ADT-CNC4240

Milling Machine Control System

User Manual

Adtech (Shenzhen) CNC Technology Co., Ltd.

Add: 5th floor， 27-29th Bulding,Tianxia IC Industrial Park,MaJiaLong,Yiyuan Road, Nanshan District,Shenzhen City, China.C:518052

TEL:+86-755-2609 9116 2672 2719 FAX:+86-755-2672 2718

E-mail:export@adtechen.com

Http://www.adtechen.com

Copyright Notice

The property rights of all the parts of the manual belong to Adtech (Shenzhen) CNC Technology Co., Ltd. (Adtech for short), and any form of imitation, copying, transcription or translation by any company or individual without the permission is prohibited. This manual does not include any form of assurance, standpoint expression, or other intimations. Adtech and the stuffs have no responsibility for any direct or indirect disclosure of the information, benefit loss or business termination of this manual of the quoted product information. In addition, the product and the information mentioned in this manual are for reference only, and the content is subjected to change without notice.

ALL RIGHTS RESERVED!

Adtech (Shenzhen) CNC Technology Co., Ltd

Version Upgrading Instruction

Procedures	Version Number	Modification Date	Instruction
XT-20061224	2.0	2009/4/22	The Fourth Version

Remarks: the meanings of the four numbers in the version number are as follows:

Procedures for the project/Bank Main Version Number/ Bank Secondary Version Number/ Reservation

Notes: the above version table only refers to the version updating of the modification of the instruction.

Contents

1 Summary of Specification

ADT-CNC4240 is a standard controlling system for milling machines characterized by the economic costs, employs the standard G codes for programming and is widely used in the automatic equipment with length control in the products. The general specification and the maintenance of this product are described as follows:

1.1 Production Specifications

Function	Name		Specification
Controllable axes	Controlled axis		4axis (X，Y，Z，A)
	Simultneous controllable axes number		4 axes linear interpolation 2 axes arc interpolation
Input command	Min setting unit		0.001mm
	Min move unit		0.001mm
	Max instruction value		±9999.999 mm
Feed	fast feedrate		X-axis、Y-axis、Z-axis、A-axis:9999mm/min（max）
	range	feed per minute	1～9999 mm/min
		feed per rotate	1～500 mm/ratio
	Auto acc and dec speed		Yes
	feed speed rate		10～150%
Hand	Hand continuous feeding		Yes
	Reference point for manual return		one or three axes return to return to reference point simultaneously
	single step /handwheel function		Yes
Interpolation	Location,Linear,Full cycle arc		G00,G01,G02/G03
Operation mode	MDI,automation,manual,single step,edit		Yes
Commissioning	Trial running,single program,hand		Yes

function	wheel	
Coordinate system and pause	Pause(sec/microsecond)	G04 X/P_
	coordinate system setting	G92
	Auto coordinate system setting	Yes
safety function	software & hardware limit check	Yes
	sudden stop	Yes
Memory	program storage capacity and quantity	Total capacity: 256M bytes; 9999 working areas; No processing document limit
Program edit	program edit	Insert,modification,delete,cancel
	program number,sequence,address, Character retrieving	Yes
	decimal point programming	Yes
Display	320×240lattice 5.7inch LCD	
	Position screen/program edition Cutter compensation/alarm display Handwheel adjusting/diagnosis screen Parameter setting/image emulation	Yes
M，S，T function	assistant function	M Code
	spindle function	S0-S15 (level control) S15-S99999（analog）
	Tool function	T Code
Compensation Function	Memory for cutter compensation	18 sets of cutter length, radius compensation.
	Reverse gap compensation	Yes
Others function	Auto halving	Yes

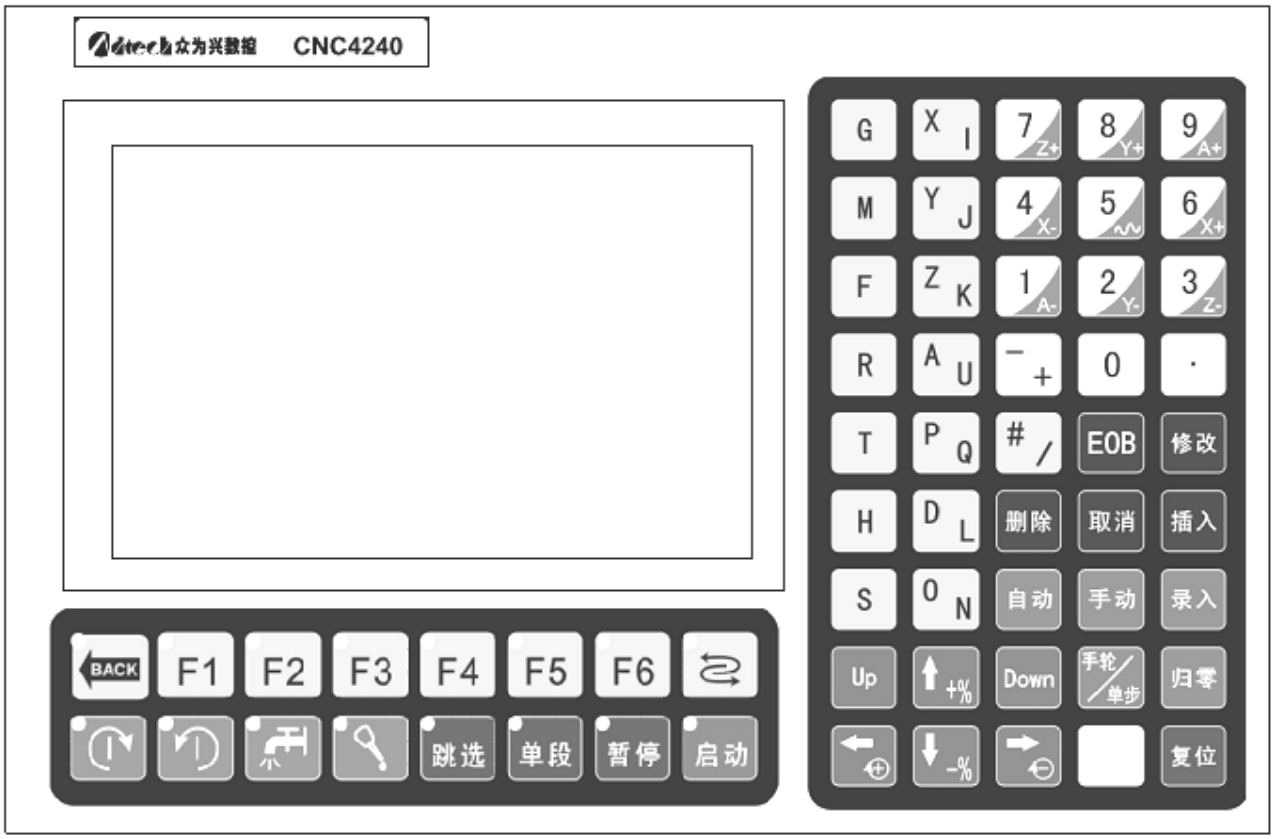
	Auto cutter calibrator	
	Designating arc radius R/central position	Yes
	Electronic gear ratio	Yes

1.2 Working environments

Working voltage	24V DC（with filter）
Working temperature	0℃— 45℃
Best working temperature	5℃— 40℃
Working humidity	10％——90％no condensation
Best working humidity	20％——85％
Tempering storage	0℃—50℃
Humidity storage	10％——90％

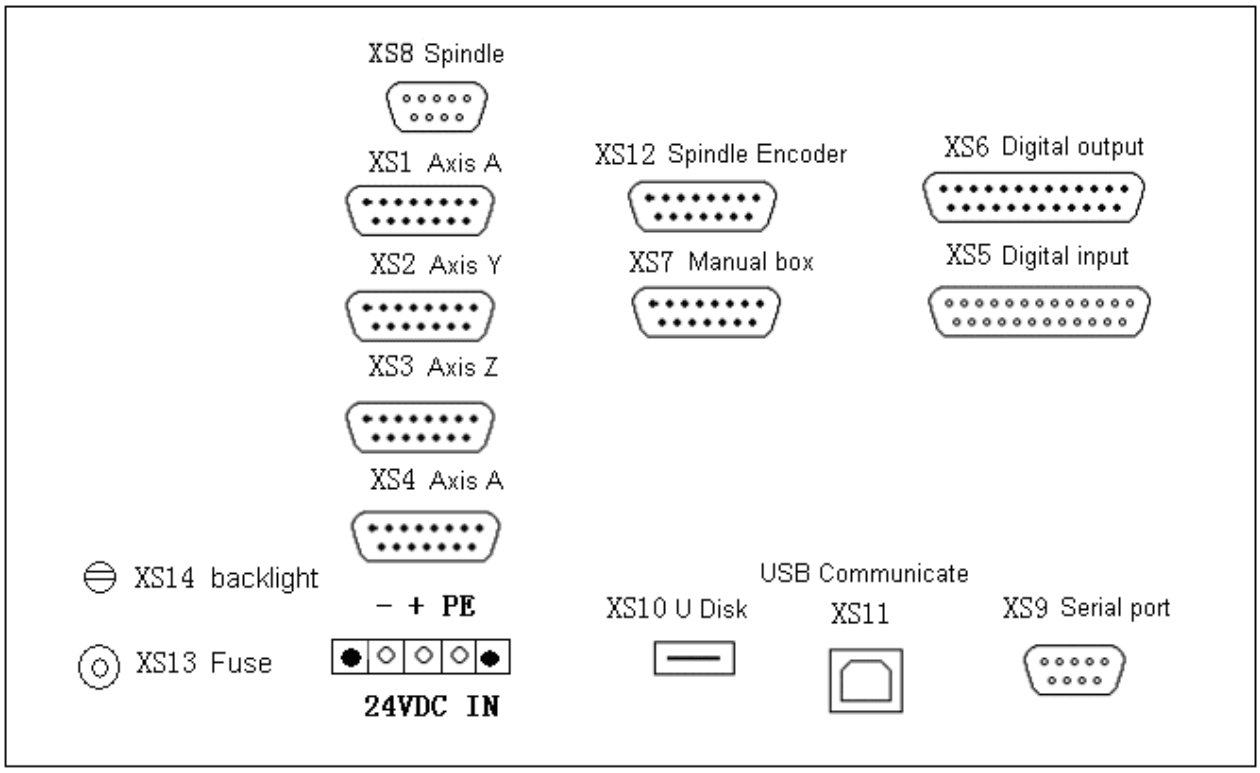
2 Hardware Interface Definition and Descriptions of Connection

2.1 Operation panel



2.2 The layout of the installation

2.2.1External interface drawing



1.X-axis、Y-axis、Z-axis、A-axis:
D type 15-core receptacle: connect stepper motor driver or AC digital servo driver.

2.XS5 Digital Input:
D type 25-core receptacle: shaft limitation and input signals of other switching value.

3.XS6 Digital Output:
D type 25-core receptacle: Output signal of switching value.

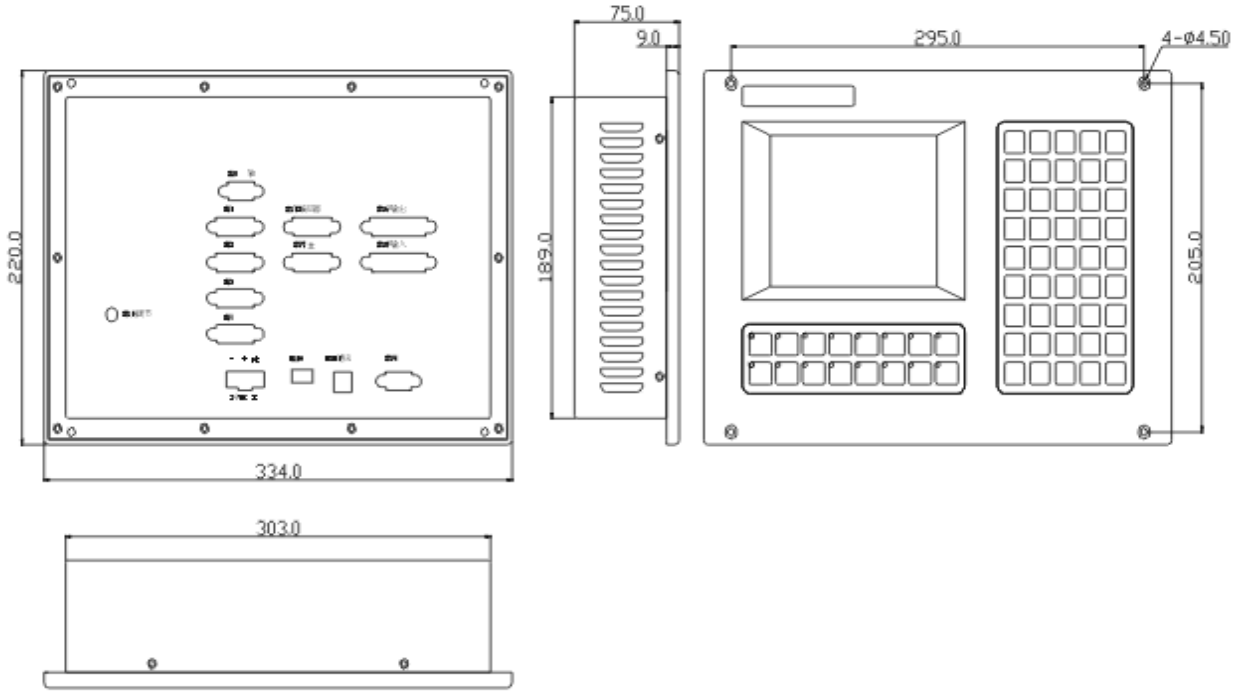
4.USB and serial interface: For file exchange between PC and CNC4240 controller and for realizing other functions.

5.CNC4240 Controller: Using DC 24V, with power consumption of 5W.

6.XS7 Additional panel:
D type 15-core receptacle: connect handwheel.

7.XS8 Spindle:
D type 9-core receptacle: connect spindle transducer.

2.2.2Plans to install size



2.2.3 Notes installation

Installation conditions:

- The distribution cabinet must be dust proof, cooling liquid proof and organic solvent proof.
- In designing the distribution cabinet, a distance of not less than 20cm must be kept between the rear cover of the system and the machine box. It must be taken into consideration that the temperature difference between inside and outside of the cabinet shall not be more than 10°C when the temperature inside the cabinet rises.
- A fan shall be installed for the distribution cabinet so as to ensure the good ventilation inside.
- The display panel shall be installed to a position which can't be spilled by the coolant.
- In designing the distribution cabinet, it must be taken into consideration that the external interference be lowered down as much as possible and interference be prevented to be sent to the system.
- Method to prevent interference：

In designing the systems, anti-interference measures such as shielding spatial EM radiation, absorbing dash current and filtering clutter wave of power have been taken, which can prevent external interferences to affect the system itself to some extent. To ensure the stable running of the system, the following measures must be taken in installing and connecting the system:

1. Keep CNC far from the equipment that can produce interferences (such as the frequency converter, AC contactor, static generator, HV generator and section devices of power line). At the same time, the switching power supply shall be separately connected to the filter so as to enhance the anti-interference capacity of CNC (see Figure 1-4).

