Voltage drop between contacts in closed circuit: 24V bellow (current 8.5mA, including cable voltage drop)

b) Signal loop of this kind of signal is shown in Fig. 3-2-2

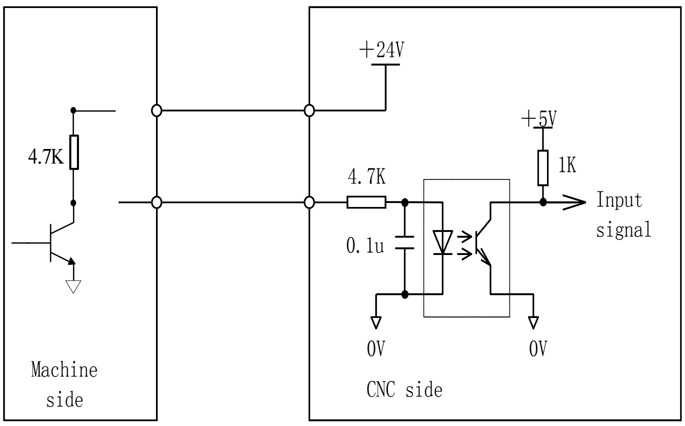


Fig. 3-2-2

Part Four Connection

3.3 Signal Expalnation

1） Input signal

a） T01～T08 tool number signal

Effective level is low level. When one of the signals is low level signal, it indicates that the tool post is at this tool number position. Connection is shown as the following figure, 8 external pull-up resistors are connected.

By setting the parameter NO:9#1, it is able to set tool post in position level signal (0: high level is effective 1: low level is effective).

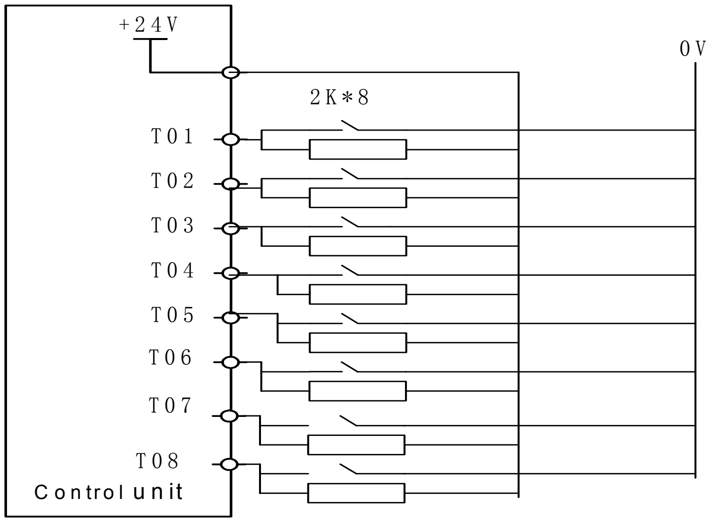


Fig. 3-3-1

b） TCP：Tool post clamping signal

In the process of tool changing, the system outputs tool post CW signal (**TL-**) when the tool post is in position and then begins to check the clamping signal. After receiving this signal, the parameter P85 setting time is delayed, and the tool post CW signal (**TL-**) is cut off, T code command is finished, and the next block is executed. After the system outputs the tool post CW signal, if the system does not receive **TCP** in the time set by the parameter **P88**, the system will issue alarm and cut off the tool CW signal. If tool post clamping signal is not output from the tool post controller, set parameter NO.9 ＃0 TCPS to 0 and the tool post clamp signal is not necessary to be connected.

Current tool number is stored at diagnosis **P49**. The system modifies this value after tool changing is finished normally. The values at the right corner are the same. After T command is specified, for some reason these two values are not the same. T displays the number of the specified tool, while P49 keeps the tool number before tool changing. When the number of specified tool is consistent with that of P49, tool changing is not performed. Therefore, when the current tool number is not identical with the one of P49, tool changing should be performed until they are the same.

In manual tool changing, T code and P49 are changed into new value after the tool changing is finished.

Tool changing sequence diagram:

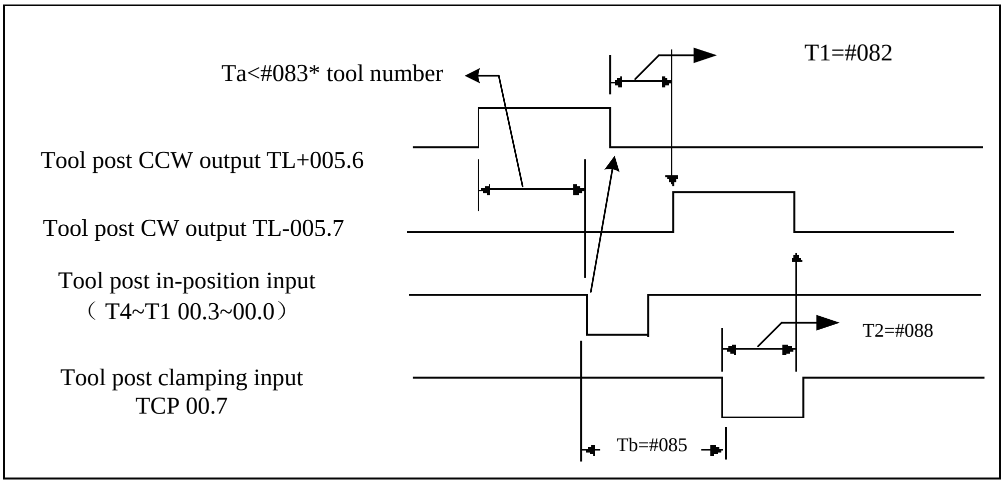


Fig. 3-3-2

#83 In the diagram is time parameter set by the corresponding parameter number.

When Ta≥#083 × currently changed tool number, alarm 185 issues: tool changing time is too long. If CCW rotation from the tool post for **Ta** (time), the tool position signal is still not received, alarm occurs.

When Tb≥#088, alarm 186 occurs: Tool post CW rotation locking signal is not received in the time of tool post CW rotation locking time.

By setting the parameter NO:9#1 TSGN, it is able to set whether high level or low level is effective for the tool post in position signal （T8～T1）.

TSGN **0 :** Tool post in-position signal is high level active（normally open）

1 : **T**ool post in-position signal is low level active（normally closed）

By setting the parameter **NO:9#0 TCPS**, it is able to set whether high level or low level is effective for the tool post clamping signal （*TCP）.

TCPS **0 :** Tool post clamping signal is low level active（normally open）

1 : Tool post clamping signal is high level active（normally closed）

c） DECX, DECZ deceleration switch signal

The signal is a normally-closed contact, which is used in mechanical reference return. The process of the reference return is as follows：

After selecting the mechanical origin mode, then press the manual feed key of the corresponding axis, and the machine tool will move to the reference point at rapid traverse speed. When reference return deceleration signal (**DECX**. **DECZ**) contact is disconnected (press down the deceleration switch), the feed rate is decelerated immediately. Then the machine tool moves at the regular low speed [set by the parameter P43. When the deceleration switch is released, the deceleration signal contact is closed again. In addition, CNC begins to detect the encoder one-turn signal or magnetic switch signal (PC signal). If this signal is high level, the motion will be halted. In addition, the corresponding zero return indicator on the operation panel lights up for machine zero return completion. Manual feed is always inactive before the zero return key is released.

Reference return direction of each axis can be set by the parameter. The connection diagram is as follows:

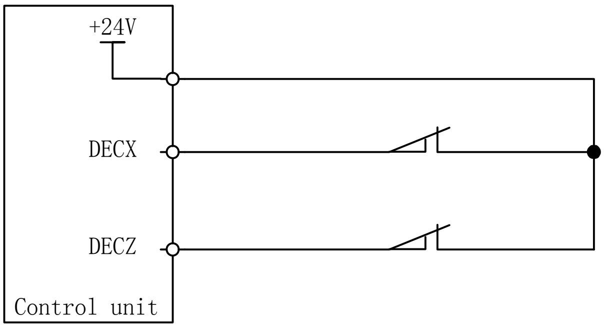


Fig. 3-3-3

d） The functions of automatic cycle start signal ST and feed pause are the same as that of the system panel. When **NO:14#0 MST=0**，**NO:14#1 M@SP=0,** signal ST and SP are active, otherwise, the signal of this point is empty. When **NO:14#1 M@SP=0,** external feed hold switch is necessary, otherwise, feed hold signal will be active all the time.

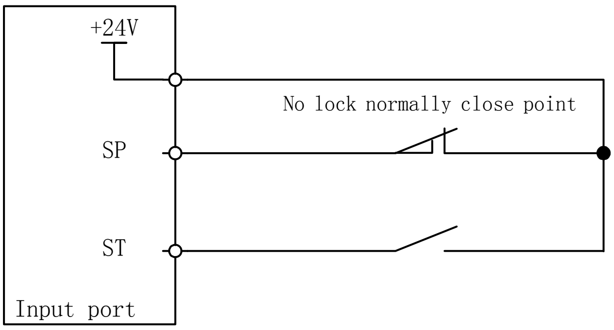


Fig. 3-3-4

e） ESP signal

This signal is a normally-closed signal. When the contact is disconnected, the control system resets, and the machine tool stops urgently. After the emergency stop, the system ready signal EN will be disconnected. At the same time, motion command output is blocked.

The system is designed with software limit function to perform overtravel check. It is unnecessary to fix a limit switch for overtravel checking. However, when the drive unit fault occurs, the motion of the machine tool exceeds the software limit zone. In order to stop the machine tool, a stroke limit switch is necessary. Its connection figure is as follows:

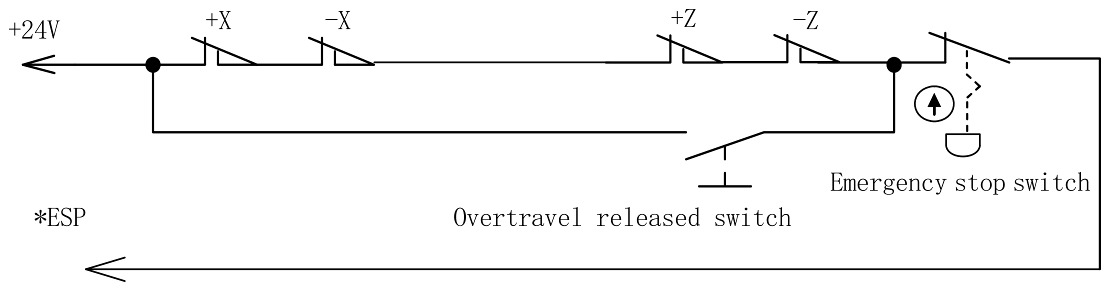


Fig. 3-3-5

When the connection in the above figure is used, if the parameter **NO:14#3 MESP=0** is set to 1, the above function is not active. The emergency signal（***ESP**）explanted here is standard connection. This connection method should be used as far as possible when users connect the signal.

f） Hardware limit signal LIMITZ, LIMITX

This signal is a normally open signal. When the contact is disconnected, the control system resets, and the machine tool stop urgently. After the emergency stop, the system hardware limit alarm occurs. At the same time, motion command output is blocked.

When this function is active, the hardware limit of each axis can be separated from **ESP**, and limitation of each axis will be reflected correctly.

When this function is used, the parameter **NO:8#2 LMTX**. **NO:8#3 LMTZ** should be set to 1. If it is set to 0, the function is not active. Moreover, by setting **NO:8#4 LTVX**. **NO:8#5 LTVZ**, it is able to set the signal is high level active or low level active. When it is active at high level, the alarm occurs when the signal is through on to +24V. When it is active at low level, the alarm occurs when the signal is through on to **0V**. When the alarm occurs, enter into the manual mode, move the overtravel axis reversely. The alarm will be released when the axis departs from the stroke switch.

g） External alarm signal **OW1, OW2**

This signal is normally open contact signal. When the contact is connected, the control system resets, making the machine tool stop urgently. After the emergency stop, the external alarm occurs in the system. At the same time, motion command output is blocked.

This signal is an optional one, which can be used as required.

When this function is used, the parameter **NO:12#6 WAR1**. **NO:12#7 WAR2** should be set to 1. If it is set to 0, the function is not active. Moreover, by setting **NO:12#3 WA2**. **NO:12#2 WA1**, it is able to set the signal is high level active or low level active. When it is active at high level, the alarm occurs when the signal is through on to +24V. When it is active at low level, the alarm occurs when the signal is through on to **0V**. When the alarm occurs, operation can be performed after reset.

h） **SPEN, SAR feed and spindle enable input signal**

These two signals are normally open signals. When SAR contact is closed, spindle can be started and the cycle start can be performed. When SPEN contact is closed, spindle can be started while the cycle start can not be performed. If these two contacts are disconnected, the spindle can not be started, and the cycle start can not be performed either.

When the spindle is rotating, if disconnect both signals, the spindle will stop immediately. After one contact is connected, the CNC starts the spindle automatically, restoring the previous speed.

If the SPEN signal is disconnected in program execution, the CNC in feed hold status. After SAR signal is closed, the current program can be performed by pressing cycle start key.

The function is active when the parameter **NO:15#1 SPEN** is set to 1. If it is set to 0, the function is not active.

i） Safety door detection signal DOOR

This signal is a normally closed signal. The system resets when the contact is disconnected, which makes the machine tool stop urgently. After the emergency stop, the alarm that the safety door is not closed occurs in the CNC. At the same time, motion command output is blocked.

This function is active when the parameter NO:16#5 SFDR is set to 1. The CNC can not run if the safety door is open, the door should be closed in operation.

When the parameter is set to 0, CNC does not check whether the safety door is closed or not. By setting parameter NO:16#6 SFDV, it is able to set the signal is high level active or low level active. When it is high level active, the safety door is closed when the signal connects to +24V. However, the alarm is issued when the signal is disconnected or it connects to 0V. When it is low level active, the safety door is closed when the signal connects to 0V. However, the alarm is issued when the signal connects to +24.

This function is active in Auto mode, safety door is not detected in other modes.

j） PCH Pressure check signal

This signal is normally closed contact signal. The system resets when the contact is disconnected, which makes the machine tool stop urgently. After the emergency stop, low pressure alarm occurs in the CNC. At the same time, motion command output is blocked.

This function is active when the parameter **NO:16#4 PSCK** is set to 1. The CNC can not be started normally if the pressure is low. By setting the parameter **NO:16#3 PSCV**, it is able to set the signal is high level active or low level active. When it is high level active, the signal is through on to +24V, which indicates the pressure reaches the specified value. If the signal is disconnected with ＋**24V** or connected to 0V, the alarm is issued. When it is low level active, the signal is through on to **0V,** which indicates the pressure reaches the specified value. The alarm is issued when the signal is through on to +24V.

When this function is active, once the low pressure alarm is detected and the signal hold time exceeds the value set by the parameter P92, the alarm is issued in CNC. At this time, the axis feed and spindle stop, automatic cycle can not be started, cooling and lubricant are cut off. The alarm can be released by resetting or powering off.

k） GR1, GR2 Gear in-position signal

This signal is normally open contact signal. See 3.5 for the functions.

l） Expansion user input signal

The system added two user defined input signals.

Code format： **M91**

M92

M93

M94

Input points correspond to the codes **M91, M92, M93, M94** are not defined. User can define it as required.

M91, M92 correspond to user defined input point 1. **M93, M94** correspond to user defined input point 2. If status of the input points conform to the one specified by the codes, the next block is executed sequentially. If not, it will wait.

M91 code detects input status of the user 1. It waits (input terminal connects with 0V) until input is inactive.

M92 code detects input status of the user 1. It waits (input terminal disconnects with 0V) until input is active.

M93 code detects input status of the user 2. It waits (input terminal connects with 0V) until input is inactive.

M94 code detects input status of the user 2. It waits (input terminal disconnects with 0V) until input is active.

3.4 Output Signal

1） Output signal

DC output signal is used for driving the relay and indicator at the machine tool side. Transistor is used in drive circuit, which includes: **S1～S4, M3, M4, M5, M8, DOTWJ（M10）, DOTWS（M11）, DOQPJ（M12）, DOQPS（M13）, M32, TL-, TL+, SPZD, M51~M70**.

a）Specifications of the transistor used for outputting

① Maximum load current when output is ON：below 200mA including transient current.

② Saturation voltage when output is ON: Max. value is 1.6V when the current is

200mA.Typical value is 1V.

③ Withstand voltage when output is OFF: below 24+20% including transient current.

④ Leakage current when output is OFF: below 100μA.

b） Output circuit

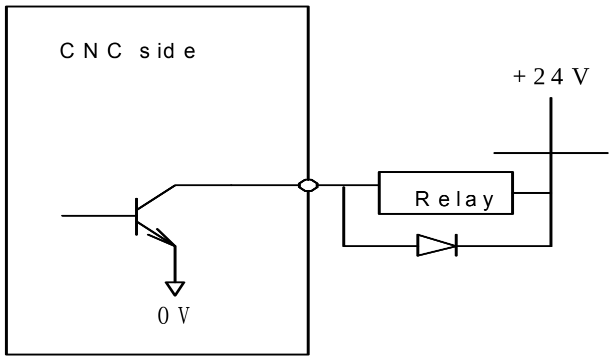


Fig. 3-4-1

All output signals for this system are provided by Darlington tube. Corresponding Darlington tube is connected when output is active. Except for pulse signals (non output hold) TL-, TL+, SPZD, others are level signals (output hold), the common port of the signal is ＋24V.

2） Explanation of output signal

a） S1～S4 Spindle speed switching value control signal

Share an output port with M41~M44. When spindle gear control is selected, S1, S2 (second gear) or S1~S4 (fourth gear) functions are active.

Only one signal in S1~S4 is active.

b） TL＋, TL－tool change signal

TL＋ is CCW tool change signal, TL－ is CW tool change signal.

c） M03，M04，M05 are spindle control M code.

M03 is for spindle CCW rotation, M04 is for spindle CW rotation, M05 is for spindle stop.

d） M08，M09 (no internal signal output) cooling control.

M08 is for cooling ON. M09 is for cooling OFF.

e）DOTWJ, DOTWS are tailstock control switches. DOTWJ: tailstock advance. DOTWS: tailstock retreat. See 3.7 for details.

f） DOQPJ, DOQPS are for chuck clamp control. DOQPJ is for chuck clamp. DOQPS is for chuck unclamp.

DOQPJ shares an output port with Y16 and DOQPS. DOQPJ is for chuck clamp. DOQPS is for chuck unclamp. The function is active when automatic chuck control is selected. It is reverse in outer chuck mode. See 3.6 for details.

g） M32，M33 (no internal signal output) lubricant control

M32 is for lubricant ON. M33 is for lubricant OFF. When **NO:13#2 JLB** is set to 0, oil is supplied continuously for the system. If it is set to 1, oil is supplied intermittently for the system. Oil supply time is set by the parameter **P107**. Intermittence time is set by the parameter **P108**.

h） SPZD Spindle brake signal SPZD

Action relationships are as follows:

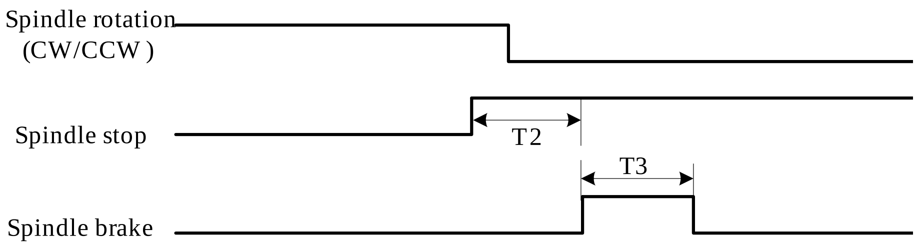


Fig. 3-4-2

T2 is delay time from outputting spindle stop signal to outputting spindle break signal. It is set on the parameter number P89.

T3 is the holding time of the spindle brake, which is set on the parameter P90.

i） STAR/TRIAN Star triangle start signal STAR/TRIAN

It is active when parameter **NO:15#7 STAR** is set to 1. **STAR** signal output is active after the spindle is started. The time set by the parameter **P93** is delayed, signal **STAR** is off and **TRIAN** signal is output.

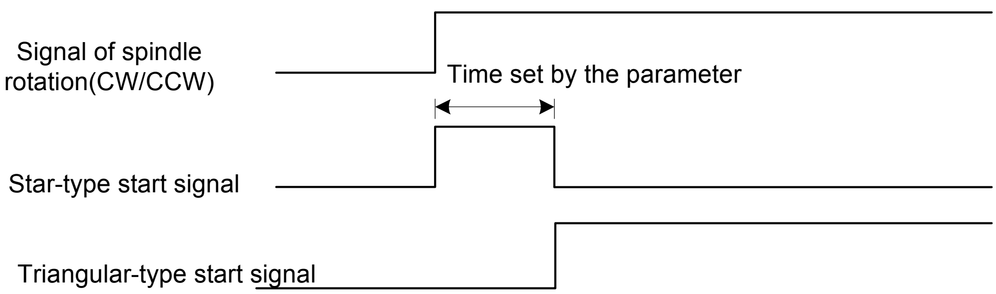


Fig. 3-4-3

j） WARN Output alarm signal WARN

It is active when the parameter **NO:15#6 OWAR** is set to 1. The signal is active if alarm is issued in the CNC. After the alarm is cancelled, this signal is inactive. The signal is used for warning customers that CNC is in alarm state.

k）Processing completion signal M30

It is active when the parameter **NO:15#5 OM30** is set to 1. After the system is powered on or M30 is executed, the system is in work completion state. At this time, the signal is active. When the CNC is processing automatically or it is running, the signal is inactive. This signal is used for warning customers that CNC is in processing completion state or in standby state.

l） Auto run signal STM

It is active when the parameter **NO:15#4 OMST** is set to 1. When the system runs automatically, the signal outputting is active. It is inactive after M30 is executed or after reset.

WARN, M30 and **OSTM** are interlocked with each other. They respectively indicate alarm, standby and processing state. These signals are mainly used for controlling of the machine tool state indicators.

m) TLOCK Tool post clamp signal

The signal is active when Taiwan LIO SHING tool post is used. Please refer to the appendix 6 for details.

n) Expansion M code

The system is added 10 user defined output ports that controlled by **M51～M70** respectively. Odd commands make the signal active, and even number of commands make the signal inactive. Long signal is output in execution. The specific definitions are as follows：

M51 code makes output of the No. 1 user output point active (output low level)
M52 code makes output of the No. 1 user output point inactive (output high level)
M53 code makes output of the No. 2 user output point active (output low level)
M54 code makes output of the No. 2 user output point inactive (output high level)
M55 code makes output of the No. 3 user output point active (output low level)
M56 code makes output of the No. 3 user output point inactive (output high level)
M57 code makes output of the No. 4 user output point active (output low level)
M58 code makes output of the No. 4 user output point inactive (output high level)
M59 code makes output of the No. 5 user output point active (output low level)
M60 code makes output of the No. 5 user output point inactive (output high level)
M61 code makes output of the No. 6 user output point active (output low level)
M62 code makes output of the No. 6 user output point inactive (output high level)
M63 code makes output of the No. 7 user output point active (output low level)
M64 code makes output of the No. 7 user output point inactive (output high level)
M65 code makes output of the No. 8 user output point active (output low level)
M66 code makes output of the No. 8 user output point inactive (output high level)
M67 code makes output of the No. 9 user output point active (output low level)
M68 code makes output of the No. 9 user output point inactive (output high level)
M69 code makes output of the No. 10 user output point active (output low level)
M70 code makes output of the No. 10 user output point inactive (output high level)

3.5 Spindle Automatic Gearing Control

When spindle frequency conversion is selected (**0~10V** analog voltage output), automatic spindle mechanical gear shifting is available for 4 gears. By setting parameter P55~P58 based on gears, making the speed of S code is consistent with the actual speed.

Relevant parameter：bit parameter **NO:01#4**：

0	0	1					MDSP				

MDSP =1: Spindle is analog volume control.

=0: Spindle is other mode.

Bit parameter P **NO:10#3. 4. 6. 7**：

0	1	0		AGER	AGIN		AGIM	SPT	TRD	G0&T	TSS

AGER =0： Spindle automatic gearing is inactive.

=1： Spindle automatic gearing is active.

AGIN =0： Not detect in-position signal when shifting to gear 1,2.

=1： Detect in-position signal when shifting to gear 1,2.

AGIM =1： Gear in-position signal is high level active.

=0： Gear in-position signal is low level active.

SPT =1： Spindle jog in auto mode is active.

=0： Spindle jog in auto mode is inactive.

Input/output signal:

GR1： Gear 1 in-position signal

GR2： Gear 2 in-position signal

Note 1: There are no gear 3, 4 in-position signals.

M41~M44：1~4 gear shifting output (Share a common port with S1~S4)

① Check whether the AGER of the parameter is 1 and analog spindle function is selected, otherwise, alarm 180 (M41~M44 can not be used when spindle is in gear control or spindle automatic gearing control is inactive) is issued.

② Check whether it is consistent with the current gear (check output state). No gearing is done if the specified gear is consistent with the current gear. If not, gear shifting will be performed based on the process ③.

③ Set the spindle speed to the one specified by the parameter 066. If there is motion, stop it temporarily.

④ After a delay (gear shifting time 1) by the parameter 063, turn off the original gear output signal and output the new gearing signal.

Part Four Connection

⑤ If the gear is 1 or 2, and the parameter 010 AGIN is 1, it jumps to ⑥, or else it jumps to ⑦.

⑥ Check the gear in-position input signal (By setting parameter 010 AGIM, it is able to set level is active at 0 or 1) of gear 1or 2, it jumps to ⑦ if the gear in-position is done.

⑦ After a delay (gear shifting time 2) by the parameter 064, output spindle analog voltage by the current gear according to a value set by parameter P055~P058 (correspond to gear1~4) and finish the gearing.

Note 2: M40 cancels M41~M44 function output. S0 cancels S1~S4 function output.

Note3: Gearing signal M41～M44 is not output when power on. Default gear is 0.

3.6 Chuck Control (Chuck Detection Function)

1．Relevant signal：bit parameter **NO:15#3**；**NO:16#0. 1. 7**：

0	**1**	**5**						**KPDW**		
0	**1**	**6**	**NWKP**						**SLSP**	**SLQP**

KPDW =1： Check chuck in-position signal.

=0： Not check chuck in-position signal.

NWKP =1： Outer chuck mode.

=0： Inner chuck mode.

SLSP =1： Check whether the chuck is clamped when the spindle is started

=0： Not check the chuck state when the spindle is started

SLQP =1： Chuck function is active

=0： Chuck function is inactive

2. Input/output signal:

NQPJ： Inner chuck clamping in-position/outer chuck unclamping in-position signal

WQPJ： Inner chuck unclamping in-position/outer chuck clamping in-position signal

DIQP： Chuck clamping/unclamping input. It is usually controlled by the foot-switch

DOQPS Chuck unclamping output

DOQPJ：Chuck clamping output

3. Action time sequence:

1）When **SLQP=1, SLSP=0 and KPDW=0**, it only controls the clamp or unclamp of the chuck, but not decides whether the chuck clamping in position or not.

2) When **SLQP=1, SLSP=1and KPDW=0**, it can not be started if the chuck is not clamped. (Alarm 044, start the spindle before the chuck is clamped)

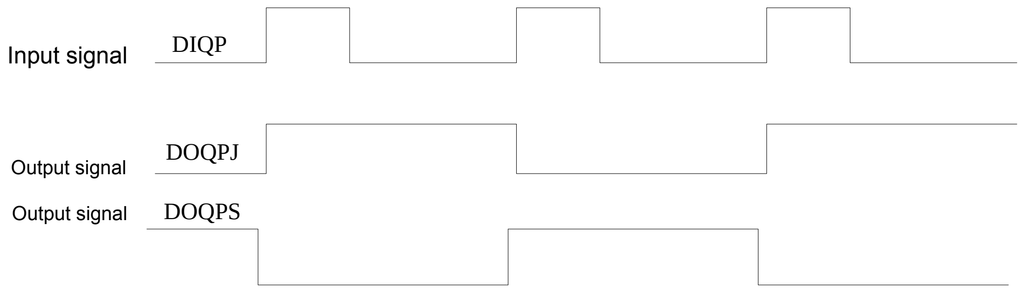


Fig. 3-6-1

DOQPS：Chuck unclamping output; DOQPJ：Chuck clamping output

DOQPJ and DOQPS output high resistance at power on. When CNC detects that the chuck input signal DIQP is active for the first time. DOQPJ（pin 4 of XS39）outputs 0V, chuck is clamped.

After M12 is executed, DOQPS（pin 10 of XS39）outputs high resistance. DOQPJ（pin 4 of XS39）outputs 0V, and chuck is clamped.

After M13 is executed, DOQPJ（pin 4 of XS39）outputs high resistance. DOQPS（pin 10 of XS39）outputs 0V, and the chuck is unclamped.

Note: CNC judges whether M12 is input. If M12 is not input, the system alarms when the spindle is started.

3）When **SLQP=1, SLSP=1, NWKP=1, KPDW=1,** CNC chooses outer chuck mode, and chuck in-position signal detecting is active:

DOQPS：chuck clamping output,　WQPJ：clamping in-position signal

DOQPJ：chuck unclamping output, NQPJ：unclamping in-position signal

DOQPJ and DOQPS output high resistance at power on. When CNC detects that the chuck input signal DIQP is active for the first time. DOQPS outputs 0V, chuck is clamped.

After M12 is executed, DOQPS outputs 0V. DOQPJ outputs high resistance, chuck is clamped and CNC waits for WQP signal to be in-position..

After M13 is executed, DOQPJ outputs 0V. DOQPS outputs high resistance, the chuck is unclamped and CNC waits for NQPJ signal to be in-position.

As the second chuck input is active, DOQPS outputs 0V, chuck is unclamped. The chuck clamping/unclamping signal is output alternatively, i.e. the output state is changed each chuck input signal is active.

Part Four　Connection

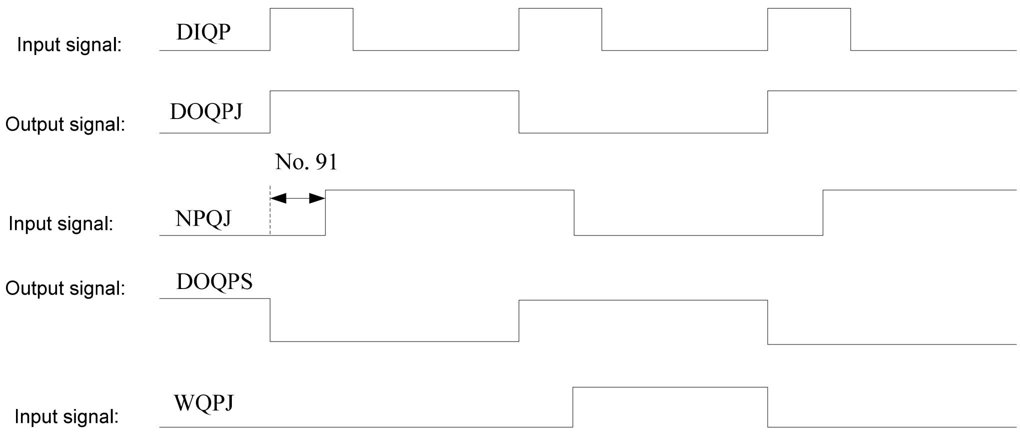


Fig. 3-6-2

Note: CNC detects clamping in-position signal altered it is clamped. If there is no clamp signal input, CNC alarms.

4）The interlock of the chuck and the spindle:

When SLQP=1, SLSP=1, M3 or M4 is active, alarm is issued if M13 is executed and the output is unchanged.

When SLQP=1, SLSP=1, KPDW=1, if M12 is executed in MDI or Auto mode, CNC does not execute next code till it detects the chuck clamping in-position signal is active. When the chuck input signal DIQP is active in Manual mode, the panel spindle CCW, CW keys are inactive till it detects the chuck clamping in-position signal is active. In spindle running or auto cycle processing, DIQP input signal is inactive. DOQPS, DOQPJ are held on at CNC reset and emergency stop.

When SLSP＝0, KPDW＝1, it does not detect whether the chuck is clamped or not. However, when chuck clamp/unclamp in-position command is executed, action is finished till in-position signal is detected. (It must be careful to use this mode)

Note 1: DOQPJ and DOQPS output 0. DOQPJ outputs 1 when chuck input signal is active for the 1st time.

Note 2: DIQP is inactive in spindle running or auto cycle processing.

3.7 Tailstock Control (Tailstock hierarchical advance/retreat control)

1. Relevant parameter：bit parameter **NO:15#3**：**NO:16#2**：

0	**1**	**5**					**WZ2**			
0	**1**	**6**						**SLTW**		

SLTW =1：Tailstock function is active

=0：Tailstock function is inactive

WZ2 ＝0：Tailstock hierarchical advancing/retracing function is inactive

＝1： Tailstock hierarchical advancing/retracing function is active

2. Input/output signal:

DITW： Tailstock clamping/unclamping input

DOTWJ： Tailstock clamping output signal

DOTWS： Tailstock unclamping output signal

DOTWJM： Tailstock slow advancing output signal

DOTWJJ： Tailstock lock output signal

WZ1J： Tailstock slow advancing detection input signal

WZ2J： Tailstock in-position detection input signal

3. Action time sequence:

1）When SLTW=1, WZ2＝0,

DITW：Tailstock clamping/unclamping input

M10/M11 is tailstock clamping/unclamping control signal respectively.

After M10 is executed, DOTWS outputs high resistance, DOTWJ outputs 0V, and tailstock advances.

After M11 is executed, DOTWJ outputs high resistance, DOTWS outputs 0V, and tailstock retreats.