2. The power supply to system shall be provided via the isolated transformer. The machine tool of the system must be grounded. CNC and the driver must be grounded via separate grounding wires.

- Method to constrain the interference:

To restrain the interference, the RC return circuit (0.01μF，100～200Ω,figure 1-5) should be connected at the two ends of the AC coil in a parallel manner, and this RC return circuit should be installed to the position as close as possible to the inductive load (figure1-6); the freewheeling diode should be reversely connected to the two ends of the DC winding in a parallel manner; the surge absorber should be installed at the winding terminal of the AC motor.(figure1-7)

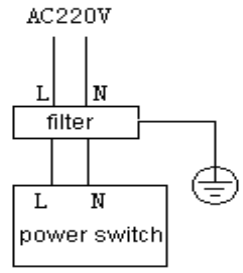


Figure 1-4

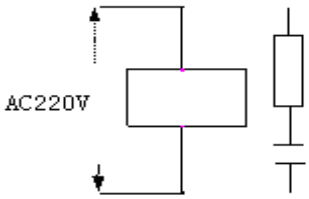


Figure 1-5

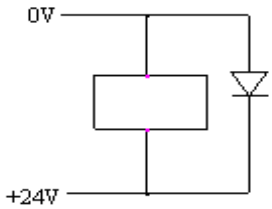


Figure 1-6

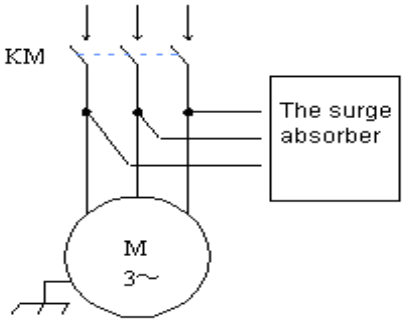


Figure 1-7

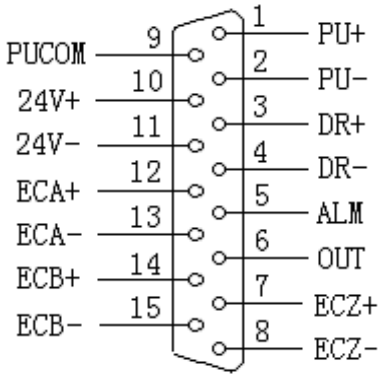
➢ To reduce the mutual interference between CNC signal cable and high-voltage cable, the following principles must be observed in wiring:

Set	Cable type	Cabling requirements
A	AC power line	Bind the cables of Group A to Group B and C separately. The further Group B is from Group C, the better. Or, cables of Group A can be shielded to avoid EM interference.
	Ac coil	
	Ac contactor	
B	Ac coil（24VDC）	Group B and A should be bounded separately or Group B be shielded. The further Group B is from Group C, the better.
	DC Relay（24VDC）	
	For cables between the System and high-voltage distribution cabinet,	
	For cables between the System and milling machine.	
C	For cables between the System and Servo motor driver.	Group C and A should be bounded separately or Group C be shielded. A distance of at least 10cm should be kept between Group C and B and twisted-pair cables be used.
	position command cable	
	cable for cable enconder	
	Handwheel cable	
	Other shielded cables.	

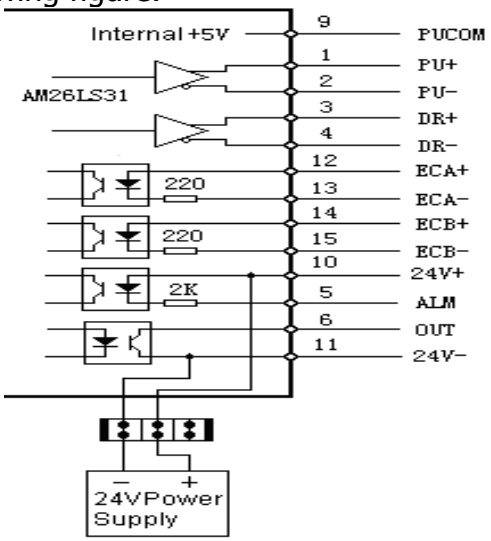
2.3 Interface definition

2.3.1Motor&driver control interface（XS1..XS4）

There are four (XS1 X-axis、XS2 Y-axis、XS3 Z-axis、XS4 A-axis) ports for the driver, whose definitions are identical. See the following figure.



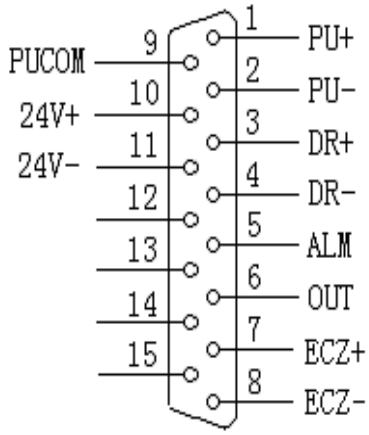
CNC4240 Pulse wiring



Internal Electric Diagram for Pulse Output.

Line No.	Definition	Function
1	PU+	pulse signal+
2	PU-	pulse signal-
3	DR+	direction signal+
4	DR-	direction signal-
5	ALM	Servo alarm signal input X-axis: IN34、Y-axis: IN35 、Z-axis: IN36、A-axis: IN37
6	OUT	Servo signal output X-axis:OUT24 Y-axis:OUT25 Z-axis:OUT26 A-axis:OUT27
7	ECZ+	Encoder Z-phase input+
8	ECZ-	Encoder Z-phase input-
9	PUCOM	used for single-end input driver.
10	24V+	The internally provided 24V power supply has already been connected to 24V terminal of the controller.
11	24V-	
12	ECA+	Encoder A-phase input+
13	ECA-	Encoder A-phase input-
14	ECB+	Encoder B-phase input+
15	ECB-	Encoder B-phase input-

- Standard cable of Pulse wiring diagram

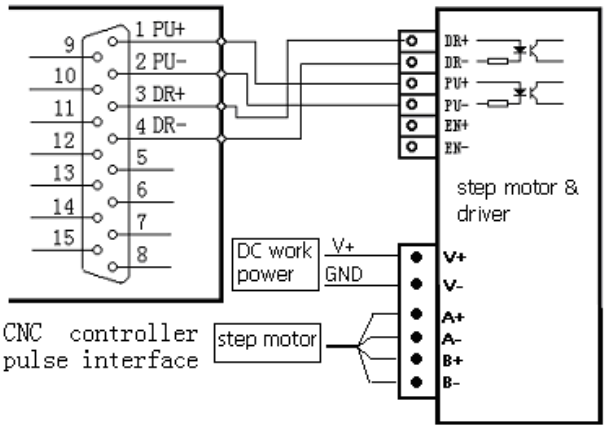


XS1..XS4pulse interface standard wiring

The standard wirings is suitable for CNC4340, CNC4240 and CNC4342 controller.

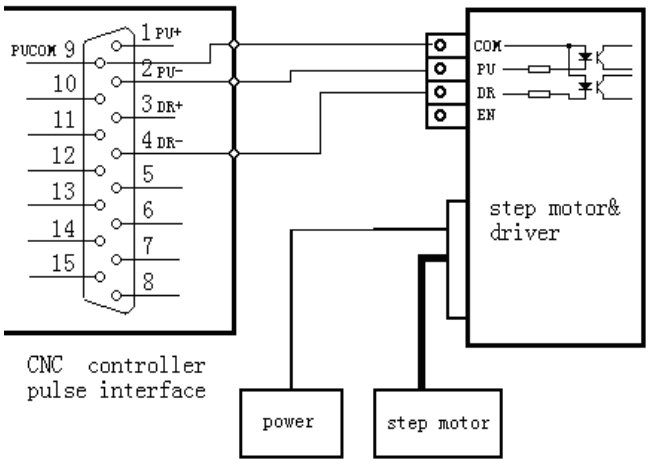
- Wiring to the driver of stepper motor with differential input

The ADTECH CNC driver should be used as the reference. As all ADTECH CNC drivers employ the differential input mode, which features its high anti-interference performance, it is recommended this mode be used. The wiring between CNC and the driver of stepping motor and the stepping motor is shown in the following figure.



- Wiring Diagram to the driver of stepper motor with single-end input

In the stepping drivers made by some companies, the cathodes of optical coupler are connected together, called co-cathode wiring method. However, this method is not suitable for CNC controller. The anodes of optical coupler can be connected together, called co-anode wiring method. To that effect, the following wiring diagram should be referred, in which PU+ and DR+ are not connected together. Otherwise, the pulse interface may be damaged.



Wiring diagram to the driver of stepper motor with common anode input

- Connect to servo motor & driver diagram

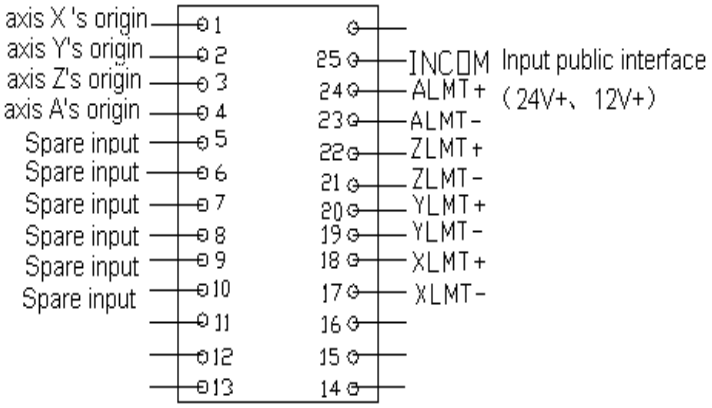
As the differential wiring method is used in most cases, this method can be referred for the pulse section. For many servo drivers that need the 12-24V power supply, the 24V power supply provided by Pin 10 and 11 can be used. The actual wiring is subject to the model of the servo driver. If you are not sure about the wiring, please contact ADTECH without hesitation.

Note: Any two pins of PU+, PU-, DR+ and DR- cannot be connected together directly, otherwise, it may damage the pulse interface.

2.3.2Digital input interface（XS5）

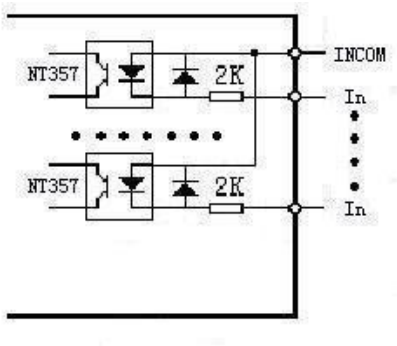
The numeric input port includes the limit signal of the hardware for each shaft. The definition is shown as follows:

XS5 Digital Input Interface Wiring Diagram



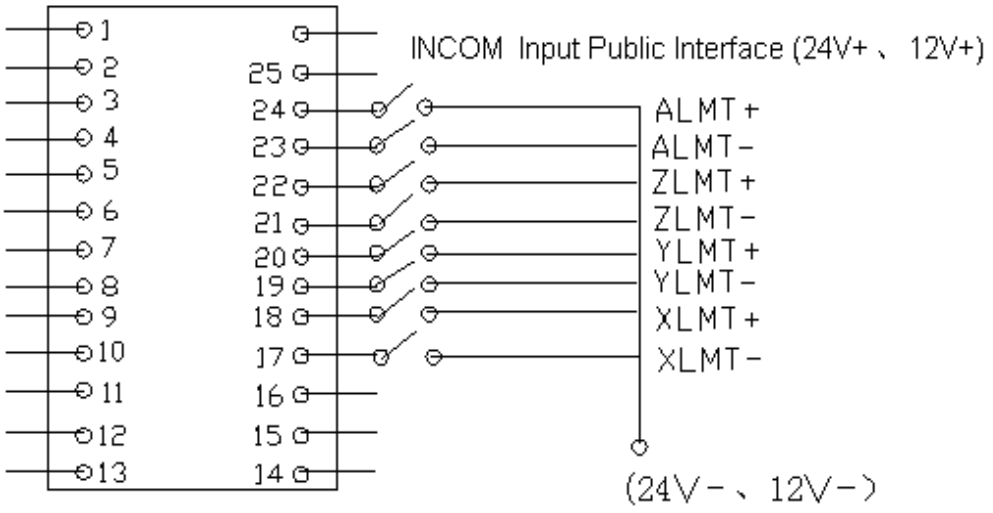
Line no	Interrupt No.	Function
1	IN0	X-axis zero

2	IN1	Y-axis zero
3	IN2	Z-axis zero
4	IN3	A-axis zero
5	IN4	Cutter calibrator position check
6	IN5	Safe signal check input
7	IN6	System voltage alarm input
8	IN7	spare input
9	IN8	spare input
10	IN9	spare input
11	IN10	System feed alarm input
12	IN11	spare input
13	IN12	spare input
14	IN13	spare input
15	IN14	spare input
16	IN15	spare input
17	IN16(XLMT-)	X-axis negative limit(standby IN32)
18	IN17(XLMT+)	X-axis positive limit(standby IN33)
19	IN18(YLMT-)	Y-axis negative limit(standby IN34)
20	IN19(YLMT+)	Y-axis positive limit(standby IN35)
21	IN20(ZLMT-)	Z-axis negative limit(standby IN36)
22	IN21(ZLMT+)	Z-axis positive limit(standby IN37)
23	IN22(ALMT-)	Z-axis positive limit(standby IN37)
24	IN23(ALMT+)	A-axis positive limit(standby IN39)
25	INCOM	INCOM(24+、12V+)Input public interface access provided by internal or external power supply