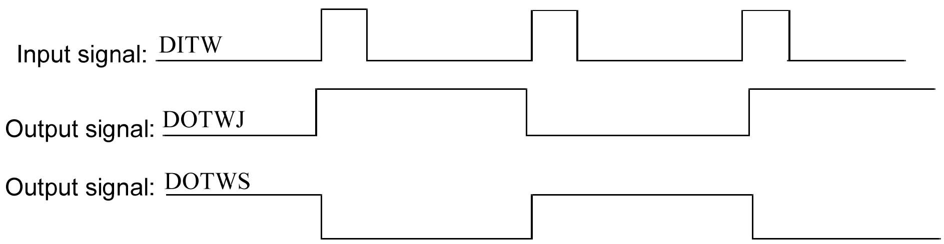


Fig. 3-7-1

2）When SLTW=1, WZ2＝1, S3/S4/M43/M44 function is inactive. If the command is executed, CNC will issue alarm: illegal command. Maximum tools number is 4.

After M10 is executed, CNC DOTWS outputs high resistance, DOTWJ outputs 0V, and tailstock advances rapidly. When WZ1J signal is active, DOTWJ outputs high resistance, DOTWJM outputs 0V, tailstock advances slowly. When WZ2J signal is active, DOTWJM outputs high resistance, DOTWJJ outputs 0V, tailstock is locked and tailstock advancing is finished.

After M11 is executed, DOTWJJ outputs high resistance, tailstock is unclamped. DOTWS outputs 0V, tailstock retreats rapidly, and the tailstock retreat is finished.

Part Four Connection

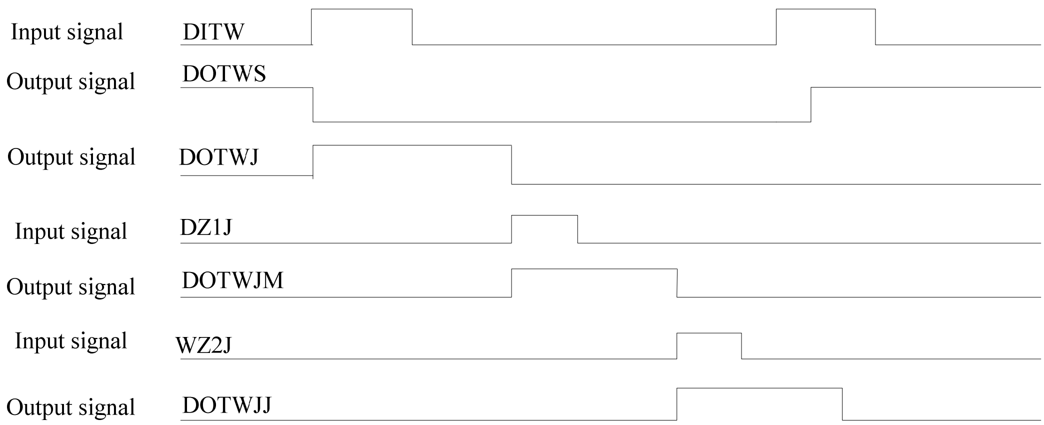


Fig. 3-7-2

Note 1: DOTWJ and DOTWS output 0 when power on. DOTWJ outputs 1 when chuck input signal is active for the 1st time.

Note 2: DITW input is inactive in spindle running or auto cycle processing.

3.8 External Program Lock

External program lock is an external key switch, which is used to prevent work programs from altering.

Relevant：bit parameter **NO:13#6. #7**：

0	**1**	**3**	**PLCKV**	**PLCK**						

PLCKV =1：Program lock is high level active

=0：Program lock is low level active

PLCK =1：Program lock function is active

=0： Program lock function is inactive

Input/output signal:

PROLOCK：External program lock

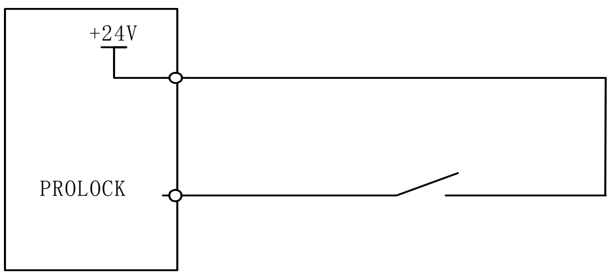


Fig. 3-9-1

3.9 External Feed Pause and Spindle Pause

External feed pause and spindle pause is a three-phase code-chosen-switch. They are used to stop program in auto running or stop the spindle. When the three-phase is on the normal position, the modes “press reset key again” or “execute program directly” can be selected.

Relevant parameter：bit parameter **NO:15#0. #1**：

0	**1**	**5**							**SPEN**	**SPENC**

SPEN =1：Feed and spindle enable input are active

=0：Feed and spindle enable input are inactive

SPENC =1： Feed and spindle control

=0： Spindle control

Note: When SPENC and SPEN are set to 0, spindle enable function is inactive. When SPENC is set to 0, SPEN is set to 1, spindle enable function is active. When SPEN signal of the input port is high level, the spindle can be started by the CNC. When SPENC and SPEN are set to 1, SAR of the input port is high level input, the spindle can be started by the CNC and feed is enabled. When SPEN signal is high level, spindle is enabled, while feed is forbidden. When SAR and SPEN are not connected to +24V, spindle rotation and feed are forbidden.

Input/ output signal:

SAR： External feed hold

SPEN： External spindle hold

This function has three connection states:

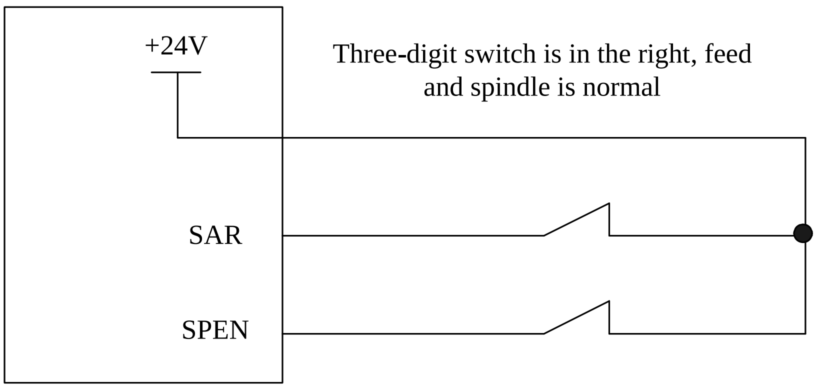


Fig.3-9-1

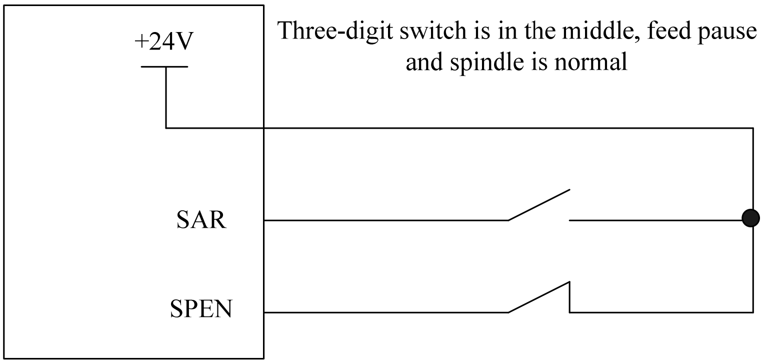


Fig.3-9-2

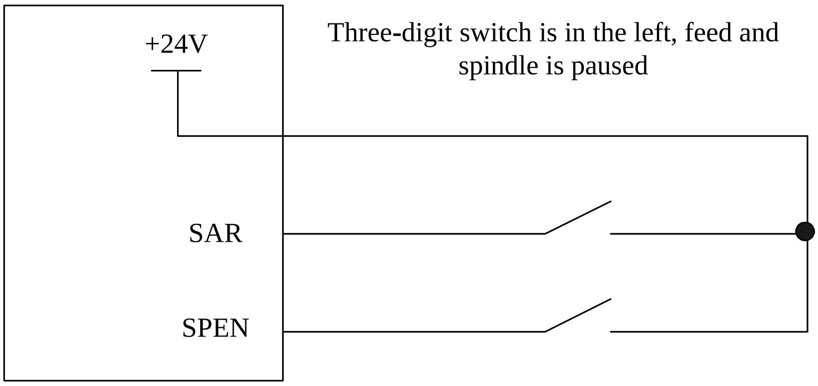


Fig.3-9-3

3.10 I/O Signal Disgnosis List

All I/O signals of the system are described in this diagnosis list.

1） List of the signals from machine tool to CNC

No.	7	6	5	4	3	2	1	0
001	X17	X16	X15	X14	X13	X12	X11	X10
	TCP	DIQP	*DECX	DITW	*SP	ST	DECZ	*ESP
Socket pin No. →	XS39: 12	XS39: 11	XS40: 1	XS40: 2	XS40: 7	XS40: 8	XS40: 9	XS40: 10
002	T8	T7	T6	T5	T4	T3	T2	T1
	XS40: 19	XS40: 20	XS40: 21	XS40: 22	XS40: 3	XS40: 4	XS40: 5	XS40: 6
003	SAR	LTZ	LTX	SPEN	PSCK	DOOR	GR2	GR1
004						LCK	OWA2	OWA1

Part Four Connection

2) List of the signals from CNC to machine tool

No.:	7	6	5	4	3	2	1	0
005	Y17	Y16	Y15	Y14	Y13	Y12	Y11	Y10
Diagnosis No.	SPZD	M12	M05	M32	M08	M10	M04	M03
Socket pin No.	XS39:17	XS39: 4	XS39:16	XS39: 6	XS39:15	XS39: 2	XS39:3	XS39: 7
006	X27	X26	X25	X24	X23	X22	X21	X20
	TL-	TL+	M13	M11	S04/M44	S03/M43	S02/M42	S01/M41
Socket pin No.	XS40:13	XS40:12	XS39: 10	XS39: 9	XS39: 8	XS39:14	XS39: 1	XS39: 5

3.11 Function Description

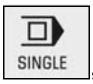
In diagnosis page, 【machine panel】 interface is shown as figure 3-12-1:

```
OPERATOR                    O0001 N0000   | Message            10:13
                                          | Axis Rev            0000
  MLK (KEY1) :    *OFF   ON               | Tool No             0100
  AFL (KEY2) :    *OFF   ON               | Fact Rate              0
  SBK (KEY3) :    *OFF   ON               | Cool Mes             OFF
  DRN (KEY4) :    *OFF   ON               | Axis Mes            Stop
  BDT (KEY5) :    *OFF   ON               | G00 G98 G97 G40
                                          |
  POSITION (RELATIVE)                     |
     U       0.000                        |
     W       0.000                        |
                                          | MDI
```

Fig. 3-12-1

On machine panel page, 1, 2, 3, 4, 5 keys are used to shift machine lock, M.S.T. lock key, single block, dry run, block skip switch. Their functions are the same as that of the keys 

, 

, 

, 

, 

.

CHAPTER FOUR　MACHINE TOOL DEBUGGING

The debugging and trial run methods and steps are described in this chapter. The corresponding operations can be performed after debugging by the following steps.

4.1　Debugging Preparation

Debug the system according to the following steps:

- CNC connection: correct connection is the foundation on which debugging is performed smoothly;
- Drive unit parameter setting: set motor type parameter, control mode setting;
- CNC parameter setting: control parameter, speed parameter setting.

4.2　System Power On

- Switch on air break switch in the electric cabinet.
- Connect to the air break controlling DC 24V or fuse, and check whether DC24V power is normal.
- Check whether the other power is normal.
- Turn on the CNC system.

4.3　Emergency Stop and Limit

This system has software limit function. It is suggested that hardware limit is employed by fixing the stroke limit switches at the positive or negative directions of the axis. The connection is shown as follows:

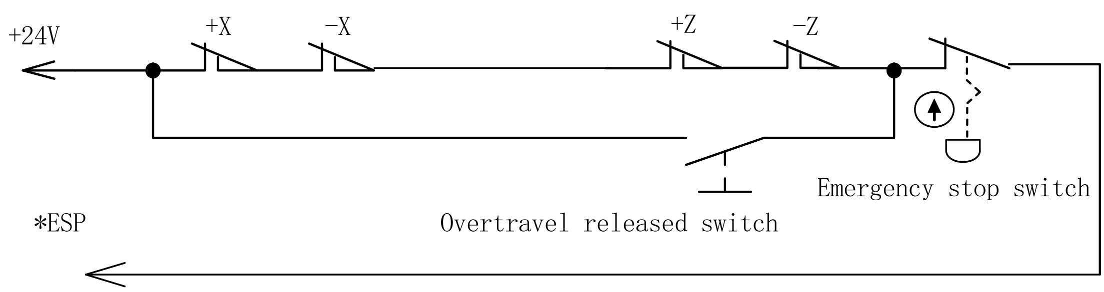


Fig. 4-3-1

ESP signal can be shielded by parameter **NO:14#3 MESP**

When **MESP=1**, emergency stop is inactive. It is for debugging.

When **MESP=0**, emergency stop is active. It can be used normally.

Diagnosis **NO:01#0** can be used to check whether the emergency signal is input or not. If it is set to 1, which indicates that the emergency stop switch is connected (switching signal is normally closed signal).

In Auto mode or MPG mode, slowly move the axes to testify the activity of the stroke limit switch, correctness of the alarm display, activity of the overtravel release button. When overtravel occurs or emergency stop button is pressed, alarm 254 “the system is not ready” will be issued by CNC system. Press the overtravel release button on the panel and move the axis reversely until it detaches the stroke switch, the alarm will be eliminated.

4.4 Drive Axis Debugging

1）Drive unit alarm

If X-axis or Z-axis drive unit alarm displays when power on, user must first check whether the drive unit alarm indicator is lighted up or not (for stepping drive unit) or check whether the alarm is displayed (for servo drive unit). If alarm occurs in drive unit, process the drive unit first, and then check whether the drive alarm line is cut off or not. If none above phenomenon occurs, the reason is that the level set by alarm parameter does not match the drive unit. Therefore, alter **#0** and **#1** the parameter **NO:8** (**ALMX** and **ALM**Z correspond to X and Z axes respectively). After the parameter alteration, press 

key to eliminate drive unit alarm.

2）Direction adjustment of the driving motor

If operation direction of the drive unit is inconsistent with coordinate direction of the computer, alter **#0** and **#1** the parameter **NO:7#0** (**DIRX** and **DIRZ** correspond to X and Z axes respectively) or change direction selection switch of the transformer drive unit.

4.5 Gear Ratio Adjustment

1）Formula

CMR：Command multiplier coefficient ［Parameter **P17**～**18**］

CMD：Command frequency division coefficient［Parameter **P19**～**20**］

a：Stepping angle (degree)

L： Movement amount corresponds to the step motor rotating one turn（mm）

Example: Z axial motor is connected directly. Screw pitch of the lead screw is 6mm. X axial reduction ratio is 3/5. Screw pitch of the lead screw is 4mm. Its servo gear ratios are as follows: calculating method for X axis: 4×2500×3/（4×1000×5）＝3/2 that is, command multiplier coefficient of the X axis=3, command frequency division coefficient of the X axis =2. Calculating method for Z axis: 4×2500/（6×1000）＝5/3, that is, command multiplier coefficient of the Z axis=5, command frequency division coefficient of the Z axis =3. This gear ratio can be set to the drive unit. When stepping drive unit is used, its stepping angle is as follows:

X axis: 360×5/（4000×3）＝0.15, Z axis: 360/6000=0.06.

4.6　Rapid Traverse Rate and Linear ACC&DEC Adjustment

Parameter number **P21**, **P22** respectively set the rapid traverse rate scope （**30 mm/min～30000mm/min**）of the X and Z axes.

Parameter number **P23**, **P24** respectively set the linear ACC&DEC time constant scope （**8～4000**）of the X and Z axes in rapid traverse.

Parameter number **P25**, **P26** respectively set the S-shape time constant value setting scope （**8～4000**）of the X and Z axes in rapid traverse.

Parameter number **P28** is lower value of the ACC&DEC low speed in cutting feed （**0～500mm/min**）.

Parameter number **P29, P30** are S acceleration/deceleration time constant in cutting feed, which scope is **8～4000**.

Parameter number **P31** are linear acceleration/deceleration time constant in cutting feed, which scope is **8～4000**.

Note: When stepping drive unit is used, it is suggested that ACC&DEC time constant are bigger than or equal to 300. When servo drive unit is used, the value can be set to 100.

4.7　Upper Speed of the Cutting Feed

Parameter P27 is used to set the upper speed of the cutting feed. It must be set to 8000.

4.8　Adjustment for Thread Machining

1）Adjustment for thread run-out width

Parameter P68 **THDCH** is used to set thread run-out length. Chamfering width **=THDCH*1/10*** screw pitch.

2) Threading exponential acceleration and deceleration

Parameter **P69** is linear ACC&DEC constant of two axes in threading. It is usually set to 100.

Parameter **P157** is ACC&DEC time constant of the shorter axis in thread run-out.

Parameter **P158** is maximum speed of the shorter axis in thread run-out.

3) Spindle setting

Parameter P59 is for threading is performed after percentage is stable.

Parameter P59 is spindle speed sampling period (×4ms).

4.9 Machine (mechanical) Zero Adjustment

Users have to make sure that zero-return deceleration switch is connected correctly. It can be checked by diagnosis NO:1#5 DECX and NO:1#1 DECZ. Make sure that zero point signal is linked correctly. It can be checked by diagnosis **NO:9#0 PCX** and **NO:9#1 PCZ**. If it is right, this digit will be changed into 1 after the motor rotates once.

Parameter **NO:6#4. 5**（**ZCX** and **ZCZ** correspond to X and Z axes respectively）are used to set the type of the zero return. They are usually set to 0. For the detailed explanations, please refer to *machine zero return*, chapter 8.

Parameter **NO:5#1. 2**（**ZMX** and **ZMZ** correspond to X and Z axes respectively）are used to set direction of zero return. They are usually set to 1, Which indicates CCW zero return.

4.10 Offset between Lead Screws

Parameters P36, P37 are offset between lead screws of the X, Z axes. Thereinto, No.36 is diameter value.

Backlash compensation should be detected by dialgaug, clock gauge or laser detector, it does't recommend to be detected by handwheel or single-step mode, because backlash compensation only compensate accuratly can improve the precision of processing, it is suggest to use the following method:

- Edit program:

```
O0001；
N10 G01 W10 F800；
N20 W15 ；
N30 W1 ；
N40 W-1 ；
N50 M30 。
```

- Backlash compensation value (No.36,37) should be set to 0 before measuring;
- Single segment to run the program, find measure basis A after twice positioning, record current data. Run 1mm in the same direcition, then run 1mm to point B reverse to read current data.

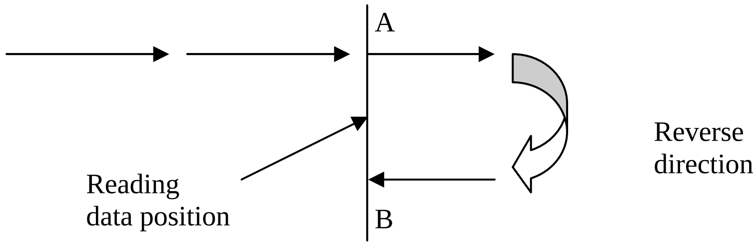


Fig.4-4 Backlash **measure method drawing**

- Backlash error compensation value= | data record from point A –data record from point A | ;input the count result to system parameter No.036(BKLX) or NO.037 (BKLZ).

Data A ：data of dial indicator read at point A;

Data B ：data of dial indicator read at point B;

pulse equivalent: 1 mu

Note 1:system parameter No.033 can set frequency of backlash compensation.

Note 2:backlash compensation should be rechecked after the machine tool has been used for 3 months.

4.11　Electronic Toolpost Debugging

Make sure the cable of tool number input signal is connected correctly. It can be checked by **NO:2#1～7**（**T01～T08**）. When it is right, in-position signal displays 0. Otherwise, in-position signal displays 1. (In-position signal displays 1, and it displays 0 for other situations)

Diagnosis **NO:6#6. 7**（**TL-. TL+**）can be used to check whether toolpost CCW/CW rotation signals are correct in tool changing.

The following parameters should be set correctly to ensure the toolpost runs normally.

Parameter **NO:9=00001110** （**BIT0=1** indicates toolpost clamping is high level active, **BIT0=0** indicates toolpost clamping is low level active**. BIT1=1** indicates toolpost clamping is low level active，**BIT1=0** indicates toolpost clamping is high level active）

Parameter P81=4: Total tool number selection

Parameter **P82=10**: Tool changing time T1 (**x4ms**)

Parameter **P83=1000**: Upper time for moving a tool in tool changing

Parameter **P84=15000**: Upper time from changing the 1^{st} tool to changing the last tool

Parameter **P85=200** : Tool post CW clamping time (x4ms)

Parameter **P88=500**: Alarm time of not receiving *TCP signal

4.12 Spindle Adjustment

When converter controls the spindle, parameter **NO:1#4=1**. When the spindle is not controlled by converter, parameter **NO:1#4=0.**

The system could be equipped with 1024 or 1200-line encoder (outputting **A**, $\overline{A}$, **B**, $\overline{B}$, **Z**, $\overline{Z}$ signal), which can be set by parameter P52 . When drive ratio between the spindle and encoder is not 1: 1, it can be adjusted by setting the gear number between spindle and encoder with parameter P53, P54 .

Diagnosis **NO:9#2 PCS** could be used to check screw head signal of the spindle encoder.

When the converter controls the spindle, adjust value of parameter P55 to make input rotational speed command (S command) is consistent with the actual speed.

Parameter P89 is the time from stopping spindle command to outputting spindle brake. P90 is spindle brake time. When **P89=50**, **P90=60**, 240ms spindle brake is performed after M05 sending 200ms.

4.13 Step/MPG Adjustment

When **NO:1#3=1**, MPG mode is active, Step mode is inactive. When **NO:1#3=0**, MPG mode is inactive, Step mode is active.

4.14 Three-state Switch Adjustment

In Auto mode, turn the knob to the right side: feed, spindle are normal; to the middle: feed stops; to the left: spindle and feed stop.

1) Three-state switch parameter setting

   When parameters **NO:15#0 SPENC and #1 SPEN are** set to 1, three-state switch is active. When they are set to 0 at the same time, three-state switch is shielded.

2) Three-state switch signal diagnosis

   Diagnosis **NO:003#7** can be used to check whether the feed pause signal is input. It indicates that three-state switch is turned to the middle if it is 0, “stop” alarm will be issued by CNC.

   Diagnosis **NO:003#4** can be used to check whether the spindle pause signal is input. It indicates that three-state switch is turned to the left if it is 0, spindle stops and “stop” alarm will be issued by CNC.

3) Notice:

   a) In operation, turn the knob to the normal gear (right side), subsequent programs can be executed after “cycle start” key is pressed again.

b) If the spindle has been started (e.g. CCW rotation), turn the knob to the left gear, the spindle will stop. Then turn the knob to the previous gear, spindle rotation state will be restored (revert to CCW rotation).

c) Thread cutting, machine zero return, program zero return, manual, MPG are not controlled by three-state switch.

4.15　Other Adjustment

Parameter No.				Diagnosis position							
0	1	0		AGER	AGIN		AGIM	SPT			
0	1	4		KEY1	SKEY			MESP	MOT	M@SP	MST
0	1	5								SPEN	
0	1	6							SLTW	SLSP	SLQP

AGER=1/0：Spindle automatic gearing active/inactive

AGIN：When it is 1, detect gearing in-position signal when gearing to gear 1, 2; When it is 0, gearing in-position signal is not detected

AGIM=0/1：**M411/*SPEN. M412/*STEN** signals active level selection (0/1 active level)

SPT=0/1：Spindle jog function is inactive/active［Spindle jog dwell time can be set by parameter **P94**］

SLTW=0/1：Tailstock function is inactive /active

SLQP：When it is 1, chuck control function is active; When it is 0, chuck control function is inactive

SLSP：When it is 1, check chuck clamping, if the chuck is not clamped, spindle is disabled (alarm 0015, movement stops).

When it is 0, chuck clamping is not checked.

MST=1：External cycle start signal（**ST**）is shielded, it is not cycle start switch now.

M@SP=1：External pause signal（**SP**）is shielded, it is not pause switch now.

M@SP=0: External pause signal is active, now external pause switch should be installed at this time, otherwise, computer displays “pause” alarm.

MOT=1：Not check software stroke limit.

MESP=1：Emergency stop signal （**ESP**）is shielded, emergency input is inactive.

SKEY=1： Program switch is shielded, edit program is forbidden.

KEY1=1： Program switch is on when power on.

CHAPTER FIVE   MEMORIZING SCREW-PITCH ERROR COMPENSATION

Only the parameter NO:2#2 is set to 1, the memorizing screw-pitch error compensation is active.

Use methods of the memorizing screw-pitch error compensation are described as follows:

1. Function

The minimum movement unit could be used for axes screw-pitch.

2. Specification

Take the reference point as the compensation origin, and set the compensation value into the parameters according to axes compensation intervals.

a) Compensated axis: X, Z axis

b) Compensation points

360 points for each axis (X, Z axis)

c) Compensation range

0～±7×compensation override (minimum movement unit) for each compensation point.

Compensation override ×1. ×2. ×4. ×8. ×16 (used for whole axis)

d) Compensation intervals

Movement unit	**Min. interval**	**Max. interval**	**Unit**
Metric	1000	9999999	0.001mm
Inch	1000	9999999	0.0001 inch

(Max. compensation range = interval ×360)

Actual compensation interval: set an appropriate value in the range above according to the maximum compensation range and mechanical travel. For linear axis, if the interval set is smaller than the min. interval, compensation is frequently unable to perform at the expected position. At this time, reduce the feedrate.

3. Parameter setting

Set the screw-pitch error parameter as methods described bellow in MDI mode or emergency stop state.

a) Screw-pitch error compensation override

0	3	8		1

Output the product of specified compensation value and the override (used for whole axis)

Part Four   Connection

b) Screw-pitch error origin

0	3	9	X-axis screw-pitch error origin

0	4	0	X-axis screw-pitch error origin

X, Z screw-pitch error origin: This is the origin on the screw-pitch error setting list.

The value from 0 to 360 in each axis can be set according to mechanical requirements.

c) Compensation interval

0	4	1	X-axis screw-pitch error compensation interval

0	4	2	Z-axis screw-pitch error compensation interval

X, Z: screw-pitch error compensation interval.

This is setting of screw-pitch error compensation interval. The value set should be bigger than 10. Moreover, if it is set to 0, compensation will not be performed.

d) Setting of compensation value

The axes screw-pitch compensation values are set according to the parameter numbers in the following table.

In screw-pitch compensation interface, input X/Z compensation from 000～255. The compensation value can be set at the range **0～±7**. If exceeding this range, the input value is in active. For detailed operations, please refer to *parameter display, alteration and setting, section 3.6, operation*.

Example：

1	2	0	-7

In the above example, -7 indicates that it is the compensation value set at point 120 of X axis.

4. Parameter setting examples

a) Screw-pitch error origin=0, compensation interval=10.000

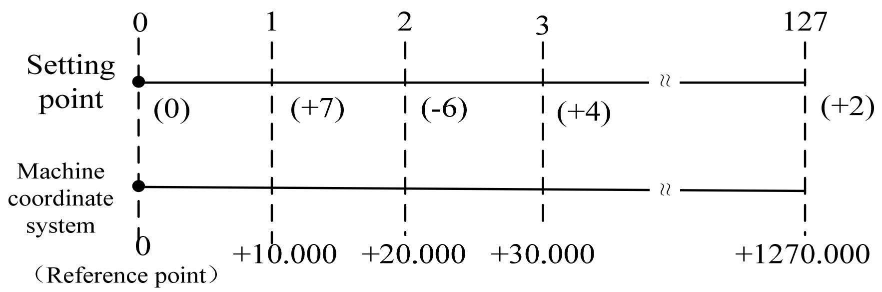


Fig. 5-1

The head of compensation point corresponds to the reference point, and compensation point1 corresponds to a point of positive 10000 moving from this reference point, and a following compensation point from this point every 10000 interval. The 127th compensation point is the compensation value at position 1270000. Therefore, at the compensation point 1, set compensation value moving from 0 to 10000, at the compensation point 2, set compensation value moving from 10000 to 20000. At compensation point N, set compensation value when moving from (N-1)×(compensation interval) to N×(compensation interval).

Above is the example for the following compensation interval error

Interval	0	～	10000	+7
	10000	～	20000	-6
	20000	～	30000	+4

Actually the machine moves from the reference point to the point of +30000, the screw-pitch error compensation is：

(+7)+(-6)+(+4)=+5

Machine moves from the reference point to the point of +30000，the screw-pitch error compensation is：

-(+4) - (-6) - (+7) = -5

b) Screw-pitch error origin=60，Compensation interval=10000

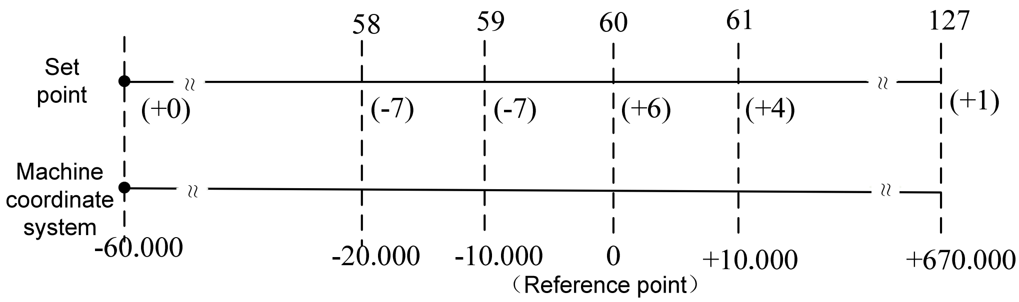


Fig. 5-2

The position No. 61, No.60 in the compensation table corresponds to the reference point, compensation point 61 corresponds to a point of positive 10000 moving from orgin, and a following compensation point from this point every 10000 interval. The 127th compensation point is the compensation value at position +670000. While the position 59 corresponds to a point of negative 10000 moving from origin. Also there is a compensation point from this point every 10000 interval. The compensation point 0 corresponds to the compensation at position -600000. Therefore, the point N is set by a compensation of moving from (N-61)×(compensation interval) to (N-60)×(compensation interval).

Above is the example for the following compensation interval error

Interval	-20000	～	-10000	-7
	-10000	～	0	-7
	0	～	+10000	+4

Actually the machine moves from -20000 to the point of +10000, the screw-pitch error compensation is：

(-7)+(-7) +(+4)=(-10)

The machine moves from+10000 to the point of-20000, the screw-pitch error compensation is：

-(+4)-(-7)-(-7) =(+10)

c) Screw-pitch error origin=127，Compensation interval=10000

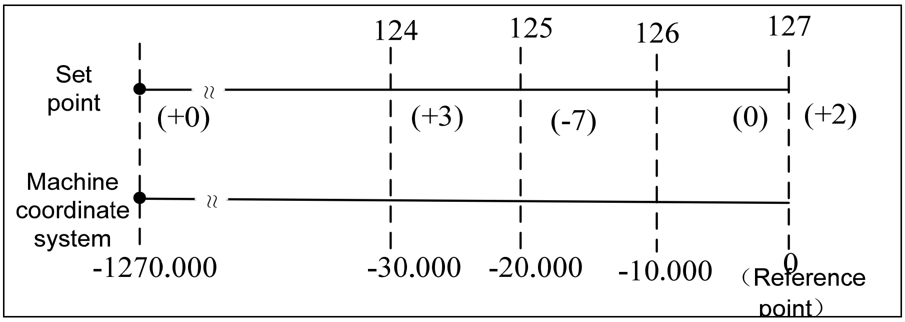


Fig. 5-3

The end of compensation table corresponds to the reference point, and compensation point126 corresponds to a point of positive 10000 moving from this reference point, and a following compensation point from this point every -10000 interval. The compensation point 1 is the compensation value at position -1260000. Therefore, at the compensation point 127, set compensation value moving from 0 to -10000, at the compensation point 26, set compensation value moving from -10000 to -20000. At compensation point N, set compensation value when moving from (N-128)×(compensation interval) to (N-127)×(compensation interval).

Above is the example for the following compensation interval error

Interval	-30000	～	-20000	+3
	-20000	～	-10000	-7
	-10000	～	-00000	0

Actually the machine moves from -30000 to the reference point, the screw-pitch error compensation is：

(+3)+(-7) +(+2)=(-2)

5. Methods for compensation value setting

As shown in the above section, the compensation values are set according to the following factors.

1. Position relations between the reference point and compensation point
2. Machine moving direction
3. Compensation interval

The compensation value of the compensation point N(N=0. 1. 2. 3. …255) is decided by the mechanical error (the differences correspond to the movement command value and movement) at the interval N, N-1.

★ Methods for inputting compensation value

Using MDI&LCD or I/O interface, and the methods of inputting common parameters, compensation value can be input.

★　Output of the compensation value

Using the same methods of outputting common parameters, the axes compensation value can be output.

★　Compensation value backup

Methods for screw-pitch compensation value backup (including input and output) are identical with that of the parameter value backup.

★Cautions for compensation setting

a)　Compensation interval setting (Parameter 41, 42)

When compensation interval is positive, use this value for compensation.

When compensation interval is 0, the compensation is not performed.

b)　After setting the screw-pitch error parameter, return to the reference point.

c)　Screw-pitch error compensation value (compensation parameter 000～255)

APPENDIX

APPENDIX I PARAMETER LIST

1.1 Explanations for parameter

Parameters can be divided into the following ones according to the data types:

Data type	Active data range	Remarks
Bit parameter	0 or 1	CNC provides default value. User can alter the setting as required.
Data parameter	Depend on the range of the parameter	CNC provides default range and default value, user can alter the setting as required.

1. For bit type and axis type parameters, each of figures consists of 8 bits. Each bit with different significance.

2. In the above table, the value of each kind of data is common active range. The specific parameter value range is not the same.