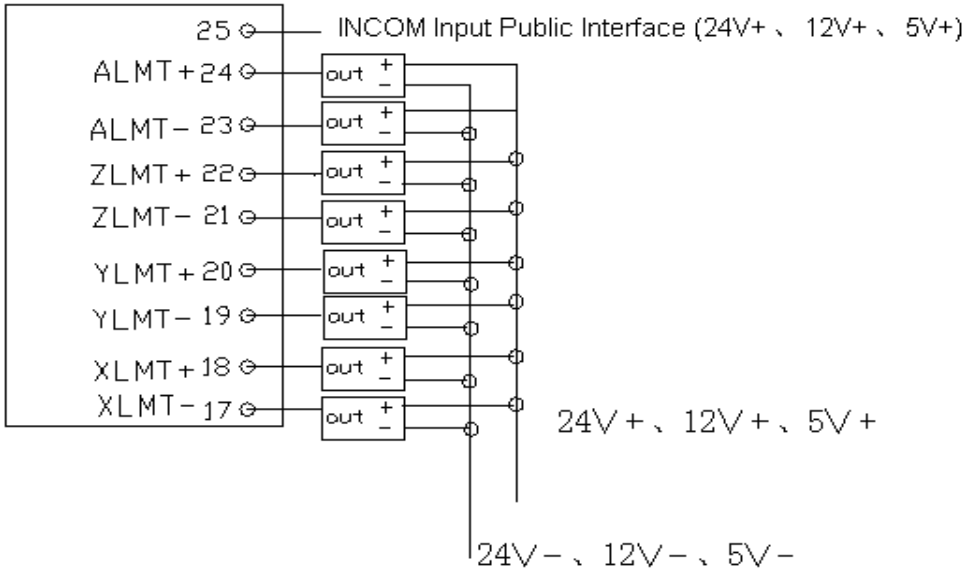


The digital input concise internal circuit

Mechanical Switch Wiring Diagram



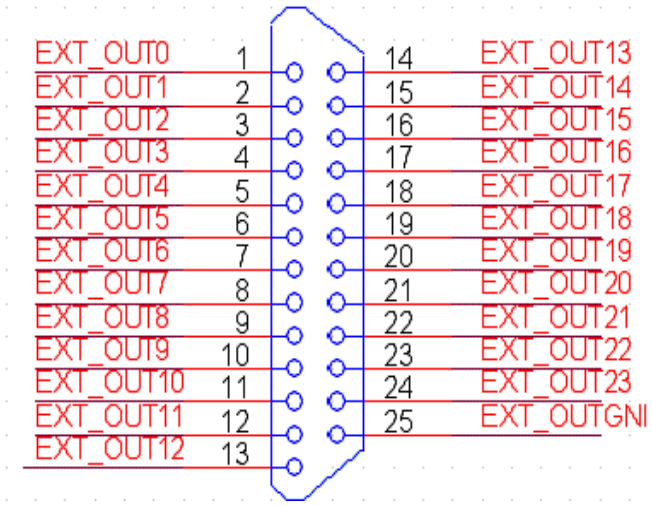
Photoelectric Switch Wiring Diagram



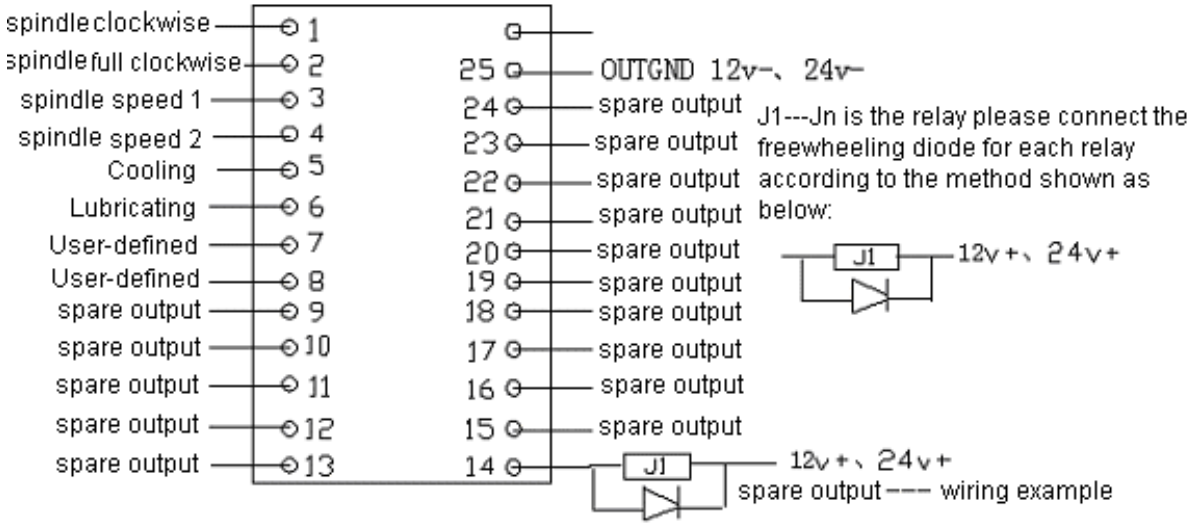
+Terminal is for the anode of power supply of the approaching switch, -Terminal is for the grounding wire of the approaching switch and the OUT terminal is for the output signal. For regular approaching switches, the operating voltage should be 10-30V, with NPN output. The photoelectric switch is also applicable.

2.3.3Digital Output Interface（XS6）

The digital output interface,.wiring definition is shown as follows:

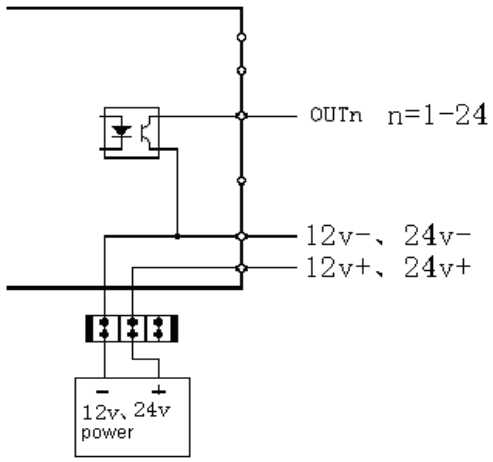


XS6 Output Interface Wiring Diagram

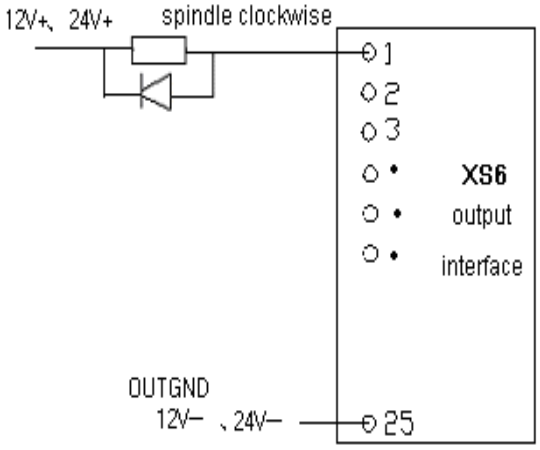


Line No.	Definition	Function
1	OUT0	spindle clockwise (M03)
2	OUT1	spindle full clockwise (M04)
3	OUT2	spare output (M56、M57)
4	OUT3	Output spare (M58、M59)
5	OUT4	cooling (M08、M09)
6	OUT5	lubricating (M32、M33)
7	OUT6	Output spare (M10、M11)
8	OUT7	System timing oil pump

9	OUT8	Output spare (M12、M13)
10	OUT9	Output spare (M14、M15)
11	OUT10	Output spare (M16、M17)
12	OUT11	Output spare (M18、M19)
13	OUT12	Output spare (M40、M41)
14	OUT13	Output spare (M42、M43)
15	OUT14	Output spare (M44、M45)
16	OUT15	Output spare (M46、M47)
17	OUT16	Output spare (M48、M49)
18	OUT17	Output spare (M50、M51)
19	OUT18	warning lights
20	OUT19	running lights
21	OUT20	Frequency-converting segment rate switch 3（M66、M67)
22	OUT21	Frequency-converting segment rate switch 32（M64、M65)
23	OUT22	Frequency-converting segment rate switch 31（M62、M63)
24	OUT23	Frequency-converting segment rate switch 30（M60、M61)
25		OUTGND12V-、24V- External output of public power

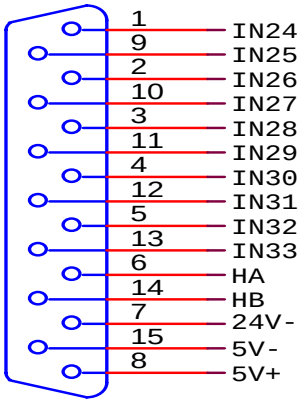


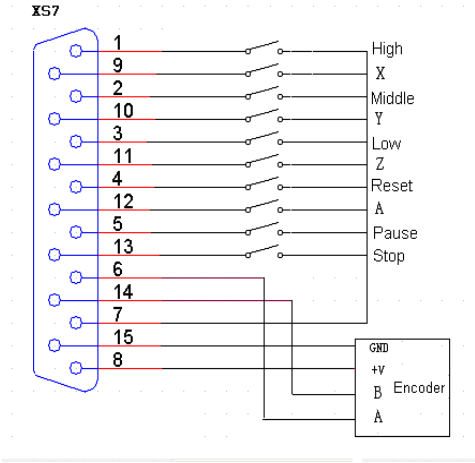
Concise internal circuit(left)



Wiring diagram of machine(right)(take spindle on CW)

2.3.4Manual Control Box Interface（XS7）

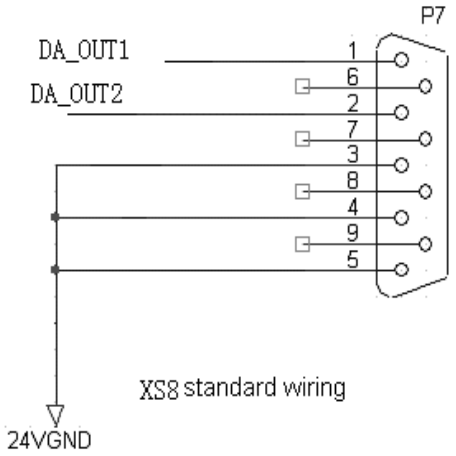




Line NO.	Definite	**Function**
1	(IN24) Stall switch	0.1 stall--- High-speed
2	(IN26) Stall switch	0.01 stall--- Middle-speed
3	(IN28) Stall switch	0.001 stall--- Low-speed
4	(IN30) button	Reset circulation
5	(IN32) button	Pause
7	24V-	24V provided by the internal negative power supply
9	(IN25) axis select	X-axis
10	(IN27) axis select	Y-axis
11	(IN29) axis select	Z-axis
12	(IN31) axis select	A-axis
13	(IN33)button	Stop
6	HA	Hand encoder A phase signal input
14	HB	Hand encoder B phase signal input
15	5V-	Negative pole of internal 5V power supply
8	+5V	Positive pole of internal 5V power supply
7	24V-	Negative pole of internal 24V power supply

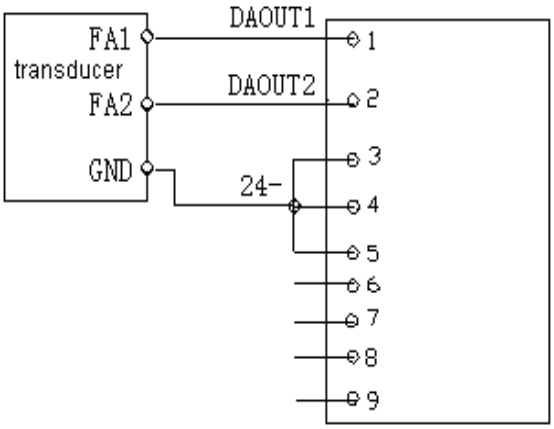
2.3.5Analog output interface（XS8）

The standard diagram of Analog output interface connection:



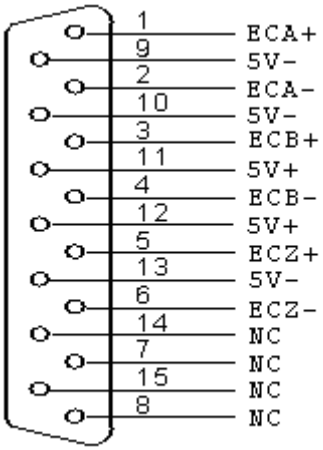
The standard wirings is suitable for XS8 interface of CNC4340,CNC4240 and CNC4342 controller.

Wiring diagram of Analog spindle XS8 and Transducer

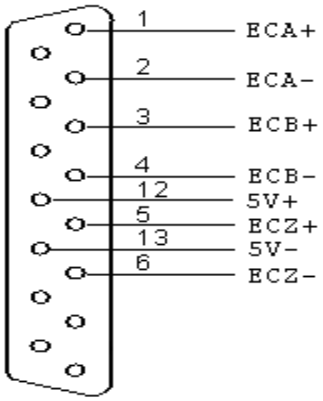


Line No.	Definition	Function
1	DAOUT1	Analog voltage output（0～10）V
2	DAOUT2	Analog voltage output（0～10）V
3	GND	GND supply provided internally 24V
4	GND	GND supply provided internally 24V
5	GND	GND supply provided internally 24V

2.3.6Interface of Spindle Encoder（XS12）



The standard wiring diagram of Spindle encoder:



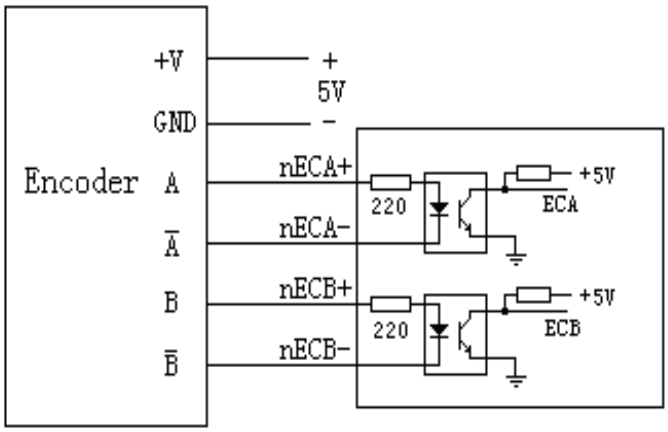
The standard wirings of Spindle encoder is suitable for CNC4240 and CNC4342 controller.

Line No.	Definition	Function
1	ECA+	Encoder A phase input+
2	ECA-	Encoder A phase input-
3	ECB+	Encoder B phase input+
4	ECB-	Encoder B phase input-
5	ECZ+	Encoder Z phase input+(standby)
6	ECZ-	Encoder Z phase input-(standby)
7	NC	Non
8	NC	Non
9	5V-	Negative pole of internal 5V power supply, cannot connect to external power supply
10	5V-	Negative pole of internal 5V power supply, cannot connect to external power supply
11	5V+	Positive pole of internal 5V power supply, cannot connect to

		external power supply
12	5V+	Positive pole of internal 5V power supply, cannot connect to external power supply
13	5V-	Negative pole of internal 5V power supply, cannot connect to external power supply
14	NC	Non
15	NC	Non

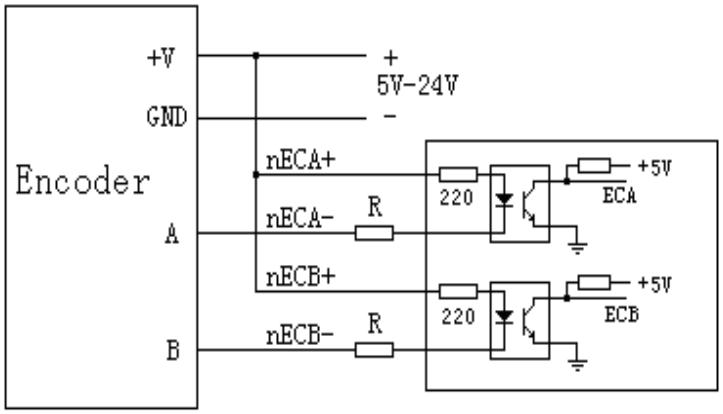
- AB-phase decoding input has differential connection and common anode connection, depending on the type of the encoder.
- Encoder output has the open collector output, complementation output, voltage output and long-line driver output generally. It can use the common anode connection for the open collector output, complementation output and voltage output, and use the differential connection for the long-line driver output.
- As shown in the following figure, AB-phase decoding input signal uses the differential connection; if use the common anode connection, it needs to connect the positive pole of A-phase with the positive pole of B-phase together; if use the common cathode connection, it needs to connect the negative pole of A-phase with the negative pole of B-phase together.