[Example]

（1）Significance of the bit parameter

Data No. **BIT7 BIT6 BIT5 BIT4 BIT3 BIT2 BIT1 BIT0**

（2）Significance of the data parameter

0	**2**	**1**	

Data No. Data

Note: The blank parameter numbers in the parameter explanation and the ones displayed on the screen but not written on the parameter list are prepared for future expansion. They should be set to 0.

1.2. Bit parameter

Parameter No.

0	**0**	**1**	**LAN**	**SCW**		**MDSP**	**WHLA**	**RAD**		

LAN =1: English（display in English）

=0: Chinese (display in Chinese)

SCW =1: Inch system for min. command increment

=0: Metric system for min. command increment

MDSP =1: Spindle is analog volume control

=0: Other mode

WHLA =1: MPG mode

=0: Step mode

RAD =1: X-axis radius programming

=0: X axis diameter programming

Standard setting：0 0 0 0 0 0 0 0

Parameter No.

0	**0**	**2**	**AUN**		**ORC**		**CCMP**	**SCRP**	**RDRN**	**PTSR**

AUN = 1： Add sequence number automatically in editing

= 0： Not add sequence number automatically in editing

ORC = 1： Tool offset by radius

= 0： Tool offset by diameter

CCMP = 1： Have tool compensation C function

= 0：Do not have compensation C function

SCRP = 1： Have screw-pitch compensation function

= 0： Do not have screw-pitch compensation function

RDRN =1： Rapid traverse command is active in Dry run

=0： Rapid traverse command is inactive in Dry run

PTSR =1： Tool compensation by coordinate offset

=0： Tool compensation by move

Standard setting：0 0 0 0 0 0 1 1

Parameter No.

0	**0**	**3**		**MNT**	**DECZ**	**DECX**	**DRDY**	**PCMD**	**HWC**	SMT

MNT =1： Machining time and parts number are not reserved when power off

=0： Machining time and parts number are reserved when power off

DECZ =1： Deceleration signal set to 1 indicates deceleration when Z axis returning to the reference point

=0： Deceleration signal set to 0 indicates deceleration when Z axis returning to the reference point

DECX =1： Deceleration signal set to 1 indicates deceleration when X axis returning to the reference point

=0: Deceleration signal set to 0 indicates deceleration when X axis returning to the reference point

DRDY =1: Drive unit/servo are ready when the signal is 1

=0: Drive unit/servo are ready when the signal is 0

PCMD =1: CNC output wave is pulse

=0: CNC output wave is square

HWC =1: MPG controls dry run interpolation

=0: Common dry run

SMT =1: lap transition precise and to the point

=0: lap transition smooth

Note:: When HWC parameter is active, after dry run is started, machine movement speed depends on the rotation speed of the MPG.

Standard setting: 0 0 0 0 0 1 0 0

Parameter No.

0	0	4	**OFT**	**SMA**	**M30**	**PPD**	**G50RP**	**CM98**	**HDZ**	**HDX**

OFT =1: offset power off do not memory

=0: offset power off memory

SMA =1: Spindle maual gearing for S code

=0: Spindle automatic gearing for S code

M30 =1: Cursor to beginning when M30 executing

=0: Cursor to beginning when cycle start after M30 is executed

PPD =1: Relative coordinate is set by G50 in absolute zero programming

=0: Relative coordinate is not set by G50 in absolute zero programming

G50RP =1: G50 reference tool nose position, that is the current coordinate system is set directly when absolute coordinate system is set by G50

=0: G50 reference tool nose position, that is tool offset value is added as the current absolute coordinate when absolute coordinate system is set by G50

CM98 =0: Input M, S, T codes other than standard ones of CNC, corresponding alarm will be issued in CNC

=1: Input M, S, T codes other than standard ones of CNC, CNC will not issue alarm, but automatically call a corresponding subprogram

1.1 : M code, When M codes other than standard ones of CNC are executed, the

Part Five Appendix

subprogram called is：

M○○：call subprogram 90○○.

1.2 ：S code，when CNC performs S10～S99，the subprogram called are：

S□□ ：call subprogram 91□□

Note : When spindle analog voltage is output, S code does not call the subprogram.

1.3 ：Tcode，when CNC performs T10～T99，the subprogram called are：

T△△ ：call subprogram 92△△.

Note 1: When M, S, T codes other than standard ones of CNC, corresponding subprogram should be input , otherwise, 078 alarm will be issued.

Note 2: Non-standard M, S, T codes can not be performed in MDI mode. Otherwise, M/S or T alarm will be issued.

Note 3:

HDZ =1： In manual mode, it is the positive direction of the Z-axis by moving left arrow key

=0： In manual mode, it is the positive direction of the Z-axis by moving right arrow key

HDX =1： In manual mode, it is the positive direction of the X-axis by moving up arrow key

=0： In manual mode, it is the positive direction of the X-axis by moving down arrow key

Note: If the direction of the manual arrow key on the panel is different from that of the actual moving direction of the machine, alter the direction of the selected axis of this parameter.

Standard setting：0 0 1 1 0 0 0 0

Parameter No.

0	0	5		WSFT	ISOT	USEJK	CMZ	CPZ		ZMZ	ZMX

WHFT =1： Workpiece coordinate translation is active

=0： Workpiece coordinate translation is inactive

ISOT =1： After power-on or emergency stop, and reference point is not returned, manual rapid is active

=0： After power-on or emergency stop, and reference point is not returned, manual rapid is inactive

USEJK =1：use JK retract valid

=0：use JK retract invalid

CMZ =1： Machine zero return is not allowed

=0： Machine zero return is allowed

CPZ =1： Program zero return is not allowed

=0： Program zero return is allowed

ZMZ =1： When power on, direction of the Z-axis reference return and initial backlash direction is positive

=0： When power on, direction of the Z-axis reference return and initial backlash direction is negative

ZMX =1： When power on, direction of the X-axis reference return and initial backlash direction is positive

=0： When power on, direction of the X-axis reference return and initial backlash direction is negative

Standard setting：0 1 0 0 0 0 1 1

Parameter No.

0	0	6			APRS	ZCZ	ZCX			RTMZ	RTMX

APRS =1： Coordinate system is set automatically after returning to the reference point

=0： Coordinate system is not set after returning to the reference point

ZCZ =1： Z-axis is magnetic switch zero return type C

=0： Deceleration switch and zero signal are necessary for machine zero return, return type B

ZCX =1： X-axis is magnetic switch zero return type C

=0： Deceleration switch and zero signal are necessary for machine zero return, return type B

RMTZ =1： Z-axis with machine zero point

=0： Z-axis without machine zero point

RMTX =1： X-axis with machine zero point

=0： X-axis without machine zero point

Standard setting：0 1 0 0 0 0 0 0

Parameter No.

0	0	7		HAND	HDSTL		BDEC	RCUR		DIRZ	DIRX

HAND =1： MPG CW rotation is positive direction

=0： MPG CW rotation is negative direction

HDSTL =1： Handhold unit

=0： Common MPG

BDEC =1：backlash compensation at the speed of ascending and descending

=0：backlash compensation at the steady rate

RCUR =1：cursor return to beginning of program by pressing RESTE

=0：cursor do not return to beginning of program by pressing RESTE

DIRZ =1： Z-axis motor rotation direction is positive

=0： Z-axis motor rotation direction is negative

DIRX =1： X-axis motor rotation direction is positive

=0： X-axis motor rotation direction is negative

Standard setting：0 0 0 0 0 0 0 0

Parameter No.

0	0	8		CLRT	CKR	LTVZ	LTVX	LMTZ	LMTX	ALMZ	ALMX

CLRT =1：for tool offset 00-064, press XYZ, invalid.

=0：for tool offset 00-064, press XYZ, valid.

CKR =1：Arc do not define R, alarm.

=0：Arc do not define R, no alarm

LTVZ =1： Z-axis hardware limit high level is active

=0： Z-axis hardware limit low level is active

LTVX =1： X-axis hardware limit high level is active

=0： X-axis hardware limit low level is active

LMTZ =1： Detect Z-axis hardware limit

=0： Not detect Z-axis hardware limit

LMTX =1： Detect X-axis hardware limit

=0： Not detect X-axis hardware limit

ALMZ =1： Z-axis drive unit alarm level is low level

=0： Z-axis drive unit alarm level is high level

ALMX =1： X-axis drive unit alarm level is low level

=0： X-axis drive unit alarm level is high level

Standard setting：11 0 0 0 0 1 1

Parameter No.

0	0	9			ACC3			NOFC	DOFSI	TSGN	TCPS

ACC3 =1： rear linear acc/dec

NOFC =0： Input offset is inactive in counting

=1： Input offset is active in counting

DOFSI =0： Input offset is inactive in tool offset measure mode

=1： Input offset is active in tool offset measure mode

TSGN =1： Tool post in-position low level is active

=0： Tool post in-position high level is active

TCPS =1： Tool clamping signal high level is active

=0： Tool clamping signal low level is active

Standard setting：0 0 0 0　1 1 1 0

Parameter No.

0	1	0		AGER	AGIN		AGIM	SPT	TRD	G0&T	TSS

AGER =0： Spindle automatic gearing is inactive

=1： Spindle automatic gearing is active

AGIN =0： Not detect in-position signal when shifting to gear 1,2

=1： Detect in-position signal when shifting to gear 1,2

AGIM =1： Gearing in-position signal is high level active

=0： Gearing in-position signal is low level active

SPT =1： Spindle jog is active in Auto mode

=0： Spindle jog is inactive in Auto mode

TRD =1： Detect tool post in-position

=0： Not detect tool post in-position

G0&T =1： G0 and T codes can not be executed simultaneously

=0： G0 and T codes can be executed simultaneously

TSS =1： Line-up tool post is selected

=0： Common tool post

Standard setting：0 0 0 0　0 1 0 0

Parameter No.

0	1	1								EAL	POD

EAL =0：It is not programmable when alarming

=1：It is programmable when alarming

POD =0： Program by decimal

=1： Not program by decimal

Standard setting：0 0 0 0　0 0 0 1

Parameter No.

0	1	2		WAR2	WAR1			WA2	WA1		RSJG

WAR2 =1： External alarm 2 is active

=0： External alarm 2 is inactive

WAR1 =1： External alarm 1 is active

=0： External alarm 1 is inactive

WA2 =1： External alarm 2 high level is active

=0： External alarm 2 low level is active

WA1 =1： External alarm 1 high level is active

=0： External alarm 1 low level is active

RSJG =1： Outputting M03, M04, lubrication, cooling are not affected by pressing reset key.

=0： M03, M04, lubrication, cooling are stopped when pressing reset key

Standard setting：0 0 0 0　0 0 0 0

Parameter No.

0	1	3		PLCKV	PLCK	M23	M21	CEIO	JLB	RTMC	SBKM

PLCKV =1： Program lock signal is high level active

=0： Program lock signal is low level active

PLCK =1： Program lock signal is active

=0：Program lock signal is inactive

M23 =1： M23 is active

=0： M23 is inactive

M21 =1： M21 is active

=0： M21 is inactive

CEIO =1： Expansion input IO port is closed after reset

=0： Expansion input IO port is not closed after reset

JLB =1： Intermittent lubrication

=0： Continuous lubrication

RTMC =1： Macro common variable is cleared after reset

=0： Macro common variable is not changed after reset

SBKM =1： Single block is active when macro code is executing

=0： Single block is inactive when macro code is executing

Standard setting：0 0 0 0　0 0 0 1

Parameter No.

0	1	4		KEY1	SKEY	CHK		MESP	MOT	M@SP	MST

KEY1 =1： Program switch is on when power on

=0： Program switch is off when power on

SKEY =1： Program switch is shielded

=0: Program switch is not shielded

CHK =1: Not check parameter tool offset

=0: Check parameter tool offset

MESP =1: Emergency stop signal （ESP）is shielded

=0: Emergency stop signal （ESP）is not shielded

MOT =1: Not check software stroke limit switch

=0: Check software stroke limit switch

M@SP =1: External signal （SP）is shielded

=0: External signal （SP）is not shielded

MST =1: External signal （ST）is shielded

=0: External signal （ST）is not shielded

Standard setting：1 0 0 0 1 1 1 1

Parameter No.

0	1	5		STAR	OWAR	OM30	OMST	WZ2	KPDW	SPEN	SPENC

STAR =1: Spindle star delta starting is active

=0: Spindle star delta starting is inactive

OWAR =1: Output alarm signal is active (indicator shall be used to display the CNC is in alarm state)

=0: Output alarm signal is inactive

OM30 =1: Output M30 signal is active

=0: Output M30 signal is inactive

OMST =1: Output STM signal is active

=0: Output STM signal is inactive

WZ2 =1: Tail stock hierarchical advancing/retracing control is active

=0: Tail stock hierarchical advancing/retracing control is inactive

KPDW =1: Check chuck in-position signal

=0: Not check chuck in-position signal

SPEN =1: Feed and spindle enable input are active

=0: Feed and spindle enable input are inactive

SPENC =1: Control feed and spindle

=0: Control spindle

Note: When SPENC and SPEN are 0, spindle enable function is inactive. When SPENC is 0, SPEN is 1, spindle rotation enable function is inactive. When SPENC and SPEN are 1, spindle and feed are enabled if SAR signal is connected. While SAR signal is not connected, spindle is enabled, feed is forbidden when SPEN signal is connected. When SAR and SPEN signals are not connected,

spindle and feed are forbidden.

Standard setting：0 0 0 0　0 0 0 0

Parameter No.

0	1	6		NWKP	SFDV	SFDR	PSCK	PSCV	SLTW	SLSP	SLQP

NWKP =1： Outer chuck mode of the chuck

=0： Inner chuck mode of the chuck

SFDV =1： Protection door is high level active

=0： Protection door is low level active

SFDR =1：Check protection door in cycle start, operation is not allowed when protection door is open

=0： Not check protection door in cycle start

PSCK =1：Check pressure signal in cycle start, it is not allowed

=0： Not check pressure signal in cycle start

PSCV =1： Pressure detect signal is high level active

=0： Pressure detect signal is low level active

SLTW =1：Tail stock function is active

=0： Tail stock function is inactive

SLSP =1： Detect chuck clamping when spindle starts

=0： Not detect chuck state when spindle starts

SLQP =1： Chuck function is active

=0： Chuck function is inactive

Standard setting：0 0 0 0　0 0 0 0

1.3. Data Parameter

GSK980TA3

Parameter No.	Parameter definition	Default value
0017	X-axis coordinate command multiplier ratio CMRX	1
0018	Z-axis coordinate command multiplier ratio CMRZ	1

CMRX，CMRZ：Command multiplier ratio of each coordinate, which is related to gear ratio, screw-pitch error and pulse equivalent.

Setting range: 1～65535

0019	X-axis command frequency division coefficient CMDX	1

0020	Z-axis command frequency division coefficient CMDZ	1

CMDX，CMDZ：Frequency division coefficient of each axis, which is related to gear ratio, screw-pitch error and pulse equivalent.

Setting range: 1～65535

0021	X-axis rapid traverse rate （mm/min）	Step 3000 Servo 4000

Setting range: 30～30000 (Maximum value shall not exceed 15000 in diameter programming. In diameter programming, the actual speed is twice of the specified value)

0022	Z-axis rapid traverse rate （mm/min）	Step6000 Servo 8000

Setting range: 30～30000

0023	X-axis linear ACC&DEC time constant (used for rapid traverse)	Step200 Servo 60

Setting range: 8～4000 (Set above 300 when using stepping drive unit)

0024	Z-axis linear ACC&DEC time constant (used for rapid traverse)	Step200 Servo 60

Setting range: 8～4000 (Set above 300 when using stepping drive unit)

0025	X-axis S ACC&DEC time constant (used for rapid traverse)	60

Setting range: 8～4000 (Set above 300 when using stepping drive unit)

0026	Z-axis S ACC&DEC time constant (used for rapid traverse)	60

Setting range: 8～4000 (Set above 300 when using stepping drive unit)

0027	Upper speed of cutting feeding （mm/min）	8000

Upper speed of cutting feeding applies to all coordinates.