Differential Connection (see as below):



5V power supply is provided externally.

Common Anode Connection (see as below):

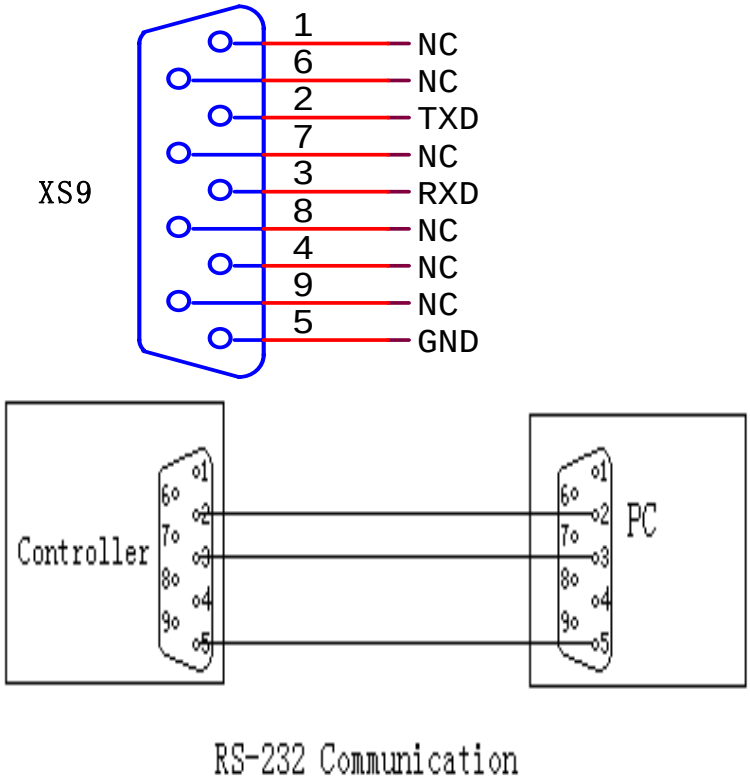


The voltage of the power supply depends on the encoder, when using 5V power supply, the resistance R is not required; when using 12V power supply, it can use 1K-2K resistance for R; when using 24V power supply, it can use 2K-5K resistance for R.

It is suggested that use the encoder with the long-line driver output, as it uses the differential connection, the anti-interference performance will be better when the line is long.

2.3.7RS232 Transmission interface（XS9）

Serial Communication Interface -9-Chip Signal Socket (male)



RS-232 Communication

line No	Definition	Function
1	NC	Non
2	TXD	Send Data
3	RXD	Receive Data
4	NC	Non
5	GND	GND
6	NC	Non
7	NC	Non
8	NC	Non
9	NC	Non

2.3.8 USB Memory interface to connect（XS10）

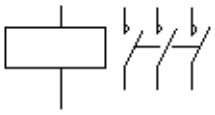
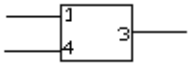
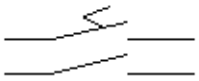
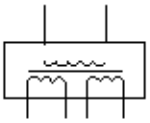
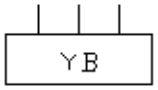
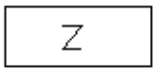
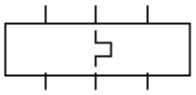
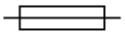
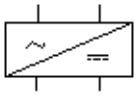
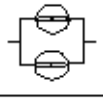
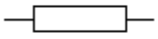
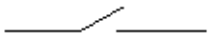
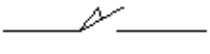
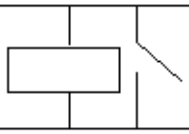
Standard USB memory interface（example of U disk）;

2.3.9 PC USB Communication interface（XS11）

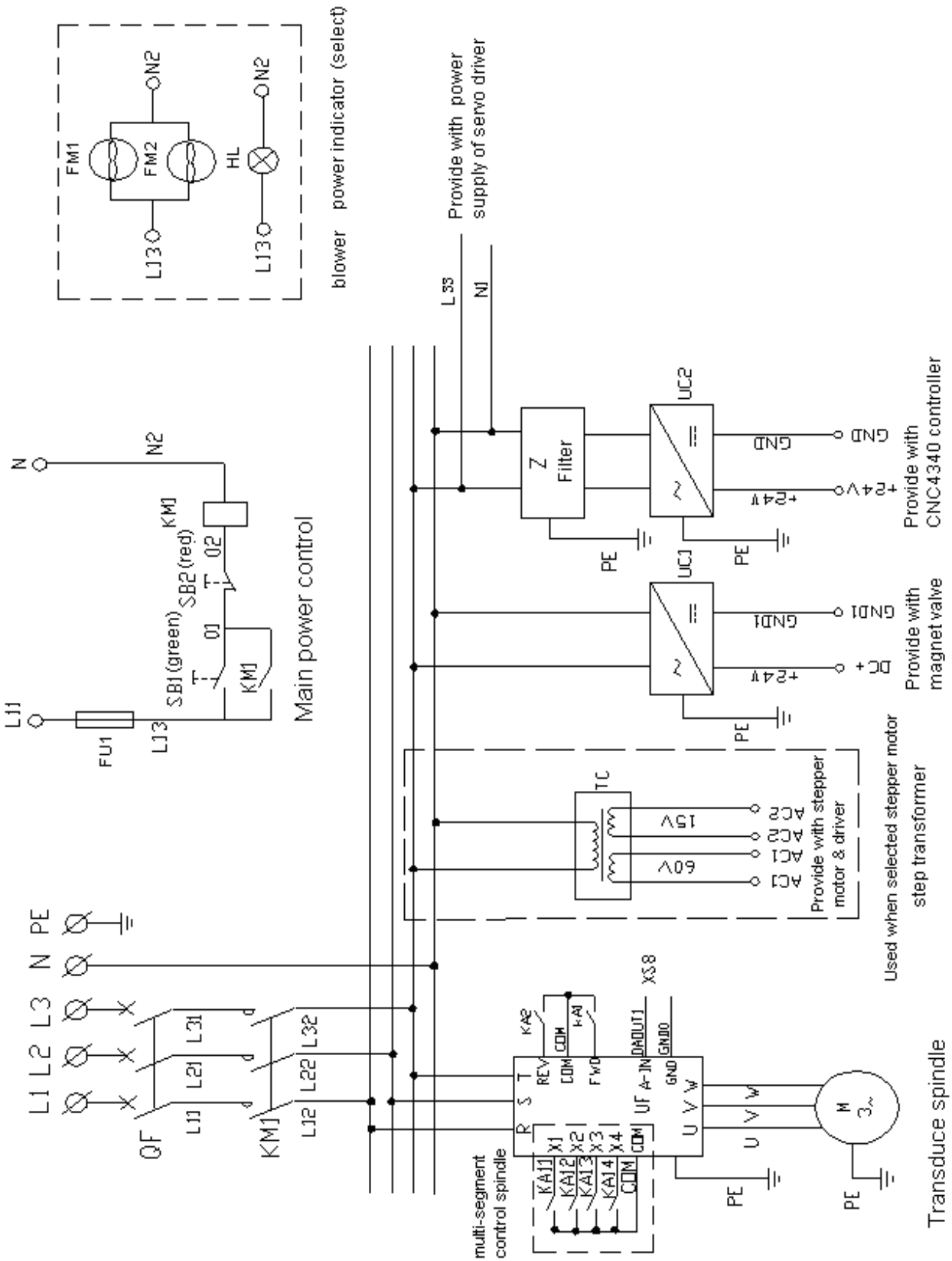
Standard USB communication interface;

2.4 Electric Connection Drawing

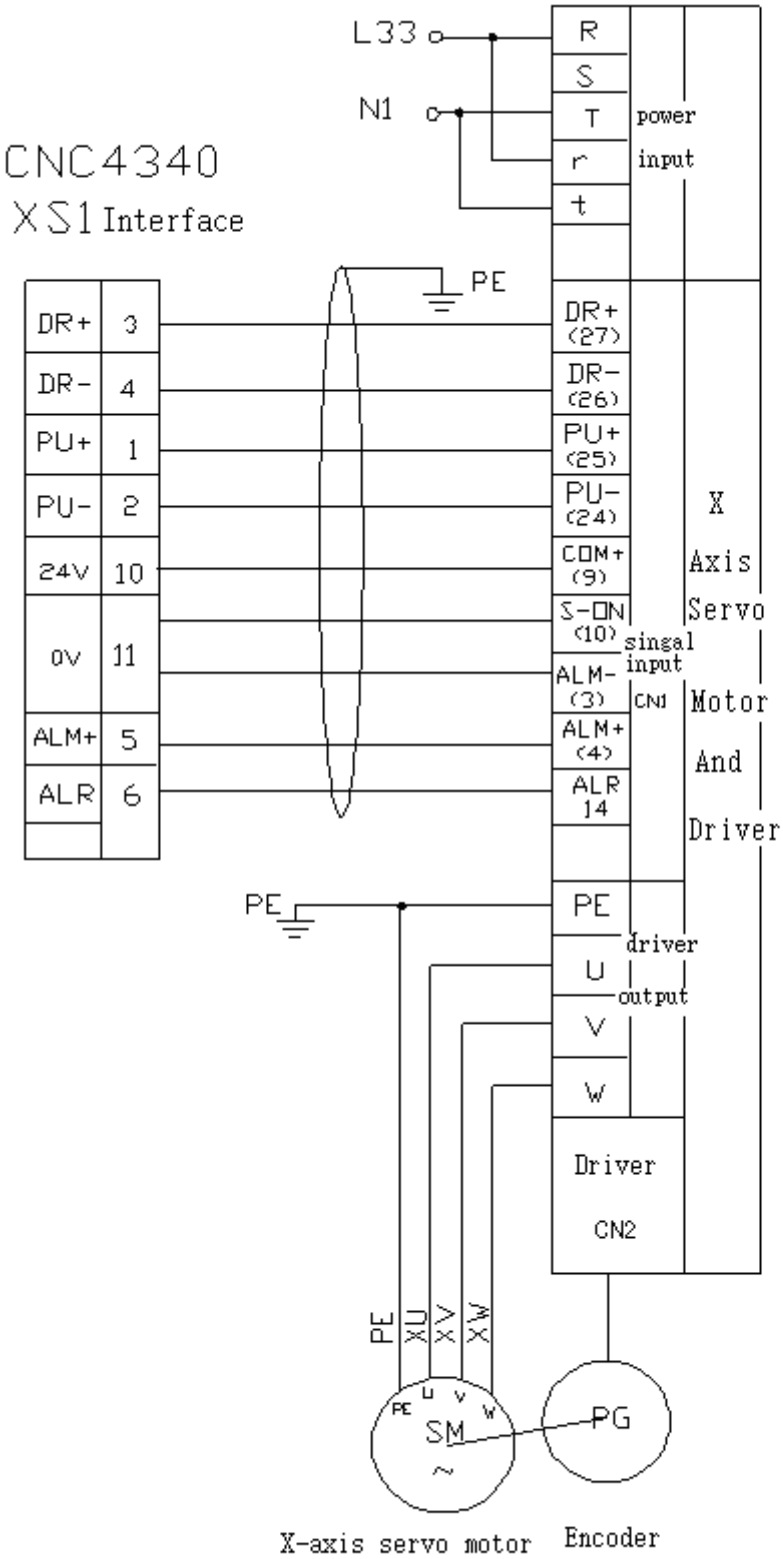
2.4.1 Schematic symbol

Symbol	Name	Subtype	Symbol	Name	Subtype
QF	Breaker		SM	Servo Motor	
KM	Contactor		M	Stepping Motor	
UF	Transducer		SQ	Proximity Switch	
M	Motor		SA	Foot Switch	
TC	Transformer		YB	Motor Brake	
Z	Filter		FR	Hot Relay	
FU	Cutout		UC	Switching Power	
SB	Button		YV	Magnet Valve	
FM	Blower		C	Capacitor	
HL	Indicator Light		R	Resistance	
QS	Tact Switch		QS	Limit Switch	
PG	Encoder		KA	Relay	

2.4.2 Power plans to connect

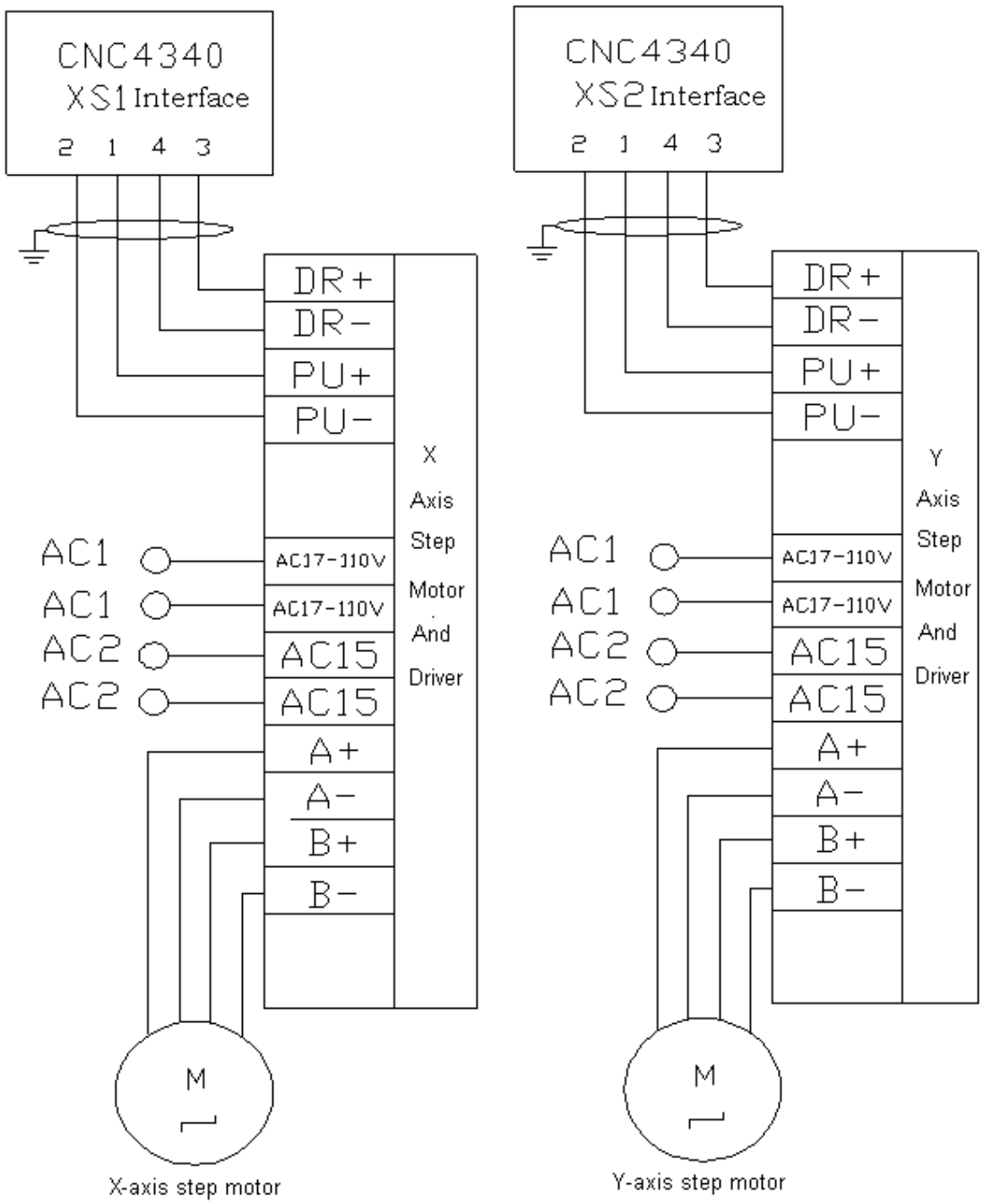


2.4.3 Servo Driver Connection Diagram



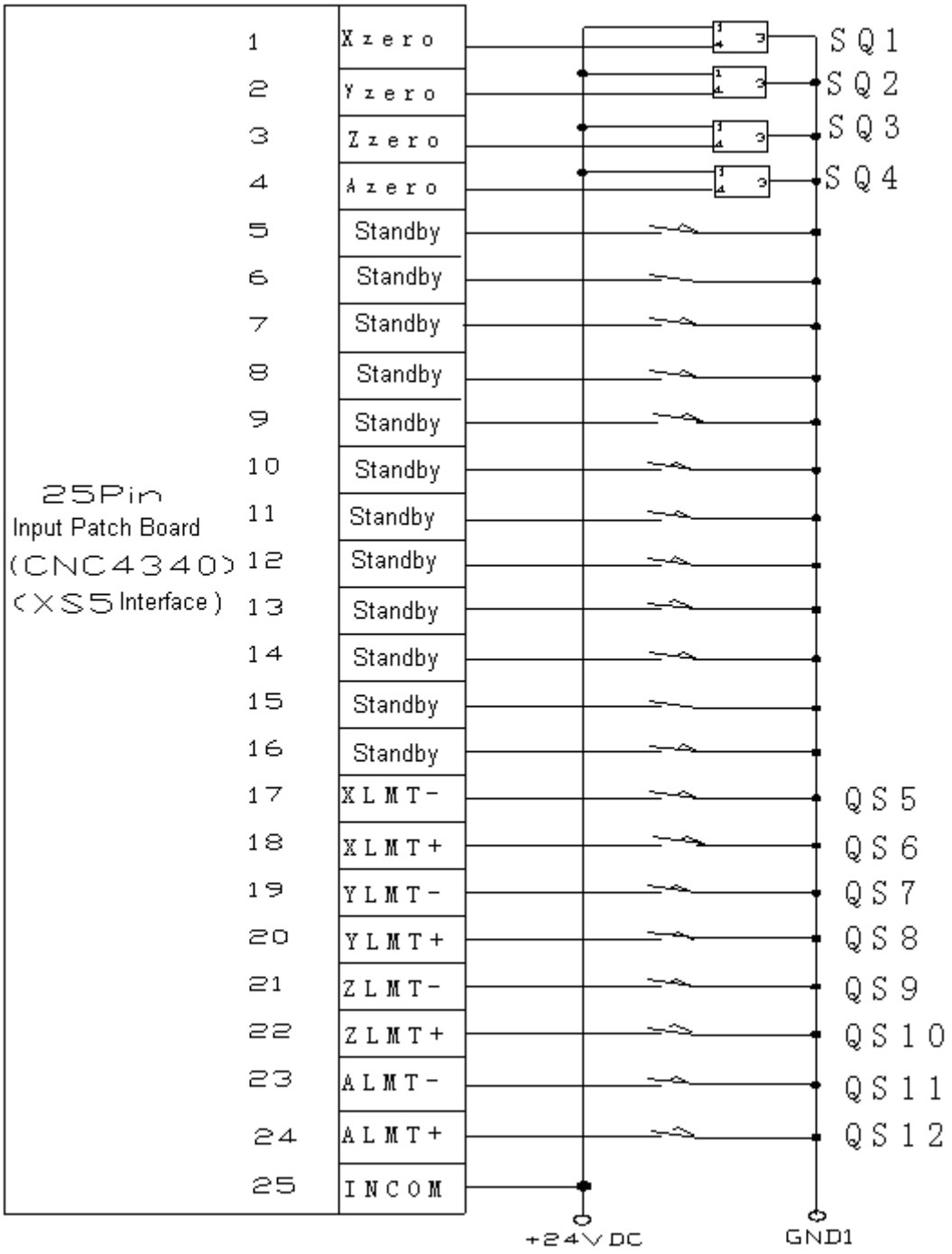
Select and use servo connection

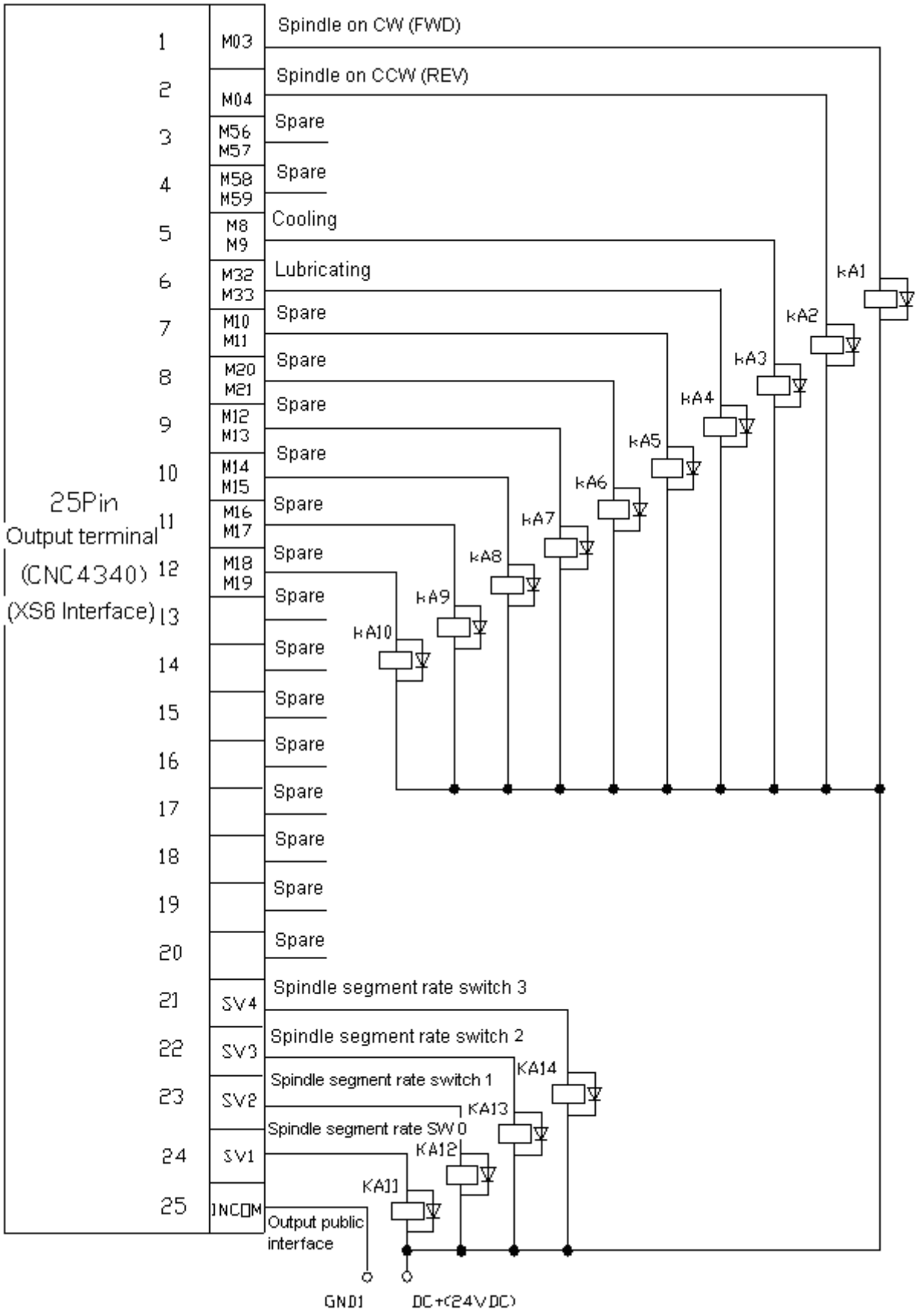
2.4.4 Stepper Connection Diagram

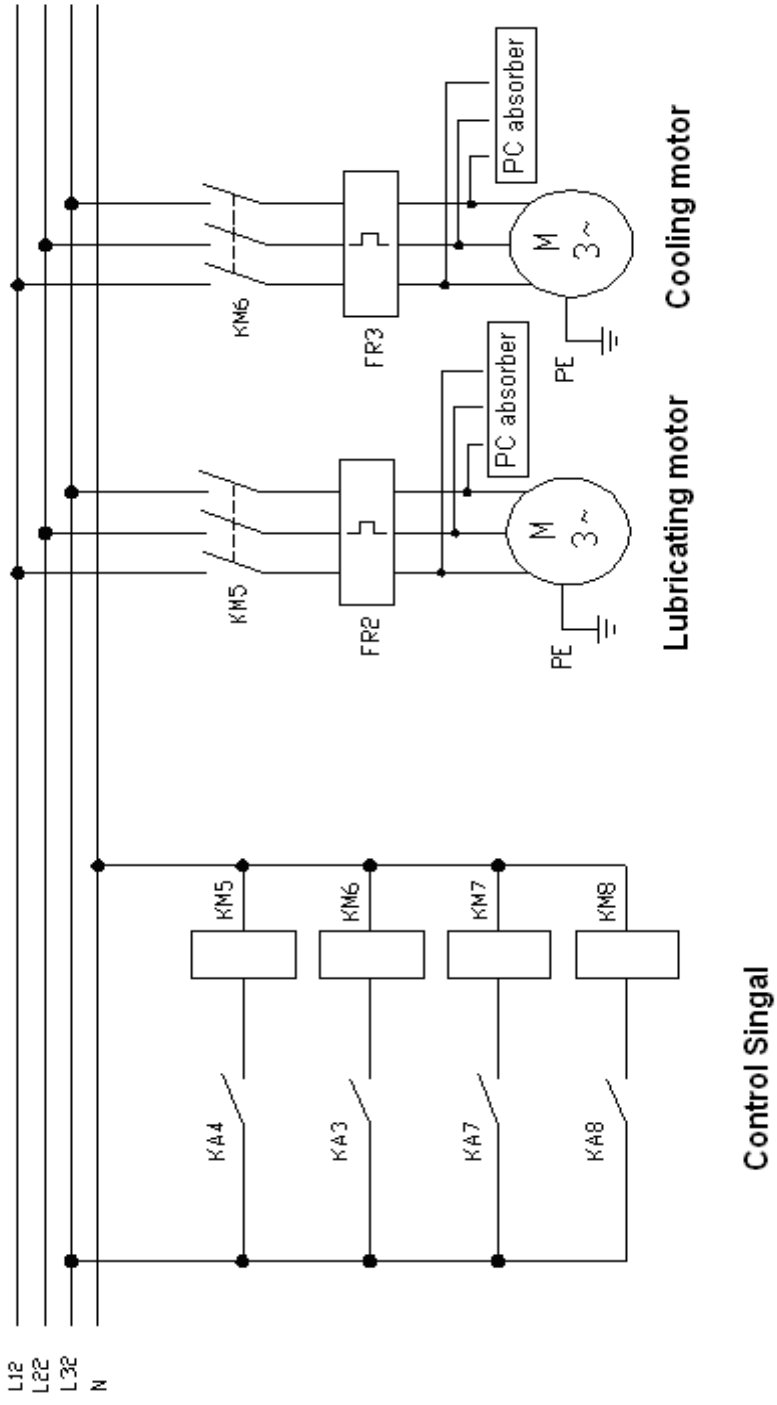


Q2BYG1106M **Step Motor System**

2.4.5 IO Electric Connection Diagram



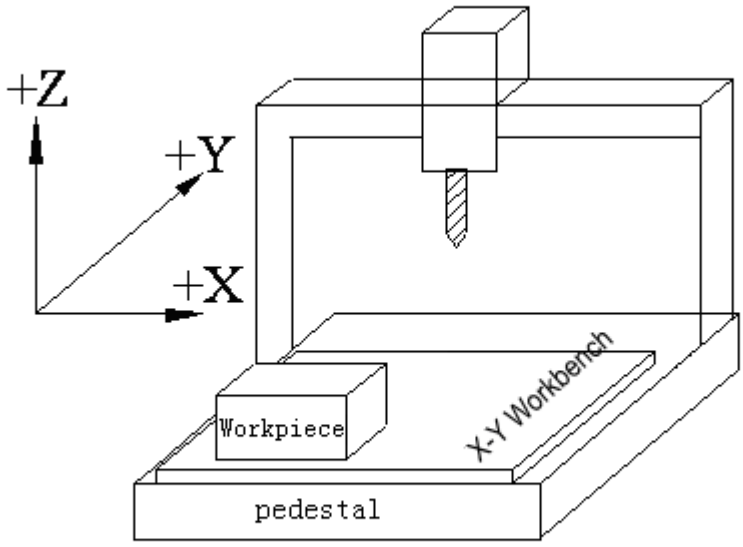

25Pin
Output terminal
(CNC4340)
(XS6 Interface)
1
2
3
4
5
6
7
8
9
10
11
12
13
14
15
16
17
18
19
20
21
22
23
24
25
M03
M04
M56
M57
M58
M59
M8
M9
M32
M33
M10
M11
M20
M21
M12
M13
M14
M15
M16
M17
M18
M19
SV4
SV3
SV2
SV1
CNC□M
Spindle on CW (FWD)
Spindle on CCW (REV)
Spare
Spare
Cooling
Lubricating
Spare
Spare
Spare
Spare
Spare
Spare
Spare
Spare
Spare
Spare
Spare
Spare
Spindle segment rate switch 3
Spindle segment rate switch 2
Spindle segment rate switch 1
Spindle segment rate SW 0
Output public interface
GND1
DC+(24VDC)
kA1
kA2
kA3
kA4
kA5
kA6
kA7
kA8
kA9
kA10
KA11
KA12
KA13
KA14