Setting range: 0～8000 Unit: mm/min

0028	Lower value of the ACC&DEC low speed in cutting feed （mm/min）	200

Setting range：0～8000 Unit：mm/min

0029	S acceleration time constant in cutting feed	Step200 Servo 60

Setting range: 8～4000 (Set above 300 when using stepping drive unit)

0030	S deceleration time constant in cutting feed	Step200 Servo 60

Setting range: 8～4000 (Set above 300 when using stepping drive unit)

0031	Linear ACC&DEC time constant in cutting feed	Step200 Servo 60

Setting range: 8～4000 (Set above 300 when using stepping drive unit)

0032	Minimum speed Fo of rapid override （mm/min）	100

Setting range：0～8000 Unit：mm/min

0033	Pulse frequency of backlash offset (Shared by axes)	80

Setting range：1～16384 Unit: Hz (Set to 5 when servo motor is fixed. Set to 3 when stepping motor is fixed)

0034	Speed of the backlash offset （mm/min）	40

Setting range：1～1000 Unit: mm/min

0035	Linear ACC&DEC time of the backlash offset	40

Setting range：1～1000

0036	X-axis offset value （μm）	0

Setting range：0～65535

0037	Z-axis offset value （μm）	0

Setting range：0～65535

0038	Screw-pitch error compensation override (Shared by axes)	1

Setting range：1～8

0039	X-axis screw-pitch error compensation origin	0

Setting range：0～255

0040	Z-axis screw-pitch error compensation origin	0

Setting range：0～255

0041	X-axis screw-pitch compensation interval （μm）	1000

Setting range：100～9999999　Unit：μm

0042	Z-axis screw-pitch compensation interval （μm）	1000

Setting range：100～9999999　Unit：μm

0043	Low speed when returning to the reference point, （FL） speed （mm/min）	200

Setting range：6～600 Unit：mm/min

0044	After reset, X value of coordinate system is set by CNC automatically（μm）	0

After reset, CNC automatically sets the absolute coordinate value to the one specified by coordinate system

Setting range: ±9999999　Unit：μm

0045	After reset, Z value of coordinate system is set by CNC automatically（μm）	0

Setting range：±9999999　Unit：μm

0046	X-axis positive stroke limit （μm）	9999999

Setting range：±9999999　Unit：μm

0047	X-axis negative stroke limit （μm）	-9999999

Setting range：±9999999　Unit：μm

0048	Z-axis positive stroke limit （μm）	9999999

Setting range：±9999999 Unit：μm

0049	Z-axis negative stroke limit （μm）	-9999999

Setting range：±9999999 Unit：μm

0050	In programming, increment for inserting program sequence number automatically	10

Setting range：0～9999

0051	Spindle analog adjusted data	999

Setting range：-999～999

0052	Line number of spindle encoder	1024

Setting range：1024. 1200

0053	Gear ratio of spindle and encoder: spindle gear number	1

Setting range：1～65535

0054	Gear ratio of spindle and encoder: spindle gear number	1

Setting range：1～65535

0055	When spindle speed is 10V, spindle speed at gear 1	9999

Setting range：1～9999

0056	When spindle speed is 10V, spindle speed at gear 2	9999

Setting range：1～9999

0057	When spindle speed is 10V, spindle speed at gear 3	9999

Setting range：1～9999

0058	When spindle speed is 10V, spindle speed at gear 4	9999

Setting range：1～9999

0059	Threading is performed after percentage is stable.	95

Setting range：0～100

0060	Spindle Max. programminG Codespeed	9999

Setting range：1～9999

0061	Spindle analog adjusted data (Adjust it slightly)	0

Setting range：-999～999. Suppose the spindle is accurate at high speed, if it is not accurate at low speed, adjust it to an appropriate value.

0062	Spindle speed sampling period (x4ms)	Step20 Servo 1

Setting range：1～200

0063	Gear shifting time 1(x4ms)	50

Setting range：0～9999999　Unit：×4ms

0064	Gear shifting time 2(x4ms)	50

Setting range：0～9999999　Unit：×4ms

0065	Max. cutting feed speed of each rotation （mm/r）	500

Setting range：0～500　Unit：mm/r

0066	Rotational speed when spindle gear shifting	0

Setting range：0～9999　Unit：r/min

0067	Spindle lower speed under the constant surface speed control （r/min）	100

Setting range：0～9999　Unit：r/min

0068	Chamfering width of screw cutting （THDCH）	10

Setting range：0～99　Chamfering width of screw＝THDCH×1/10×screw-pitch

0069	X-axis linear ACC&DEC constant in thread cutting （G92）	Step200 Servo 40

Setting range：6～4000

0070	Lower value of S type ACC&DEC of each axis in thread cutting （G92）	Step200 Servo 40

Setting range：6～4000　Unit：mm/min

0071	Cutting depth in multiple fixed cycle （G71，G72）	0

Setting range：0～9999999　Unit：0.001mm

0072	Tool retraction in multiple fixed cycle （G71，G72）	0

Setting range：0～9999999　Unit：0.001mm

0073	X-axial total cutting value in multiple fixed cycle （G73）	0

Setting range：-9,999,999～9,999,999　Unit：0.001mm

0074	Z-axial total cutting value in multiple fixed cycle （G73）	0

Setting range：-9,999,999～9,999,999　Unit：0.001mm

0075	Cycle cutting times in multiple fixed cycle （G73）	0

Setting range：0～9999999　Unit：times

0076	Tool retraction in multiple fixed cycle （G74，G75）	0

Setting range：0～9999999　Unit：0.001mm

0077	Repetitions of finish matching in multiple fixed cycle （G76）	0

Setting range：0～9999999　Unit：time

0078	Tool nose angle in multiple fixed cycle （G76）	0

Setting range：0，29，30，55，60，80

0079	Min. cutting depth in multiple fixed cycle （G76）	0

Setting range：0～9999999　Unit：0.001mm

0080	Finish allowance in multiple fixed cycle （G76）	0

Setting range：0～9999999 Unit：0.001mm

0081	Total tool number selection	4

Setting range：0～8

0082	Tool changing time T1 (x4ms)	30

Setting range：0～9999999 Unit：×4ms

0083	Upper time for moving a tool in tool changing (x4ms)	1000

Setting range：0～9999999 Unit：×4ms

0084	Upper time from changing the 1st tool to changing the last tool (x4ms)	15000

Setting range：0～9999999 Unit：×4ms

0085	Tool post CW rotation clamping time (x4ms)	250

Setting range：0～9999999 Unit：×4ms

0086	M code execution time (x4ms)	1

Setting range：0～9999999 Unit：×4ms

0087	S code execution time (x4ms)	1

Setting range：0～9999999 Unit：×4ms

0088	Alarm time of not receiving *TCP signal (x4ms)	500

Setting range：0～9999999 Unit：×4ms

0089	Duration from spindle command stop to spindle brake output (x4ms)	100

Setting range：0～9999999 Unit：×4ms

0090	Spindle brake output time (x4ms)	100

Setting range：0～9999999 Unit：×4ms

0091	Chuck clamping in-position time	100

Setting range：0～9999999　Unit：×4ms

0092	Setting of pressure detection time (x4ms)	100

Setting range：0～9999999　Unit：×4ms

0093	Star triangle starting time constant (x4ms)	100

Setting range：0～9999999　Unit：×4ms

0094	Spindle jog dwell time (x4ms)	100

Setting range：0～9999999　Unit：×4ms

0095	Baud rate of serial communication	9600

Setting range：4800. 9600. 19200. 38400. 57600. 115200

0096	(x4ms)Page starting time (x4ms)	Step600 Servo 100

Setting range：0～9999 Unit：×4ms

0097	Duration time for signal removing vibration	2

Setting range：2～15（Do not change standard setting without special circumstances）

0098	Error limit of circular arc 2*R (um)	2000

Setting range：1～9999999

0099	Undefined	0

0100	Undefined	0

0101	Tool retraction error adjustment value of G33 tapping (adjustable range from -400 to 400)	Step-45 Servo 12

Tool retraction error adjustment parameter G33 of tapping, which is adjusted and used according to the actual circumstances (converter spindle, gear spindle, matching with stepping drive or servo drive etc. are fixed). In principle, turn it up if tool retraction lags behind, and turn it down if tool retraction performs ahead of time.

Setting range: -400～400, no unit

0102	Degree of corner between lines in post ACC&DEC mode	30

The smaller the precision is, the higher the efficiency is, and vice versa. The angle is the biggest when it is 0%, while there is no angle when it is 100%. Namely, it is a linear. Corners between lines are formed automatically in post ACC&DEC mode, which are affected by ACC&DEC time and current speed. When non-mark connection is required, set this parameter to 0%.

Setting range: 0～100

0103	Initial speed of cutting (mm/min)	40

After power on, interpolation speed is not specified by the program when interpolation command is being executed, perform interpolation at speed specified by this parameter. When this parameter is 0, and the speed is not specified by the program, alarm will be issued by CNC.

Setting range: 0～2000

0104	Reversing speed of reference return in C mode (mm/min)	45

In return type C, when returning to reference point, it touches deceleration block and decelerates. After departing from the deceleration block, it continuously travels short distance and then returns. The parameter sets the reversing speed. The bigger the value is, the worse the precision is.

Setting range: 0～1000 Unit: mm/min

0105	M12, M13 routine output/pulse output time (x4ms)	0

When the chuck control signal is controlled by pulse, set signal output time by this parameter. If the specified time is delayed, CNC cancels outputting signal, while the state of the CNC is not changed. When this parameter is set to 0, the signal is long output signal until the command is changed.

Setting range: 0～9999999 Unit: ×4ms

0106	Undefined	0

0107	Intermittent lubrication time	1000

Setting range: 0～9999999 Unit: ×4ms

0108	Pause time of intermittent lubrication	1000

Setting range：0～9999999 Unit：×4ms

Part Five Appendix

0109	Rapid speed when returning to the reference point （mm/min）	6000

Setting range：0～9999999

0110	Undefined	0
0111	Undefined	0

0112	Undefined	0

By setting the following parameter, it is able to set whether the output signal is pulse or level. 0: level Non zero: pulse

0113	S01 output time (x4ms)	0

Setting range：0～9999999 Unit：×4ms

0114	S02 output time (x4ms)	0

Setting range：0～9999999 Unit：×4ms

0115	S03 output time (x4ms)	0

Setting range：0～9999999 Unit：×4ms

0116	S04 output time (x4ms)	0

Setting range：0～9999999 Unit：×4ms

0117	M03 output time (x4ms)	0

Setting range：0～9999999 Unit：×4ms

0118	M04 output time (x4ms)	0

Setting range：0～9999999 Unit：×4ms

0119	M05 output time (x4ms)	0

Setting range：0～9999999 Unit：×4ms

0120	To be added	0

Setting range：0～9999999 Unit：×4ms

0121	M08 output time (x4ms)	0

Setting range：0～9999999 Unit：×4ms

0122	M10 output time (x4ms)	0

Setting range：0～9999999 Unit：×4ms

0123	M11 output time (x4ms)	0

Setting range：0～9999999 Unit：×4ms

0124	To be added	0

Setting range：0～9999999 Unit：×4ms

0125	To be added	0

Setting range：0～9999999 Unit：×4ms

0126	M30 output time (x4ms)	0

Setting range：0～9999999 Unit：×4ms

0127	M32 output time (x4ms)	0

Setting range：0～9999999 Unit：×4ms

0128	To be added	0

Setting range：0～9999999 Unit：×4ms

0129	To be added	0

Setting range：0～9999999 Unit：×4ms

0130	STM output time (x4ms)	0

Setting range：0～9999999 Unit：×4ms

0131	WARN output time (x4ms)	0

Setting range：0～9999999 Unit：×4ms

0132	STAR output time (x4ms)	0

Setting range：0～9999999 Unit：×4ms

0133	TRIAN output time (x4ms)	0

Setting range：0～9999999　　Unit：×4ms

0134	TLOCK output time (x4ms)	0

Setting range：0～9999999　　Unit：×4ms

0135	M51 output time (x4ms)	0

Setting range：0～9999999　　Unit：×4ms

0136	M53 output time (x4ms)	0

Setting range：0～9999999　　Unit：×4ms

0137	M55 output time (x4ms)	0

Setting range：0～9999999　　Unit：×4ms

0138	M57 output time (x4ms)	0

Setting range：0～9999999　　Unit：×4ms

0139	M59 output time (x4ms)	0

Setting range：0～9999999　　Unit：×4ms

0140	M61 output time (x4ms)	0

Setting range：0～9999999　　Unit：×4ms

0141	M63 output time (x4ms)	0

Setting range：0～9999999　　Unit：×4ms

0142	M65 output time (x4ms)	0

Setting range：0～9999999　　Unit：×4ms

0143	M67 output time (x4ms)	0

Setting range：0～9999999　　Unit：×4ms

0144	M69 output time (x4ms)	0

Setting range：0～9999999　　Unit：×4ms

0145	G31 type: 0 is for standard; >0 is for expansion (see manual for details)	0

Setting range：0～9999999 Unit：×4ms

0146	Number of G31 command signal input port and high/low level (default 151)	151

Setting range：0～9999999 Unit：×4ms

0147	To be added	0

Setting range：0～9999999 Unit：×4ms

0148	Tool post expansion type	0

Setting range：0～9999999

0149	To be added	0

Setting range：0～9999999 Unit：×4ms

0150	To be added	0

Setting range：0～9999999 Unit：×4ms

0151	To be added	0

Setting range：0～9999999 Unit：×4ms

0152	1Tool post expansion auxiliary time 1	0

Setting range：0～9999999 Unit：×4ms

0153	Tool post expansion auxiliary time 2	0

Setting range：0～9999999 Unit：×4ms

0154	Tool post expansion auxiliary time 3	0

Setting range：0～9999999 Unit：×4ms

0155	Tool post expansion auxiliary time 4	0

Setting range：0～9999999 Unit：×4ms

0156	Pulse stationary constant	0

Setting range：0～9999999　　Unit：×4ms

0157	Linear ACC&DEC time constant in thread run-out	Step200 Servo 40

Setting range：6～999　　Unit：×4ms

0158	Upper speed of the thread run-out	Step6000 Servo 7600

Setting range：30～30000　Unit：×4ms

0159	Additional alarm time of start key	0

Setting range：0～9999999　　Unit：×4ms

Note: When it is set to be 0, system default to alarm after 3s by pressing start key, so the alarm time start point is 3s.

0160	To be added	0

Setting range：0～9999999　　Unit：×4ms

0161	If Lubrication alarm is valid(0:invalid, 1:valid)	0

Note: parameter 162~167 are to be added parameters.

0177	CS axis type 1 2 3	0

0178	CS check input point max. overtime	0

Setting range: 0～9999999　　Unit:×4ms

0179	CS axis clamp delay time	0

Setting range：0～9999999　　Unit:×4ms

0180	CS axis positioning coordinate of Y axis, negative value do not input .	0

Setting range：0～9999999

0181	CS output delay time before torque reduction	0

Setting range：0～9999999　　Unit:×4ms：×4ms

0182	Interpolation period（0：2MS；1：4MS）	0

Setting range：0～9999999　　Unit:×4ms：×4ms

0183	System type（1：TA；2：TB）	0

Setting range：0～9999999　　Unit:×4ms：×4ms

0184	+Hard limit input point of X axis(valid after restart)	0

Setting range：0～9999999　　Unit:×4ms：×4ms

0185	-Hard limit input point of X axis(valid after restart)	0

Setting range：0～9999999　　Unit:×4ms：×4ms

0186	+Hard limit input point of Z axis(valid after restart)	0

Setting range：0～9999999　　Unit:×4ms：×4ms

0187	-Hard limit input point of Z axis(valid after restart)	0

Setting range：0～9999999　　Unit:×4ms：×4ms

0188	3rd section speed of each axis zero (valid after restart)	0

Setting range：0～9999999　　Unit:×4ms：×4ms

0189	Tail control type 0/1/2 (valid after restart)	0

Setting range：0～9999999　　Unit:×4ms：×4ms

0190	Truck control type 0/1/2 (valid after restart)	0

Setting range：0～9999999　　Unit:×4ms：×4ms

0191	Truck manual control allowable speed (valid after restart)	0

Setting range：0～9999999　　Unit:×4ms：×4ms

Note: parameter 192～256 are to be added.

System parameter No.

8	**0**	**1**		**SPI**	**SPC**	**CSL**	**CSP**	**CSZ**	**CSM**	**YCS**	**PEMG**

SPI =1：spindle clamp input high-level valid

=0：spindle clamp input low-level valid

SPC =1：check spindle clamp input

=0：Do not check spindle clamp input

CSL =1：CS pos input high-level valid

=0：CS pos input low-level valid

CSP =1：CS position mode high-level valid

=0：CS position mode low-level valid

CSZ =1：CS zero speed high-level valid

=0：CS zero speed low-level valid

CSM =1：CS axis control M code type

=0：CS axis do not control M code type

YCS =1：Y axis set as CS axis

=0：Y axis do not set as CS axis

PEMG =1：MPG emergency stop signal high-level valid

=0：MPG emergency stop signal low-level valid

Standard setting：0 0 0 0　0 0 0 0

System parameter No.

8	0	2		TAR	TL1	TL0	TM1	TM0	TR1	TR0	ALMC

TAR =1：spindle alaem high-level valid

=0：spindle alaem low-level valid

TL1 =1：spindle alaem high-level valid

=0：spindle alaem low-level valid

TL0 =1：spindle alaem high-level valid

=0：spindle alaem low-level valid

TM1 =1：spindle alaem high-level valid

=0：spindle alaem low-level valid

TM0 =1：spindle alaem high-level valid

=0：spindle alaem low-level valid

TR1 =1：spindle alaem high-level valid

=0：spindle alaem low-level valid

TR0 =1：spindle alaem high-level valid

=0：spindle alaem low-level valid

ALMC　=1：spindle alaem high-level valid

=0：spindle alaem low-level valid

standard setting：0 0 0 0　0 0 0 0

System parameter No.