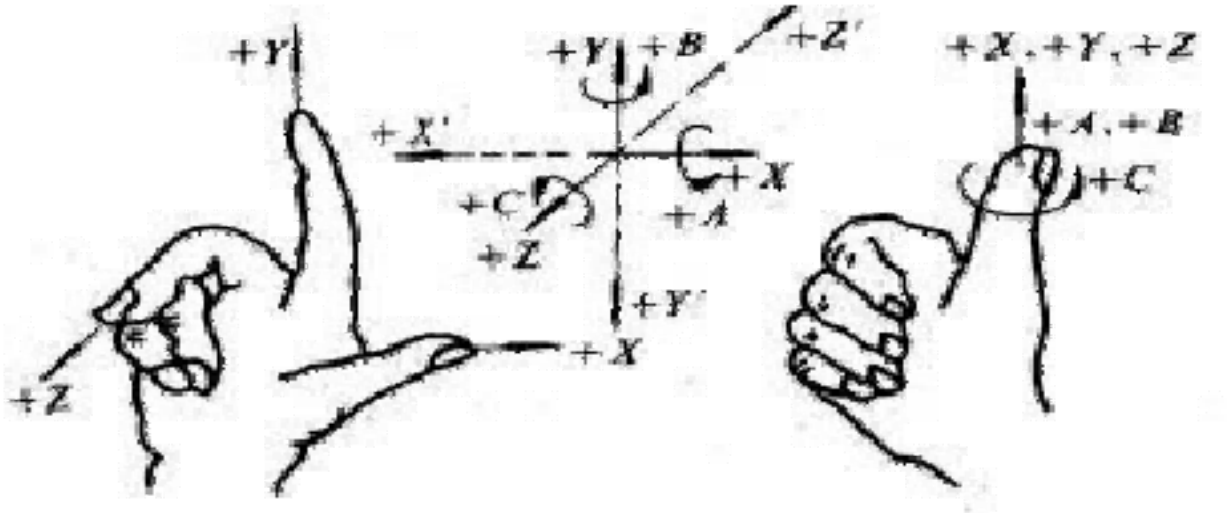

L12
L22
L32
N
KM5
KM6
KM7
KM8
KA4
KA3
KA7
KA8
Control Singal
KM5
FR2
PC absorber
M
3~
PE
Lubricating motor
KM6
FR3
PC absorber
M
3~
PE
Cooling motor

3 G Code Program

3.1 Basic knowledge of program

3.1.1 Motion direction and name of control axis





This system can control the fast moving for four axes. For feeding, it can control the interpolation for three axes.

The definition of the axis direction, adopt Cartesian coordinate system, as follows, (in the face of machine tool):

Z: When you face the machine tool: The upward and downward movements of the cutter relative to the workpiece is called the axis Z movement. The upward

movement of the cutter is called the positive-direction movement of axis Z, whereas downward movement negative-direction movement of axis Z.

X: The leftward and rightward movements of the cutter relative to the workpieve is called the axis X movement. The leftward movement of the cutter is called the negative -direction movement of axis X, whereas rightward movement positive-direction movement of axis X.

Y: The forward and backward movements of the cutter relative to the workpieve is called the axis Y movement. The forward movement of the cutter is called the positive-direction movement of axis Y, whereas backward movement negative-direction movement of axis Y.

Main shaft: look down the workpiece, the clockwise rotation is the natural rotation of the main shaft, anticlockwise is the opposite rotation.

A,B,C: the positive direction of the rotation coordinate axis is the positive directoin of the X, Y, Z coordinate axis accordingly, according to the onward direction of the right hand whorl to confirm.

Note: In this User's Manual, the movements described on X, Y and Z axes refer to the movement relative to the workpiece. In other words, a coordinate system is assumed for the workpiece.

3.1.2 Machine tool coordinate system and workpiece coordinate system（G53、G54～G599）

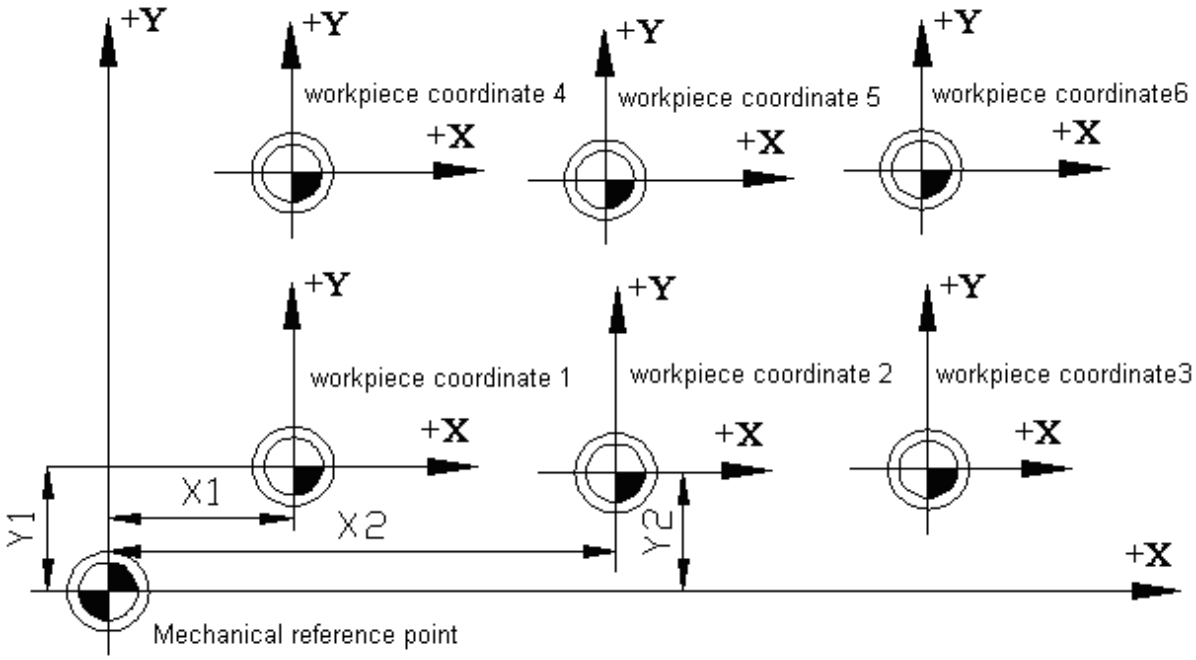
1）Machine tool coordinate system

The coordinate system of this machine tool is a fixed one on it. The establishment of this coordinate system is based on the operation each time the system returns to the reference point after NC is electrified. To select the coordinate system of the machine tool, G53 instruction is used.

2）Workpiece coordinate system

The workpiece coordinate system is used when the program is activated for machining, for which some benchmark point is set as the origin. Normally, in the process of programming, the programmers do not know where the workpiece is on the machine tool. The workpieve programs they compiled often take some point on the workpieve as the reference point. Therefore, the coordinate system set on the basis of this reference point is called workpieve coordinate system. When the workpiece to be processed is fixed on the machine tool, first the cutter will be moved to the designated reference point, and the coordinate value of this point of the machine tool is set at the origin of the workpiece coordinate system. Thus, when the system executes the machining programs, the cutter will perform the machining actions by taking this workpiece coordinate system as its reference object. For above reasons, the offset of the coordinate system's origin is of great significance for the CNC machine tools.

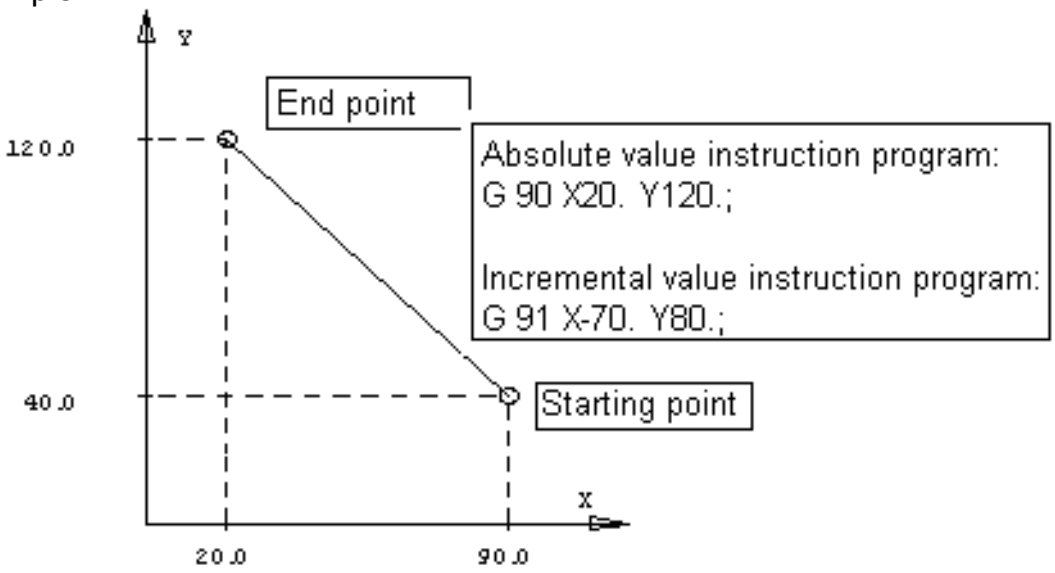
This System can be set with six workpieve coordinate systems (nine expansion coordinate systems, ranging from G591 to G599, are added for the new version system). In operation, the offset value of the coordinate system's origin of each workpiece relative to the origin of the machine tool's coordinate system should be set. Then G5X (5X represents the number of the actual workpieve coordinate system. It is same for the following part) instruction is used to select them. G5X serves as the mode status instruction, respectively corresponding to the pre-set workpieve coordinate systems ranging from 1#-6#.



3）Absolute coordinate program and relative coordinate program（G90、G91）

Cutter movement instructions are classified as absolute value instruction and incremental value instruction. In the mode status of absolute value instruction, what's designated is the coordinate value of the end point of movement in the current coordinate system; In the mode status of increment value instruction, is the designated axes relative to the movement away from the starting point.

G90……absolute value instruction
G91……incremental value instruction
For example：



From above introduction, we may better understand the programming with both absolute value method and increment value method.

3.1.3 The mode status function and the non-mode status function

The mode status function means that once a code is designated in the current program segment, it will be effective till another code of the same group in the program segment appears. And if this instruction is used in the next program segment again, it doesn't need to be designated.

The non-mode status function means a code can function only in its program segment. If this instruction is used again for the next program segment, it must be re-designated.

For example：

N0 G54 G0 X0 Y0； (Select the workpiece coordinate system, fast position to X0 Y0)

N1 G01 X150. Y25. F100 ；(Linear interpolating to X150, Y25)

N2 X50. Y75. F120； (Linear interpolating to X50, Y75. G01 is a mode status instruction and can be omitted)

N3 X0； (Linear interpolatig to X0, Y75. F120 is a mode status instruction and can be omitted)

3.1.4 Feeding

The feed of CNC machine tool can be classified as two types: fast locating feed and cutting feed.

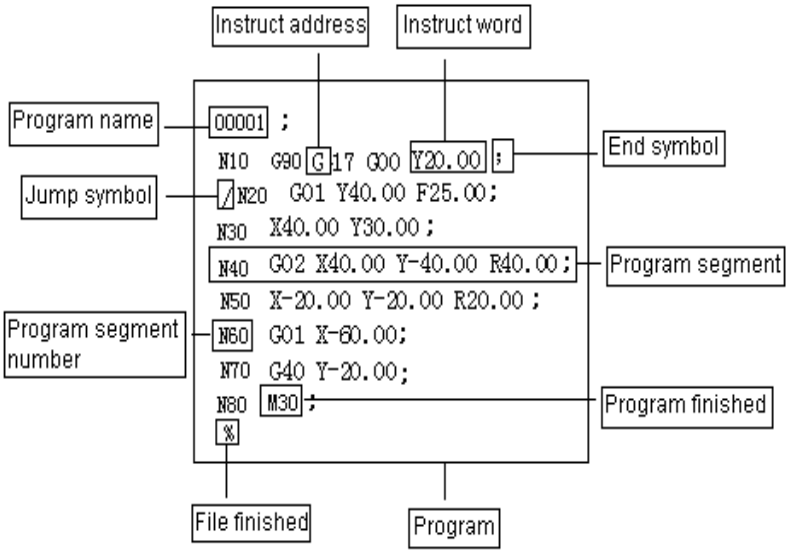
The fast locating feed appears when G00, fast manual move and the movement between fast feeding and locating in the fixed cycle are engaged. The speed of fast locating feed is determined by the machine tool's parameters. When this mode is used, the movements of the axes engaged in the feeding are irrelevant to each other. These axes move respectively at the rate set by the parameter. Normally, the locus of the cutter is shaped as a fold line or straight line.

Cutting feed is used in the case of G01, G02 and 03 and when machining feed in fixed cycle is involved. The speed of the cutting feed is determined by the address F, with its unit as mm/min. In the machining program, F is the value of a mode status. In other words, the originally programmed F value remains effective before the new F value is given. At the beginning of time the CNC system is electrified, the F value is set by the system parameter. The interpolation relation is remained between the axes engaged in feeding. The combination of their movements become the cutting feed movement.

The max. value of F is determined by the system parameter. If the programmed F value is greater than this value, this value will remain unchanged for the actual cutting feedrate.

The cutting feedrate can also be controlled by the switch of feed percentage on the control panel. The actual cutting feedrate should be the product of the given F value and feed percentage. The rate range is 10%-150%.

3.1.5 Program structure



In the text of a machining program, one English letter is called a instruction address that's followed by a numeric number to form the a instruction word. One or multiple instruction word s suffixed by the mark ";" constitute one program segment. And multiple program segments form a machining program. The instruction word serves as the basic unit to constitute the program segment. Each address has different meaning, whose following numeric number has different format and value range accordingly. Please refer to the table below:

Function	Add	Range	Meaning
program name	O	1～9999	program number
program segment No.	N	1～9999	Serial No.
Prepared to function	G	00～99	NC designated function
Size definition	X，Y，Z	±99999.999mm	Location coordinates value
	R	±99999.999mm	Radius, fillet radius
	I，J，K	±9999.9999mm	Coordinate of center of circle
feedrate	F	1～100,000mm/m	feedrate
Spindle Speed	S	1～4000 rotate per minute	Spindle Speed Value
Select Cutter	T	0～99	Cutter No.
Assistant function	M	0～99	Assistant function of M code
Cutter offset number	H，D	1～200	Designated cutter offset number
Pause time	P，X	0～65 second	Pause time(millisecond)

Designated subprogram number	P	1～9999	Invoke subprogram number
The number of repeat	P，L	1～999	Invoke subprogram number
Parameter	P，Q，R	P is 0～99999.999 Q is ±99999.999 mm R is ±99999.999	fixed cycle parameter

In addition, an optional number N × can be used at the beginning of a program segment for identifying it. It must be noted that the execution order of program segment is related only to the position in the memory where the program is saved, not to the program segment number. In other words, even if the program segment numbered as N20 is in front of the one numbered as N10, the one with the number of N20 will be executed earlier.

If the first character of some program segment is "/", it means this is a conditional program segment. That is to say, when the jump switch is at the upper position, this program segment won't be executed, whereas when the jump switch is at the lower position, this program segment can be executed.

1）Main program and subprogram

The machining program consists of the main program and subprogram. Basically, NC executes the instructions from the main program. When it executes a evoke instruction from the subprogram, NC will change to execute the subprogram. It will return to the main program when it executes the return instruction from the subprogram.

When the machining program needs to run the same locus for multiple times, we can program this locus into a subprogram and save it in the program memory of the machine tool. Then each time this locus is executed in the program, we can invoke the subprogram.

When a main program invokes a subprogram, this subprogram can also invoke another subprogram. This is called dual nest of subprogram. A machine tool can allow a subprogram of quadruple nest at maximum. When the subprogram instructions are invoked, the invoked subprogram can be repeatedly executed through the instruction, with a max. repetition number up to 999 times.

A subprogram should has the structure as below:

O××××; subprogram number
……;
……; subprogram contents
……;
M99; Return to main program

The program should begin with a subprogram number designated by address O. At the end of the program, the instruction M99 for returning to main program must be included. M99 may not be seen in a individual program segment. As the end of the subprogram, such a program segment is acceptable:

G90 G00 X0 Y100. M99；

In the main program, the program segment that invokes the subprogram must include the contents below:

M98 P×××××××；

Here, in the numbers following address P, the last four digits are used for designating the number of the subprogram to be invoked, the front three digits for designating the repeated times to be invoked.

M98 P51002； To invoke subprogram No.1002, repeat 5 times.

M98 P1002； To invoke subprogram No.1002, repeat 1 times.

M98 P50004； To invoke subprogram No.4, repeat 5 times.

The invoke instruction can appear in the same program segment as the motion instruction:

G90 G00 X-75. Y50. Z53. M98 P40035；

This program segment instructs axis X, Y and Y to move to the designated position with the speed of fast locating feed, then invoke to execute subprogram No.35 for four times.

Unlike other M codes, when M98 and M99 are executed, no signal is sent to the side of machine tool.

When NC can't find out the program number designated by address P, the alarm will be sent out.

The invoke instruction of subprogram—M98 can't be executed under the MDI mode. If a subprogram needs to be executed individually, you can edit the program in the programming mode as follows and execute it in the auto running mode.

O×××；

M98 P××××；

M30；

2）Program finished

When the following codes are seen at the end of the program, it means the program part is finished.

EIA	ISO	Define
M30 CR	M30 LF	The end of the program and return to the beginning of the program
M99 CR	M99 LF	subprogram finished

In executing the program, if the abovementioned program-end code is detected, the device will finish executing the program and the system will enter the reset state. In the case of M30, CR or M30 LF, the system will return to the beginning of the program (in an auto way). In the case of end of subprogram, the system will return to the program which invokes the subprogram.

3）File finished

EIA	ISO	Define
ER	%	program finished

Remark: If ER(EIA) or %(ISO) is executed without M30 at the end of the program, CNC will change to the reset state.

3.2 Preparatory Functions（G Code）

3.2.1 G Code of list

G Code	Set	Function
G00	01	Locate(fast move)
G01		Linear interpolation (cut feed)
G02		Arc-circle interpolation CW
G03		Arc-circle interpolation CCW
G04	00	Pause, Stop
G17	02	XY plane selection
G18		ZX plane selection
G19		YZ plane selection
G20	06	Input data of British system
G21		Input data of metric system
G28	00	Return to reference point
G29		Return from reference point
*G40	07	Write-off of cutter radius compensation
G41		Compensation of left cutter radius
G42		Compensation of right cutter radius
G43	08	Length of positive-direction cutter
G44		Length of negative-direction cutter
*G49		Write-off of cutter length offset
*G54	05	Workpiece coordinate system 1
G55		Workpiece coordinate system 2
G56		Workpiece coordinate system 3
G57		Workpiece coordinate system 4
G58		Workpiece coordinate system 5
G59		Workpiece coordinate system 6
G591		Coordinate system of expansion workpiece 7
G592		Coordinate system of expansion workpiece 8
G593		Coordinate system of expansion workpiece 9
G594		Coordinate system of expansion workpiece 10
G595		Coordinate system of expansion workpiece 11
G596		Coordinate system of expansion workpiece 12
G597		Coordinate system of expansion workpiece 13
G598		Coordinate system of expansion workpiece 14
G599		Coordinate system of expansion workpiece 15
G65	00	Macro program command (not developed for 4340, test version)
G73	09	Fixed cycle for drilling and cutting deep holes
G74		Fixed cycle for reverse-thread tapping

G76		Fixed cycle for fine boring
*G80		Cancel fixed cycle
G81		Fixed cycle for drilling and cutting
G82		Fixed cycle for drilling and cutting
G83		Fixed cycle for drilling and cutting deep holes
G84		Fixed cycle for tapping
G85		Fixed cycle for boring and cutting
G86		Fixed cycle for boring and cutting
G87		Fixed cycle for reverse boring and cutting
G88		Fixed cycle for boring and cutting
G89		Fixed cycle for boring and cutting
*G90	03	Absolute value program
G91		Incremental value program
G98	10	Return to initial plane in fixed cycle
G99		Return to R point plane in fixed cycle

Note: Items with " * " are the defaulted values of mode status for G codes of groups in the system.

3.2.2 Interpolation Functions（G00、G01、G02、G03）

1）Fast locating（G00）

Format:

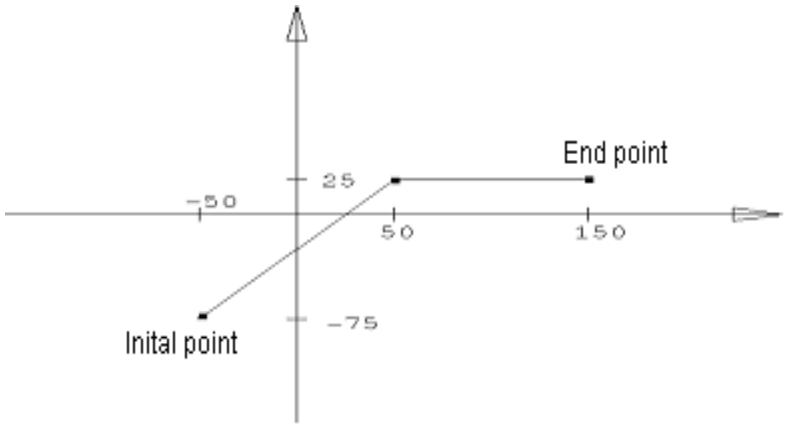
G00 X_Y_Z_;

X_Y_Z_: coordinate value, whether it is a absolute position value or incremental position value will be determined by the mode status value of G90 or G91.

The instruction G00 allows each shaft to move to the designated position with the set fast speed. The instructed shafts are irrelevant to each other. In other words, the locus of the cutter is a straight line or fold line. The moving speed of each shaft under the instruction G00: at axis X, Y and Z, the shaft will move according to the set parameter, and this speed is not controlled by the current F value. When all shafts reach the end points, CNC will consider that this program segment is finished and the system will change to execute the next program segment.

Example of G00 programming:

The starting point is set as X and instruction as Y. The cutter will move to form the locus as shown in the figure below.



2）Linear interpolation（G01）

Format：

G01 X_Y_Z_F_；

X_Y_Z_ ： It refers to the coordinate value. It can be absolute or incremental value according to the current state of G90 or G91.

F ： It refers to the speed.

The instruction G01 allows the current interpolation mode status to be changed to linear interpolation mode status. The cutter will move from the current position to IP designated position, whose locus is a straight line. F- designates the speed with which the cutter moves along the line, with its unit as mm/min.

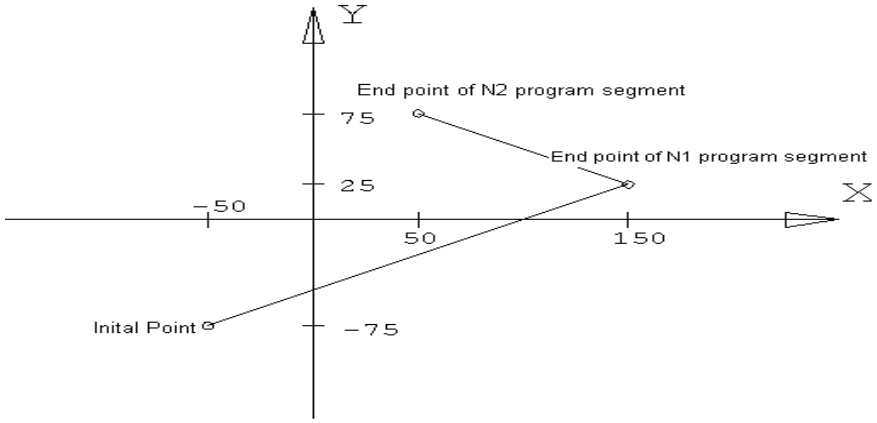
G01 for example:

Suppose the current cutter is at the point X-50. Y-75., the program segment is as follows:

N1 G01 X150. Y25. F100 ；

N2 X50. Y75.；

Out of the tool will track as follows.



3）Arc-circle interpolation（G02/G03）

The instructions listed below can enable the cutter to move along the arc locus:

In X-Y plane

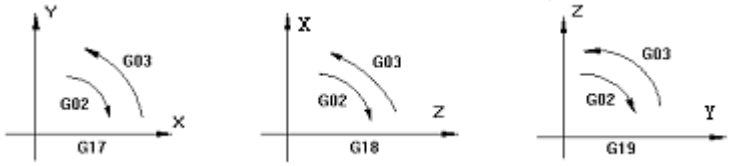
G17 { G02 / G03 } X__ Y__ { （ I__ J__ ） / R__ } F__ ；

In X-Z plane

G18 { G02 / G03 } X__ Z__ { （ I__ K__ ） / R__ } F__ ；
In Y-Zplane
G19 { G02 / G03 } Y__ Z__ { （ J__ K__ ） / R__ } F__ ；

No.	Content		Command	Define
1	select plane		G17	Designate the arc interpolation on X-Y plane
			G18	Designate the arc interpolation on Z-X plane
			G19	Designate the arc interpolation on Y-Z plane
2	Arc direction		G02	Arc interpolation of clockwise direction
			G03	Arc interpolation of counter-clockwise direction
3	End position	G90 mode	Two-axes instruction in X, Y and Z	Coordinate value of end position in the current workpiece coordinate system
		G91 mode	Two-axes instruction in X, Y and Z	Distance between the start point and origin (with direction)
4	Distance between the start point and origin		Two-axes instruction in X, Y and Z	Distance between the start point and origin (with direction)
	Arc radius		R	Arc radius
5	Feed rate		F	speed of along-the-arc movement

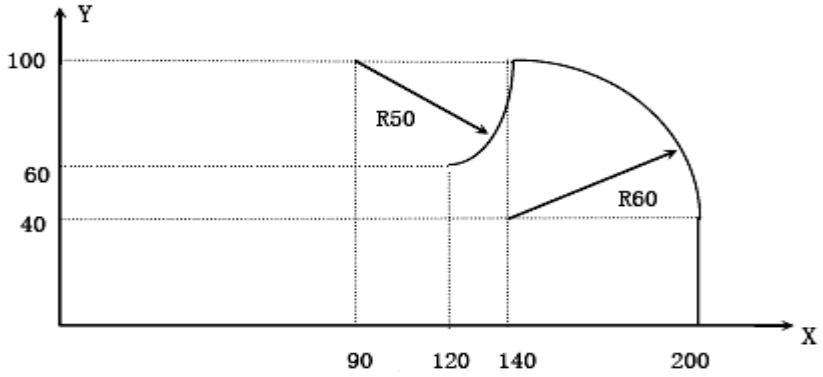
The arc direction mentioned here refers to the direction for which the XY plane is viewed from the positive direction of Z axis to its negative direction. Similarly, for XY or YZ plane, the observing direction should be from the positive direction of Y axis or X axis to its negative direction (this is applicable for right-hand coordinate system, as shown below).



The end point of the arc is determined by the address X, Y and Z. In G90 mode status, which is the absolute mode status, the address X, Y and Z tell the coordinate value of the arc's end point in the current coordinate system. In G91 mode status, which is the incremental mode status, what X, Y and Z tell are the distances between the current point of the cutter and the end point along the coordinate axes.

To X direction, the address I tells the distance between the point of current cutter and the center of circle. To X and Y direction, the distance between the point of current cutter and the center of circle is given the address J and K. The symbol of I, J and K are determined by the respective movement direction.

To program a segment of arc, in addition to the method of given end point position and circle center position, we can also use the given radius and end point position, and use address R to tell the radius and replace the address of given circle center. The R value can be positive and negative. Normally, a positive R value is used for programming a segment of arc which is less than 180°, whereas a negative R value is used for programming a segment of arc which is more than 180°. To program a whole circle, we have to use the method of given center of the circle.



Use absolute value method and incremental value method respectively to program the locus in the diagram.

(1) absolute value method

G00 X200.0 Y40.0 Z0 ;
G90 G03 X140.0 Y100.0 I-60.0 F300.0 ;
G02 X120.0 Y60.0 I-50.0 ;
or
G00 X200.0 Y40.0 Z0 ;
G90 G03 X140.0 Y100.0 R60.0 F300.0 ;
G02 X120.0 Y60.0 R50.0 ;

(2) incremental value method

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

Use F to designate the feedrate of arc interpolation, which is the cutter's speed along the tangent direction of the arc.

3.2.3 Pause Instruction（G04）

Function: To cause a pause between two program segments.

Format： G04 P-

G04 X-

Address P tells the time of pause. When there is no decimal, the min. value of the instruction is 0.001 second.

Address X tells the time of pause. When there is no decimal, the min. value of the instruction is 1 second.

Example：G04 P 1000 ： Pause 1000millisecond,as 1second.

G04 X 1 ： Pause 1 second.

3.2.4 Select Plane（G17、G18、G19）

This group of instructions are used for the plane of selected arc interpolation and of cutter radius compensation. The method is shown below:

G17……Select XY plane

G18……Select ZX plane

G19……Select YZ plane

G17, G18 and G19 are in the program segment without instruction, the plane remains unchanged.