8	0	3							VMB	LTC	XZL

VMB　=1：spindle alaem high-level valid

=0：spindle alaem low-level valid

LTC　=1：spindle alaem high-level valid

=0：spindle alaem low-level valid

XZL　=1：spindle alaem high-level valid

=0：spindle alaem low-level valid

Standard setting：0 0 0 0　0 0 0 0

Parameter 192～256 are to be added.

APPENDIX II　ALARM LIST

No.	Content	Trouble shooting
000	Parameter for cutting off the primary power source is set, please cut off power supply.	Power on the CNC
001	Fail to open file.	Open the right file.
002	This character string can not be found in Edit mode	Wrong operation, and search the existed character string again.
003	Instruction pulse or speed is too big. Reset or tool setting is needed.	Set these parameters again: P21/P22/P23/P24. See parameter list and initialization disk of manual.
004	Address is not found	Wrong operation, sequence number should be put before the program
005	No data follows the address.	Modify the program. See programming manual for details.
006	Negative sign entry error	Modify the program. See programming manual for details.
007	Decimal point entry error	Modify the program. See programming manual for details.
008	File creation is unsuccessful.	Delete the unused program, and then create a new file.
009	Input address of the illegal word	Modify wrong address in the program. See programming manual for details.
010	Inactive G code is specified.	Modify wrong G code in the program. See programming manual for details.
011	F command value of cutting feed is wrong.	Set cutting federate F again. Feedrate per minute: 1-8000mm/min, Feedrate per rev: 0.001-500mm/r
012	Digits of input value exceed the maximum allowable range	Modify input data. See programming manual for details.
013	The program specified the program number or the address to be searched are not found.	Wrong operation, and search the existed program number again.
014	The file to be deleted does not exist.	Delete the existed file again.
015	Storage capacity of the memory is insufficient.	Delete unneeded program
016	In thread interpolation, spindle speed is low	Start spindle when turning thread. Check

	or spindle is not started.	whether the speed of the peripheral spindle is normal or whether the spindle is started.
017	Zero clearing is wrong	Modify the program
018	F,I command values are wrong in thread cutting.	Set thread lead again, metric thread 0.001-500mm, inch thread 0.06-25400 tooth/inch.
019	The length of the block exceeds 255 characters.	Delete unneeded characters
020	F value is not specified in tapping cycle.	Lead F value is needed in tapping processing
021	In program zero return, zero return fails if there is no zero or zero point is at the edge of the block.	Firstly execute G50, and then perform program zero return. Adjust position of zero return block, and perform zero return again.
022	The factory setting value can not be covered.	When operate illegally, press reset key to cancel the alarm.
023	When circular interpolation defined by radius R is used, R defined a negative value.	Circular radius R should not be a negative value, modify the program.
024	Radius of circular interpolation is too big.	Modify radius R. See programming manual.
026	The value specified in circular interpolation can not form a circular arc.	Modify program, radius, starting coordinate value or ending coordinate values of the circular arc.
027	In circular interpolation, R and (K,I) are specified simultaneously.	Either R or (K,I) is used to specify radius of circular arc. Delete unneeded characters.
028	In circular interpolation, R and (K,I) are zero.	Specify radius of circle correctly when checking program.
029	The offset value specified by T code is too big.	Perform tool setting again, modify the offset value specified by T code.
030	Tool offset number exceeds the range.	Modify tool offset number, the range: 1-64
031	The file can not be found	Search the file again
032	Communication error	Check whether the baud rate of CNC is consistent with that of the computer
033	There is no cross point in tool nose radius compensation	Modify the program
034	G2 or G3 is executed when starting or canceling the tool radius compensation.	G2 or G3 can not be performed when starting or canceling the tool compensation. Modify the program.
035	Compensation direction of the next block changes after the compensation is started.	Tool compensation direction shall not be changed after the compensation is started. Modify the program.

036	Tool radius compensation is performed before tool offset number is specified.	Tool offset number shall be specified before executing tool compensation. Modify program, e.g. Add T0101.
037	Block is non-movinG Codewhen compensation starts.	Block shall not be non-movinG Codewhen compensation starts.
038	In tool nose radius compensation, excessive cutting occurs as starting point or terminal point of the circular arc coincides with the center point,	Modify the program
039	30 non-movinG Codes appear continuously in the process of tool compensation.	Delete unneeded blocks.
040	Illegal command appears in the process of tool compensation.	Check the program. See programming manual for details.
041	Excessive cutting will occur in tool nose radius compensation	Check the program. See programming manual for details.
042	In the process of tool nose radius compensation, G code is specified repeatedly in circular arc command.	Modify the program. See programming manual for details.
043	Data input exceeds the allowable range.	Modify data or check whether the destination address value exceeds the range.
044	Spindle disabled if chuck is not clamped.	Clamp the chuck before starting the spindle.
045	Chuck unclamping is not allowed when spindle is in operation.	Execute M05 before unclamping the chuck.
046	Storage is wrong at power off. Please power on the CNC.	Coordinate value is too big. Cancel the alarm by pressing reset key, and power on again.
047	Spindle CW (rotates negatively) is specified before stopping spindle CCW (rotates positively).	Perform M05, an then execute M03 or M04.
050	An unusable tool nose angle is specified in G76.	Modify the program, and specify a correct degree. See programming manual for details.
051	In G76, minimum specified cutting depth or finishing allowance is bigger than thread height.	Modify the program, and specify a correct cutting depth. See programming manual for details.
052	Thread height or initial cutting depth specified in G76, which is 0 or negative number.	Modify the program. See programming manual for details.
053	P or Q value is not specified in G70, G71,G72,G73 code.	Modify the program. See programming manual for details.

054	# I or # K in G74 or G75 is a negative value.	Modify the program. See programming manual for details.
055	. G76 finishing allowance is a negative value.	Range of finishing allowance: 0-9999.999 mm. Modify the program.
062	(1) Cutting depth in G71 or G72 is 0 or negative value. (2) G73 repetition time is 0 or negative value (3) D-i or D-k command in G74 or G75 is negative value (4) While D-i or D-k command in G74 or G75 is not 0, address U or W is specified as 0 or negative value (5) While tool retraction direction of G74 or G75 is specified, D-d is a negative number.	Modify the program. See programming manual.
063	In G70, blocks specified by P or Q can not be found.	Modify the program, and add correct blocks.
064	Tool compensation is not cancelled when G code is G32,G33,G90～G94,G70～G76.	The above G codes can not be performed in tool compensation state. Execute G40 first.
065	(1) In code G71,G72 or G73, in the block whose sequence number is specified by address P, G00 or G01 is not defined. (2) In code G71 or G72, in the block whose sequence number is specified by address P, address Z(W)(G71) or X(U)(G72) are defined.	The first block in NS～NF should be G00 or G01. In the first block of NS～NF specified by G71, X or U shall be defined separately. In the first block of NS～NF specified by G72, Z or W shall be defined separately.
066	In code G70, G71,G72,G73, in the blocks specified by P or Q, non-movinG Codeor unusable G code is specified.	Modify the program. See programming manual for details.
067	M98,M99 or M30.is specified in the blocks that is specified by address P or Q.	Modify the program and delete incorrect codes.
068	P or Q value of G70 is specified repeatedly.	Modify the program. See programming manual for details.
074	The program number is out of the range of 0001-9999.	Modify the program.
076	In the block M98, P is not specified.	Modify the program, and specify the correct P code.
077	Subprogram calls too many nests.	Called subprograms exceed the limit. Subprogram is a four-embedded program.

		Modify the program nest structure.
078	The specified program number or sequence number has not been found in subprogram call.	Modify the program, and check the program number or sequence number specified by P.
079	Macro code alarm does not exist.	Specify the correct macro code alarm number.
081	NC program and customer macro program exist simultaneously.	Modify the program. See programming manual for details.
100	Parameter switch is ON.	Set parameter switch to the state of "OFF" (In MDI mode, press character "W" on the setting page), or press reset key.
101	In program editing, power is off. Please edit the program again.	Press reset key to cancel the alarm, and edit the program again.
102	The number of lines exceed the maximum value.	Delete redundant lines.
103	Memory data of parameter, tool compensation and coordinate are initialized.	Read initialized disk again. If they still exist after power on, please change the main board.
104	Because CNC abnormality is detected, it needs to check parameter again.	Read initialized disk again. If they still exist after power on, please change the main board.
105	It detects data of tool length compensation is abnormal. Please set the tool again.	Read initialized disk again. If they still exist after power on, please change the main board.
106	It detects data of the absolute data is abnormal. Please set tool again.	Read initialized disk again. If they still exist after power on, please change the main board.
107	It detects the drive, spindle etc. are changing.	Restart the system again.
110	When copied and renamed, the program name already exists	Select a new program name, and perform again.
114	In G65 block, undefined H code is specified.	Check program, and specify the correct H code.
115	Illegal variable number is specified.	Modify the program, and specify the correct variable number.

128	When transferring the command, the sequence number of address to be transferred is not in the allowable range, or it is not found.	Modify the program, and use the correct sequence number.
129	A block must not contain identical function words or same series of G code.	Modify the program. See programming manual for details.
130	The value of the address exceeds the range.	There are specified ranges for addresses. Modify the program. See programming manual for details.
131	Group G00 and group G01 must not appear at the same time.	Modify the program, and delete redundant G code.
132	Coordinate value X and U or Z and W appear at the same time.	Modify the program, and delete redundant coordinate value.
133	Two repeatable canned cycle G code are specified continuously.	Modify the program. See the programming manual for details.
134	When specifying G32, G33, G92, I, F appear at the same time.	Only F or I specify the screw pitch. Delete unwanted characters.
137	Too many blocks in repeatable canned cycle G71 or G72, G73.	Number of block shall not exceed 100. Delete unwanted blocks.
138	N must be increased progressively.	Modify sequence numbe N. It must be increased progressively.
140	There must be a space between addresses.	Modify the program. See programming manual for details.
141	Constant speed command is specified when the spindle uses gear control.	When using constant speed, spindle shall use frequency conversion control. Change parameter NO.1Bit4 to 1.
142	Program ends illegally without M30 or M99.	Modify the program. See programming manual for details.
144	Infeed value, retraction value, chamfer allowance, finishing times or teeth height of the screw cutting in G70, G71, G72, G73, G74, G75, G76 are wrong.	Modify the program. See programming manual for details.
145	The value of address in G70, G71, G72, G73, G74, G75, G76	Modify the program. See programming manual for details.
146	Variable shall not be used by address O and N.	Address O and N should not be used for macro variable. Modify the program.
147	P in G65 should not be a constant.	Modify the program. See programming manual for details.

149	Tool radius compensation is not cancelled before entering or exiting the subprogram.	Tool compensation should be cancelled before calling the subprogram. Add G40.
150	Some coordinates are not changed monotonously in repeatable canned cycle.	Check the program. See the programming manual for details.
151	U and W in G71 and G72 code are not consistent with the track path, or U(W) do not exist or the values exceed the range. The coordinates are not changed monotonously.	Check the program. See the programming manual for details.
152	Exceed the circumstances that G76,G90,G92,G94can process.	Check the program. See the programming manual for details.
153	F, I values in G33, G92 codes are wrong.	Set thread lead again, metric thread 0.001-500mm, inch thread 0.06-25400 tooth/inch.
155	Constant surface speed command is wrong.	Check the program. See the programming manual for details.
156	In G71, G72, G73, the blocks specified by P are not closed to each other. G71, G72, G73 blocks or the blocks specified by P or Q do not exist.	Check the program. See the programming manual for details.
158	G70,G71,G72 or the P value specified by G73 are bigger than or equal to Q value.	P must be bigger than Q. Modify the program.
162	P is non-integral value or a negative value in the blocks of the called program.	Check the program. See the programming manual for details.
163	Subprogram is called too many times.	Subprogram call times: 1-999. Modify the program.
164	Exceed the maximum storage number (500)	Delete unnecessary program.
165	The sentence for calling subprogram or returning subprogram is in illegal block (G90-G94, G70～G76).	Check the program. See the programming manual for details.
166	Pno and Qno values assigned by G70, G71, G72, and G73 exceed the range or they are non-integral.	Check the program. See the programming manual for details.
168	P,U and X in the pause command can not be specified at the same time.	Delete unnecessary address characters.
169	Q and R in macro program are specified incorrectly.	Check the program. See the programming manual for details.
170	Please input correct password in setting	It is necessary to input password before

	interface before modifying parameter and screw pitch compensation data.	modifying parameter and screw pitch. See programming manual for details.
171	Spindle enable signal is not received.	Check SPEN input state of parameter NO.3, and check whether circuit of the external spindle enable signal is normal, or change parameter NO.15 Bit1 to 0: detection for spindle enable signal is shielded.
180	M41～M44 can not be used when spindle is in gear control or spindle automatic gearing is inactive.	Requirements for using M41-M44: frequency conversion and spindle automatic gearing is active. Change parameter NO.1 Bit4 and NO.10 Bit7 to 1.
181	M code is wrong. Illegal M code is edited in the program.	Corresponding parameter is not set. See parameter list of the manual.
182	S code is wrong. Illegal S code is edited in the program.	Corresponding parameter is not set. See parameter list of the manual.
183	T code is wrong. Illegal T code is edited in the program.	Corresponding parameter is not set. See parameter list of the manual.
185	Time for tool setting is too long. Specified tool arrival signal has not been received Ta time after the start of tool CCW rotation. Alarm occurs.	According to input state of NO.2 T01-T08, check whether the setting of NO.9 Bit1 is correct or adjust P82 tool changing time.
186	Tool post CW rotation locking signal is not received in the time of tool post CW rotation locking time.	According to TCP input state of parameter NO.1 Bit7, check whether the setting of NO.9 Bit0 is correct
187	Tool changing is not ready. Tool number is not consistent with the current tool number.	Press reset key to cancel the alarm or adjust position of signal sending disk, or according to the state of NO.2 T01-T08, adjust NO.9 Bit1 level setting or change NO.10 Bit2 to 0: Shield tool number detection signal.
188	M45--M49 commands are executed while corresponding in-position input signal is not received.	In-position input signal is necessary for executing M45-M49. Check whether the circuit is correct.
189	Chuck in-position signal is not received within the stipulated time.	Chuck in-position detection is active. Check whether the wire of external alarm is correct. Adjust parameter P91 arrival time or modify parameter NO.15 Bit2 to 0: Shield chuck in-position.
190	Axis number is changed, and data	Press reset key to cancel the alarm when

	initialization is successful. After turning off the parameter switch, press reset key to continue.	CNC alarms.
191	Key pressing is overtime.	It is not allowed to press cycle start key continuously. Adjust parameter P159 key pressing time.
193	Please pull out U-disk!	Operation is wrong. U-disk should not be plugged in during processing.
201	Exceed X-axis positive stroke limit.	Modify the range of X-axis positive stroke of the CNC parameter, or cancel alarm by manual negative feed.
202	Exceed X-axis negative stroke limit.	Modify the range of X-axis negative stroke of the CNC parameter or cancel alarm by manual negative feed.
203	Exceed X-axis positive stroke limit.	Modify the range of Z-axis positive stroke of the CNC parameter or cancel alarm by manual negative feed.
204	Exceed Z-axis negative stroke limit.	Modify the range of Z-axis negative stroke of the CNC parameter or cancel alarm by manual negative feed.
205	Exceed Y-axis positive stroke limit.	Modify the range of Y-axis positive stroke of the CNC parameter or cancel alarm by manual negative feed.
206	Exceed Y-axis negative stroke limit.	Modify the range of Y-axis negative stroke of the CNC parameter or cancel alarm by manual negative feed.
207	X-axis hardware overtravel (negative).	When hardware detection is active, check whether the external alarm circuit is correct, or change alarm level parameter NO.8 Bit4, Bit5 or parameter NO.8 Bit2, Bit3 to 0: Shield hardware detection.
208	Z-axis hardware overtravel (negative).	
209	X-axis hardware overtravel (positive).	
210	Z-axis hardware overtravel (positive).	
211	X-axis drive unit is not ready.	Level of the alarm signal is not correct. Adjust CNC parameter NO.8 again. Check whether the drive unit is damaged.
212	Z-axis drive unit is not ready.	
213	X/Z -axis drive unit is not ready.	
221	X-axis drive unit alarm.	Level of the alarm signal is not correct. Adjust CNC parameter NO.8 Bit0, Bit1. Check whether the drive unit is damaged.
222	Z-axis drive unit alarm.	
223	X-axis overtravel alarm.	

Part Five Appendix

224	Z-axis overtravel alarm.	
250	Tailstock advance/retreat can not be performed in Auto mode or after the spindle is started.	Operation is wrong. Execute M05 and then perform M10/M11.
251	External alarm 1.	External alarm 1 is active. Check whether the external alarm circuit is correct, or change alarm level parameter NO.12 Bit2 or NO.12 Bit6 to 0: Shield pressure detection.
252	External alarm 2.	External alarm 2 is active. Check whether the external alarm circuit is correct, or change alarm level parameter NO.12 Bit3 or NO.12 Bit7 to 0: Shield pressure detection.
253	Pressure is too low.	Pressure detection is active. Check whether the pressure circuit is correct, or change alarm level parameter NO.16 Bit3 or NO.16 Bit4 to 0: Shield pressure detection.
254	Preparation is not ready.	CNC is in emergency stop state. Check whether the emergency stop button is pressed down, or change parameter NO.14 Bit3 to 1: Shield emergency stop.
255	Safety door is open in automatic operation.	Operation is wrong. Safety door should not be opened during the operation. Change alarm level parameter NO.16 Bit6 or parameter NO.16 Bit5 to 0: Shield safety door function.
258	IO port setting is wrong. It needs to be set again in PLC-V. Wrong setting of IO port will cause accident. Please contact the manufacturer for help.	Main board has faults. Please contact the manufacturer.
259	Timer has faults. Please contact the manufacturer for help.	Main board has faults. Please contact the manufacturer.

APPENDIX III DIAGONOSIS LIST

1. DI/DO Diagnosis Message

1.1 Input Signal from Machine Side (I/O board)

Input signal diagnosis message from the machine side directly corresponds to CNC hardware circuit. If there is a fault on the hardware circuit, content described in this chapter can be used for confirmation. For the specific position of signals on PCB (printed circuit board), please refer to logic diagram.

1）Diagnosis list:

Diagnosis No.			Significance							
0	0	1	*TCP	DIQP	*DECX	DITW	*SP	ST	DECZ	*ESP
0	0	2	T08	T07	T06	T05	T04	T03	T02	T01
0	0	3	SAR	LTZ	LTX	SPEN	PCH	DOOR	GR2	GR1
0	0	4	M01	M93	M91	LTY	DECY	LCK	OWA2	OWA1

2) Significance of the signal

(a) *TCP： Tool post clamping signal

(b) DIQP： Chuck foot switch (input signal, input is active when it connects to +24V)

(c) DITW： Tailstock input signal

(d) *DECX. *DECZ： Deceleration signal for corresponding axis

(e) *SP： Feed hold signal

(f) ST： Automatic cycle start key

(g) *ESP： Emergency stop signal

(h) T01~T08： Tool post in-position signal

(i) SAR： Spindle speed in-position signal

(j) LTZ， LTX： Stroke limit signal for corresponding axis

(k) SPEN： Spindle enable signal

(l) PCH： Pressure check signal in cycle start

(m) DOOR： Safety door signal

(n) GR1. GR2： Gear 1, 2 shifting in-position signal

(o) M01： Conditional pause signal input port

(p) M91 . M93： Two user-defined input ports

(q) LTY： Stroke limit signal for corresponding axis

(r) DECY： Zero point deceleration signal for corresponding axis

(s) LCK： Keyboard locking signal (temporarily inactive)

(t) OWA1. OWA2： External alarm 1, 2 signal

1.2 Signal Input to Machine Side (I/O board)

When the corresponding bit is "1", its corresponding output transistor in the hardware circuit is connected. If not, it means malfunction in output circuit. For its relationship with hardware, please refer to logic diagram.

1） Diagnosis

Diagnosis No.			Significance							
0	0	5	SPZD	DOQPJ	M5	M32	M8	M10	M4	M3
0	0	6	TL－	TL＋	DOQPS	M11	M44	M43	M42	M41
0	0	7	M53	M51	TLC	STAR	TRIAN	WAR	M30	MST
0	0	8	M69	M67	M65	M63	M61	M59	M57	M55

2) Significance of the signal

(a) SPZD： Spindle brake

(b) DOQPJ： Chuck clamp output signal

(c) M5： Spindle stop

(d) M32： Lubrication ON

(e) M8： Cooling ON

(f) M10： Tailstock advance

(g) M4： Spindle CW

(h) M3： Spindle CCW

(i) TL+. TL-： Tool changing signal

(j) TL+ is positive tool changing signal. TL- is negative tool changing signal.

(k) DOQPS； Chuck releasing output signal

(l) M11： Tailstock retreat

(m) M41~M44： Select spindle frequency conversion control. Gearing output in automatic gear shifting.

(n) STAR. TRIAN： Machine start mode output control (star triangle starts)

(o) WAR： Alarm signal

(p) M30： Program end signal

(q) MST： Automatic run signal

(r) M51～M69； User-defined output port

2. CNC Interface Signal

1） Diagnosis list

Diagnosis No. Significance

Diagnosis No.			Significance							
0	0	9			RFZ	RFR		PCS	PCZ	PCX
0	1	0							ALMZ	ALMX
0	1	1	MPG data							
0	1	2	Spindle feedback data							
0	1	3	Spindle feedback data							
0	1	4	Spindle analog output value							
0	1	5	Spindle analog output value							
0	1	6		*MZRO			TYD	TYC	TYB	TYA

2) Signal significance

(a) PCS： Spindle encoder one turn signal

(b) PCX. PCZ： Zero point signal for corresponding axis

(c) ALMX. ALMZ： Drive unit alarm for corresponding axis

These two diagnosis signals lie in ALM of the hardware circuit, and X stands for x axis, Z stands for z axis. №008 ALMZ，ALMX relate to the active level of this signal. E.g. If set No.8 ALMX to 1, drive unit of corresponding axis will alarm when ALMX is set to 0. On the contrary, corresponding axis will alarm when ALMX is set to 1. If the drive unit you used can not provide this signal, please do not connect this signal when wiring. In addition, №008 ALMZ，ALMX shall be set to 0. When drive unit alarms, please judge troubles at CNC side or at drive unit.

(d) 011：MPG data

MPG data are mainly used for diagnosing the total number of pulse sent from X-axis or Z-axis to the motor in MPG mode. MPG speed can be seen from the speed of data changing.

(e) Spindle feedback data

Number 012 and 013 display low-order 2 digits and high-order 2 digits of the spindle current speed in the form of BCD code.

(f) Spindle analog output value

After spindle speed is defined, it is changed to the speed of controlling analog voltage output through gear and conversion control. Diagnosis number 013 and 014 display low-order 2 digits and high-order 2 digits of the speed after frequency conversion in the form of BCD code.

3） No alarm issues, and the motor does not move

(a) CNC position display does not change

Check whether the CNC message is correct.

(b) CNC position display changes

First, check whether the CNC is in machine lock mode. If it is in this mode, machine coordinate display will not change. Ensure the display is changing.

3. Operation Panel Key

№017~022. 027~028：machine operation panel key diagnosis.

№023~026：keyboard key diagnosis

When the cursor under these diagnosis numbers, details for the corresponding keys are displayed at the lower part of the LCD. Press down the key on the operation panel, corresponding digit will display “1”, and it displays “0” after releasing. Otherwise, it indicates fault occurs.

4. CNC Input/Output Signal

Signals described in this part are signals transmitted between PMC of internal CNC and CNC. They help users to understand CNC internal working state, which is not relevant to hardware circuit directly.

Signals to CNC

Diagnosis No. Significance

0	3	3	HX/RV1		*DECX		-X	+X		
			MPGX Rapid override 1		X-axis decelerateon		Manual -X	Manual +X		

0	3	4	HX/RV2		*DECZ		-Z	+Z		
			MPGX Rapid override 1		X-axis deceleration		Manual -Z	Manual +Z		

0	3	5	DRN				GR2	GR1		
			Dry run				Gear shifting signal 2	Gear shifting signal 1		

0	3	6	MLK	MP2	MP1	MP0	SBK	BDT		
			Machine lock	Handwheel incremen selection			Single block	Block skip		

0	3	7	ZRN	*SSTP	SOR	SAR	FIN	ST	STLK	MIX
			Zero return switch	Spindle stop	Spindle orientation or gearing	Spindle speed in-position	MST function finish	Cycle start	Interlock signal	X-axis

0	3	8	ERS	RT	*SP	*ESP				
			External reset	Manual rapid feed selection	Feed hold	Emergency stop				

0	3	9	PN8	PN4	PN2	PN1	KEY	MD4	MD2	MD1
			PN8~PN1：External program number selection				KEY: Program protection key/switch MD4~MD1：Mode selection			

Part Five Appendix

5. Signals from CNC

1） Diagnosis list

0	4	1	OP	SA	STL	SPL	ENB		ZPZ	ZPX
0	4	2	MA				DEN		RST	AL
0	4	3			DST		TF	SF		MF
0	4	4	M28	M24	M22	M21	M18	M14	M12	M11
0	4	5	S28	S24	S22	S21	S18	S14	S12	S11
0	4	6	S48	S44	S42	S41	S38	S34	S32	S31
0	4	9	T8	T7	T6	T5	T4	T3	T2	T1

2） Signal significance

(a) OP： In auto running

(b) SA： Servo ready

(c) STL： Cycle start

(d) SPL： Feed hold

(e) ENB： Spindle enable

(f) ZPX，ZPZ： X, Z axis zero return finish signal

(g) MA： CNC ready signal

(h) DEN： Axis move end signal

(i) RST： Reset signal

(j) AL： Alarm signal

(k) DST： MDI start

(l) TF. SF. MF： M, S, T code strobe signal

(m) No. 044: BCD code of the current M code

No. 045: Low-order 2 BCD codes of the current S code

No. 046: High-order 2 BCD codes of the current S code

No. 049: BCD code of the current T code

6. CNC Internal State Message

Current CNC working state can be viewed by No. 096 and No. 097 diagnosis messages if there is no alarm and movement during CNC auto running.

0	5	0		CSCT	CITL	COVL	CINP	CDVVL	CMTN	CFIN

When corresponding display is “1”, its significance as follows:

CFIN： M, S, T code is being executed

CMTN： Move code is being executed

CDWL：G04 (pause) code is being executed

CINP： Doing bit check

COVZ： Override is 0%

CITL： Interlock signal is ON

CSCT： Waiting for SAR signal

0	5	1				CRST			CTRD	CTPU

When corresponding display is “1”, its significance as follows:

CTPU： RS232 interface is transmitting

CTRD： RS232 interface is receiving

CTST： Emergency stop, external reset or reset key on the MDT panel is ON

0	5	2	STP	REST	EMS		RSTB			CSU

This diagnosis number is for the state when auto run stops. It is used for checking source of the trouble.

STP: Interpolation assignment stops, it is set in the following cases:

1) External reset is ON
2) Emergency stop is ON
3) Feed hold is On
4) Reset key on the MDI panel is ON
5) In Auto run, the mode is changed to Manual mode

REST： External reset key, reset key, emergency stop is ON

EMS： Emergency stop is ON

RSTB： Reset key on the MDI panel is ON

CSU： Emergency stop/ servo alarm active

0	5	3	Pulse sent to servo unit from (X) axis

0	5	4	Pulse sent to servo unit from (Z) axis

Pulses sent to servo unit are pulses from X or Z axis, which can be positive or negative.

APPENDIX IV　SPECIFICATION LIST

● Functions

Name	Specification
Controlled axes	2-axis（X. Z）
Simultaneous controlled axes	2-axis
Minimum setting unit	0.001mm
Minimum travel unit	0.001mm
Maximum command	±7 bits
Rapid traverse speed	Standard: 7.6m/min, max. speed: 30 m/min
Rapid override	F0. 25%. 50%. 100%
Feed per minute	1-8000mm/min
Maximum spindle speed	9999
Thread lead	Metric: 0.001 mm～500.000mm Inch: 0.06 teeth/inch～25400 teeth/inch
Automatic acceleration/deceleration	Available
Feedrate override	0%～150% 16-gear in total
Manual continuous feed	2-axis simultaneously
Interpolation	Linear/circular
Reference return	Available
Display	TFT color LCD 480×240 or blue LCD 320×240
Step feed	0.001mm. 0.01mm. 0.1mm
MPG	0.001mm. 0.01mm. 0.1mm
I/O interface	RS232C
Pause(s)	Available
Machine lock	All axes
Storage stroke check	Available
Exact stop	Available
Reference return when power off	Available
MDI operation	Available
Reset	Available
Dry run	Available
Single block	Available

Self diagnosis function	Available
Emergency stop	Available

- Display

Name	Specification
Display	Chinese
Time, machined workpiece number display	Available
Actual speed display	Available

- Program input

Content	Specification
Coordinate system setting	Available
Decimal point input	Available
User macro program	Available
Tool compensation	Available
Canned cycle	Available
Backlash compensation	Available
Radius R specification	Available

- M. S. T function

Name	Specification
MST function	M 2-digit
MST lock	Available

- Spindle function

Name	Specification
Spindle function	S 2-digit ＠ S 4-digit
Spindle analog output	Available
Spindle override	50%～120%

- Tool function

Name	Specification
Tool function	T01～T64
Tool compensation storage unit	±6 digits 64 groups
Tool length compensation	Available

- Edit, Operation

Name	specification
Program storage capacity	24544KB
Number of the saved programs	500
Display of the program number	Available
Sequence number search	Available
Program number search	Available
Program protection	Available

APPENDIX V WIRING FOR CNC MATCHING TAIWAN LIO SHING TOOL POST

When GSK980TA3 is used with Taiwan LIO SHING tool post, change parameter P81 to 6, P152 to 5, and P148 to 2.

XS40	23	+ 24V	+ 24V
	12	TL +	Tool post CCW magnetic valve
	13	TL -	Tool post CW magnetic valve
	6	T1	Sensor A
	5	T2	Sensor B
	4	T3	Sensor C
	3	T4	Sensor D
	19	T8	Sensor E
XS39	12	TCP	Sensor F
XS42	16	TLCOK	Tool post push-off magnetic valve
	25	+ 24V	+ 24V
	24	AGND	0V

TCP （XS39：12）	Tool post clamping signal
+24V（XS40：23）	+24V
TL+ （XS40：12）	Tool post CCW signal
TL- （XS40：13）	Tool post CW signal
T1 （XS40：6）	Tool detection sensor signal A
T2 （XS40：5）	Tool detection sensor signal B
T3 （XS40：4）	Tool detection sensor signal C
T4 （XS40：3）	Tool detection sensor signal D
T8 （XS40：19）	Tool pan stop and clamp detection signal E
TLOCK（XS42：16）	Tool post push-off signal
+24V（XS42：25）	+24V
AGND（XS42：24）	0V

APPENDIX VI　FACTORY PARAMETER

GSK980TA3 factory parameters:

Parameter No.	A-disk (servo)	B-disk (step)	C and D disks (user -defined)
1	00000000	00000000	
2	00000011	00000011	
3	00000100	00000100	
4	00110000	00110000	
5	01000011	01000011	
6	01000000	01000000	
7	00001000	00001000	
8	11000011	11000011	
9	01001100	01001100	
10	00000000	00000000	
11	00000001	00000001	
12	00000000	00000000	
13	00000001	00000001	
14	10001111	10001111	
15	00000000	00000000	
16	00000000	00000000	
17	1	1	
18	1	1	
19	1	1	
20	1	1	
21	4000	3000	
22	8000	6000	
23	60	200	
24	60	200	
25	60	60	
26	60	60	
27	8000	8000	
28	100	100	
29	60	200	
30	60	200	

31	60	200	
32	100	100	
33	80	80	
34	40	40	
35	40	40	
36	0	0	
37	0	0	
38	1	1	
39	0	0	
40	0	0	
41	1000	1000	
42	1000	1000	
43	200	200	
44	0	0	
45	0	0	
46	9999999	9999999	
47	-9999999	-9999999	
48	9999999	9999999	
49	-9999999	-9999999	
50	10	10	
51	999	999	
52	1024	1024	
53	1	1	
54	1	1	
55	9999	9999	
56	9999	9999	
57	9999	9999	
58	9999	9999	
59	95	95	
60	9999	9999	
61	0	0	
62	1	20	
63	50	50	
64	50	50	
65	500	500	
66	0	0	

67	100	100	
68	10	10	
69	60	200	
70	60	200	
71	0	0	
72	0	0	
73	0	0	
74	0	0	
75	0	0	
76	0	0	
77	0	0	
78	0	0	
79	0	0	
80	0	0	
81	4	4	
82	30	30	
83	1000	1000	
84	15000	15000	
85	200	200	
86	1	1	
87	1	1	
88	500	500	
89	100	100	
90	100	100	
91	100	100	
92	100	100	
93	100	100	
94	100	100	
95	9600	9600	
96	200	600	
97	2	2	
98	2000	2000	
99	0	0	
100	0	0	
101	12	-45	
102	30	30	

103	40	40	
104	45	120	
105	0	0	
106	0	0	
107	1000	1000	
108	1000	1000	
109	0	0	
110	0	0	
111	0	0	
112	0	0	
113	0	0	
114	0	0	
115	0	0	
116	0	0	
117	0	0	
118	0	0	
119	0	0	
120	0	0	
121	0	0	
122	0	0	
123	0	0	
124	0	0	
125	0	0	
126	0	0	
127	0	0	
128	0	0	
129	0	0	
130	0	0	
131	0	0	
132	0	0	
133	0	0	
134	0	0	
135	0	0	
136	0	0	
137	0	0	
138	0	0	

139	0	0	
140	0	0	
141	0	0	
142	0	0	
143	0	0	
144	0	0	
145	0	0	
146	151	151	
147	0	0	
148	0	0	
149	0	0	
150	0	0	
151	2	2	
152	0	0	
153	0	0	
154	0	0	
155	0	0	
156	0	0	
157	40	200	
158	7600	6000	
159	0	0	
160	0	0	