For example：

G18 X_ Z_ ；ZX plane

X_ Y_ ；No change plane (ZX plane)

In addition, the move instruction is irrelevant to the plane. For example, under the following instruction, Z axis is not on XY plane, and the movement of Z axis is irrelevant to XY plane.

G17 Z_ ；

For relevant instructions of the plan selection, please refer to the instructions of the circular interpolation and the cutter compensation.

3.2.5 Coordinate Instruction（G53 ~ G59、G591 ~ G599、G92）

1）Selecting coordinate of machine tool（G53）

Format： G53 X_Y_Z_；

X_Y_Z_：The absolute coordinate value or relative position in the coordinate system

When this instruction is executed under G90 mode status, the cutter moves to the IP-designated coordinate position in the machine tool coordinate system at the fast feedrate. When this instruction is executed under G91 mode status, the cutter moves at the incremental value of the selected coordinate system. G53 is a non-mode status instruction. That is to say, it can only function in the current program segment.

The distance between the zero of machine tool coordinate system and the reference point is set bythe parameter. Unless otherwise stipulated, the reference point of each axis coincides with the zero of the machine tool coordinate system.

2）Use presetting workpiece coordinate system（G54～G59，G591～G599）

Based on the mounted position of workpiece on the machine tool, this System can provide six workpiece coordinate systems via presetting (the new version is expanded to 9 coordinate systems). Through the operations via the LCD panel, the offset of the origin of each workpiece coordinate system relative to the origin of that for machine tool can be set. Then the instruction G is used to select them. G is a mode status instruction, which corresponds to the preset workpiece coordinate systems ranging from 1＃～15＃. See the example below:

Preset the offset of 1# workpiece coordinate system: X-150.000 Y-210.000 Z-90.000。

Preset the offset of 4# workpiece coordinate system: X-430.000 Y-330.000 Z-120.000。

Program segment	Coordinates value of end point in the machine tool coordinate system	Define
N1 G90 G54 G00 X50. Y50.;	X-100, Y-160	Select 1# coordinate system, fast locating
N2 Z-70.;	Z-160	
N3 G01 Z-72.5 F100;	Z-160.5	Linear interpolating, F value is 100
N4 X37.4;	X-112.6	(Linear interpolating)
N5 G00 Z0;	Z-90	Fast locating
N6 X0 Y0 A0;	X-150, Y-210	
N7 G53 X0 Y0 Z0;	X0, Y0, Z0	Select to use machine tool coordinate system
N8 G57 X50. Y50. ;	X-380, Y-280	Select 4# coordinate system
N9 Z-70.;	Z-190	
N10 G01 Z-72.5;	Z-192.5	Linear interpolating, F value is 100 (mode status value)
N11 X37.4;	X392.6	
N12 G00 Z0;	Z-120	
N13 G00 X0 Y0 ;	X-430, Y-330	

From above example, we can see that the role of G54-G59 is to move the origin of the coordinate system NC uses to the point with the preset coordinate value in the machine tool coordinate system. For the presetting method, please refer to the part describing operations in this Manual.

Once the system returns to zero after started up, the workpiece coordinate systems ranging from 1-6 will be established. G54 is the initial mode status at the time of electrifying. The absolute position of the position image is the coordinate value of the current coordinate system.

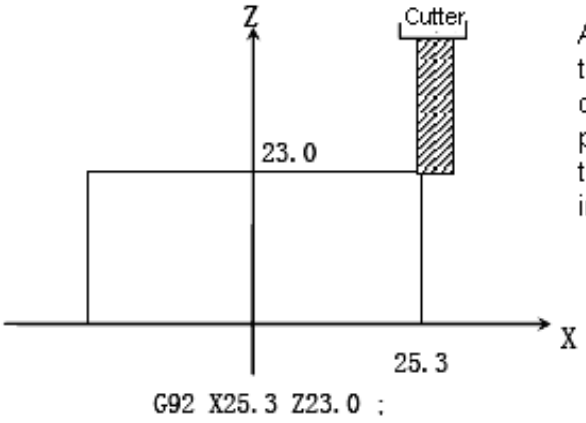
In the numeric control programming for the machine tools, the interpolation instruction and other instructions related to the coordinate value refer to the coordinates in the current coordinate system (the system when the instruction is executed), unless otherwise stipulated. In most cases, the current coordinate system is the one from G54-G59. It is a rare case that the machine tool coordinate system be used directly.

3）Programmable workpiece coordinate system（G92）

Format：（G90）G92　X_Y_Z_；

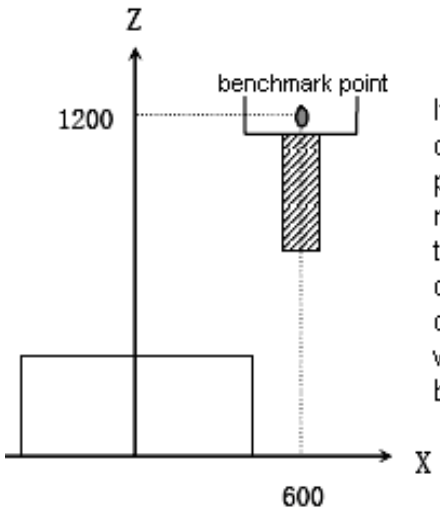
This instruction help establish a new workpiece coordinate system, in which the coordinate of the current cutter's point is the IP-designated value. G92 is non-mode status instruction. However, the workpiece coordinate system established on the basis of this instruction is of mode status nature. In reality, this instruction also gives a offset in a indirect manner, which is the coordinate value of the origin of the new workpiece coordinate system in the original coordinate system. From the performance of G92, we can see that this offset is the difference between the coordinate value in the original system and the IP-designated value. If G92 is used for many times, the offset provided each time G92 is used will be added up. For each preset workpiece coordinate system (G54-G59), this added offset is effective.

The new coordinate system of the part is therefore established by using the abovementioned instructions. For example, the coordinate value of the cutter tip can be IP-. Once the coordinate is determined, the position of the absolute value instruction is the coordinated value in this coordinate system.



As shown in the figure, if the cutter tip is considered as the start point of the program, use the program start instruction G92.

G92 X25.3 Z23.0 ;



If some benchmark point on the hilt is considered as the start point, use the program start instruction G92. If the cutter moves according to the absolute value in the program, the cutter length must be compensated when it moves to the designated position. The compensation value is the difference between the benchmark point and the cutter tip.

Use G92 X600.0 Z1200.0 ; Use instruction for setting the coordinate system (some benchmark point on the hilt as the cutter start point)

Note: a. If G2 is used for setting the coordinate system in cutter offset, the coordinate system set by G92 will be employed for the compensation of cutter length.

b. For compensation of cutter radius, cutter offset should be cancelled when G92 is used.

For example：

Preset the offset of 1# workpiece coordinate system：X-150.000 Y-210.000 Z-90.000。

Preset the offset of 4# workpiece coordinate system：X-430.000 Y-330.000 Z-120.000。

Program segment content	In the end of the machine tool coordinate system of coordinates	Define
N1 G90 G54 G00 X0 Y0 Z0;	X-150, Y-210, Z-90	Select 1# coordinate system and fast position to origin of coordinate system.
N2 G92 X70. Y100. Z50.;	X-150, Y-210, Z-90	Don't move the cutter, and establish the new coordinate system, in which the current point has the following coordinate values: X70, Y100, Z50. Fast position to new origin of coordinate system.
N3 G00 X0 Y0 Z0;	X-220, Y-310, Z-140	fast position to new origin of coordinate system.
N4 G57 X0 Y0 Z0;	X-500, Y-430, Z-170	Select 4# coordinate system and fast position to origin of coordinate system. (already offset)
N5 X70. Y100. Z50.;	X-430, Y-330, Z-120	fast position to primary origin of coordinate system.

4）Local coordinate system（G52）

G52 can establish a local coordinate system, which equals to the sub-coordinate system in G54-G59 system.

Format:G52 X_Y_Z_；

In this instruction, IP-gives an offset which equals to the current G54-G59 coordinate systems. In other words, IP-gives the origin of the local coordinate system the position coordinate in the current G54-G59 coordinate systems, even if a local coordinate system is established by a G52 instruction before the instruction G52 is executed. To cancel the local coordinate system, you can simply use G52 IP0.

3.2.6Instructions related to reference point（G27、G28、G29）

The coordinate system of the machine tool is established by returning to the reference point each time NC is electrified. The reference point is fixed on the machine tool, whose position is determined by the installation place of baffle switch of each shaft and the zero position of each shaft's servo motor. In this machine tool, the coordinates of the reference point in the machine tool coordinate system are X0, Y0 and Z0.

Auto return to reference point（G28）

Format:G28 IP_；

This instruction enables the instruction shaft to return to the reference point of the machine tool via IP-designated middle point at the fast feedrate. The middle point can be designated by either the absolute value or incremental value, depending on the current mode status. Basically, this instruction is used to enable the workpiece to move out of the processing area after the machining program is finished so that the finished parts can be removed and the parts to be machined can be loaded.

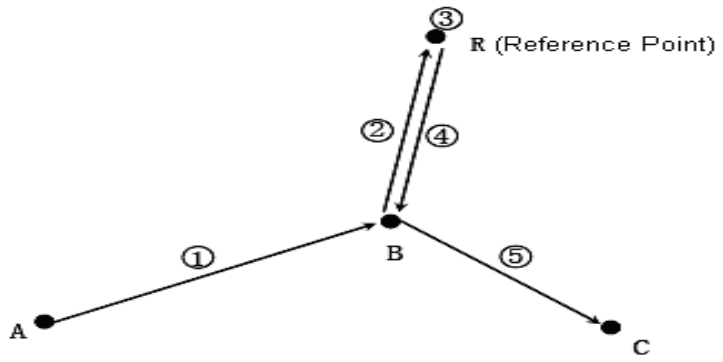
When instruction G28 is executed before the system manually returns to the reference point, the movement direction of each shaft from the middle point is positive, like the movement for manually returning to the reference point.

The coordinate value of instruction G28 will be saved by NC as the middle point. On the other hand, if one shaft is not included within instruction G28, the coordinate value of the middle point of this shaft saved by NC will the previous value given by instruction G28.

For example:

N0010 X20.0 Y54.0;
N0020 G28 X-40.0 Y-25.0; the coordinate value of the middle poin（-40.0,-25.0）
N0030 G28 Z31.0; the coordinate value of the middle poin（-40.0,-25.0,31.0）

The coordinate value of this middle point is mainly used by instruction G29.



☞Notes：

Under the mode status of cutter offset, the cutter offset is also effective to instruction G27. Therefore, for the sake of safety, the cutter offset (radius offset and length offset) should be cancelled before instruction G28 is executed.

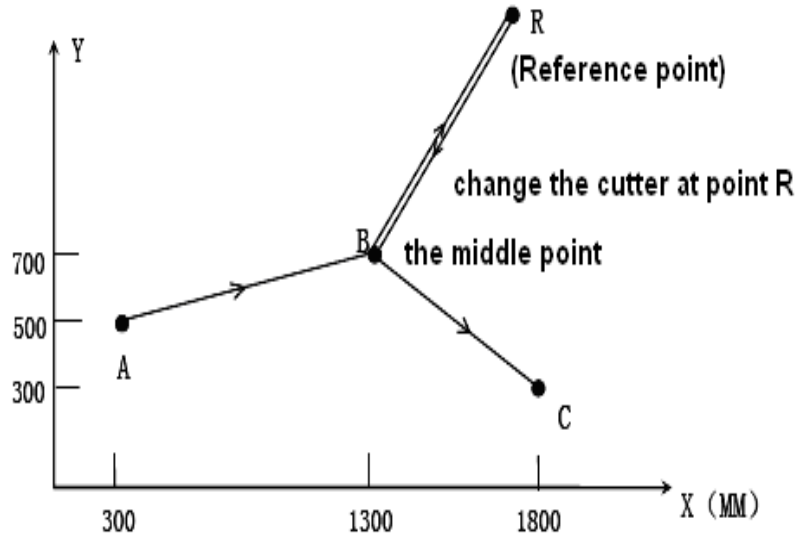
Auto return from reference point（G29）

Format：G29 IP-；

This instruction enables the instruction shaft to move to the instruction position from the reference point through the middle point at the fast feedrate. The position of the middle point is determined by the previous instruction G28. Normally, this instruction is used behind G28 when the instructed shaft is located at the reference point or the second reference point.

Under mode status of incremental value, the instruction value is the distance between the middle point and the end point (instruction position).

Application examples for G28 and G29.



G28 X1300.0 Y700.0 ； (program from A to B)

.....

G29 X1800.0 Y300.0 ； (program from B to C)

From the above example, we can see that it is unnecessary to calculate the actual movement from the middle point to the reference point .

Note: After the middle point is passed to reach the reference point when instruction G28 is used, the middle point will also be moved to the new coordinate system once the coordinate system is changed for the part. After that, when instruction G29 is executed, it is will be located at the designated place via the middle point.

Return for inspection from reference point （G27）

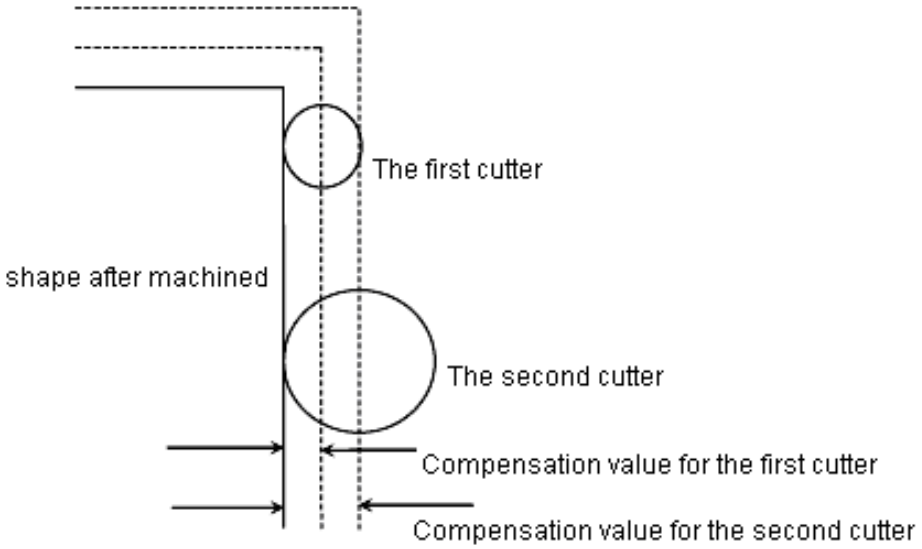
Format：G27 IP_；

This instruction enables the instruction shaft to move to the IP-designated position at the fast feedrate, then check whether this point is the reference point. If so, the system will send out the completion signal that this shaft returns to the reference point (the indicator for reaching the reference point by this shaft will be illuminated). If not, an alarm will be sent out and the running of the program will be stopped.

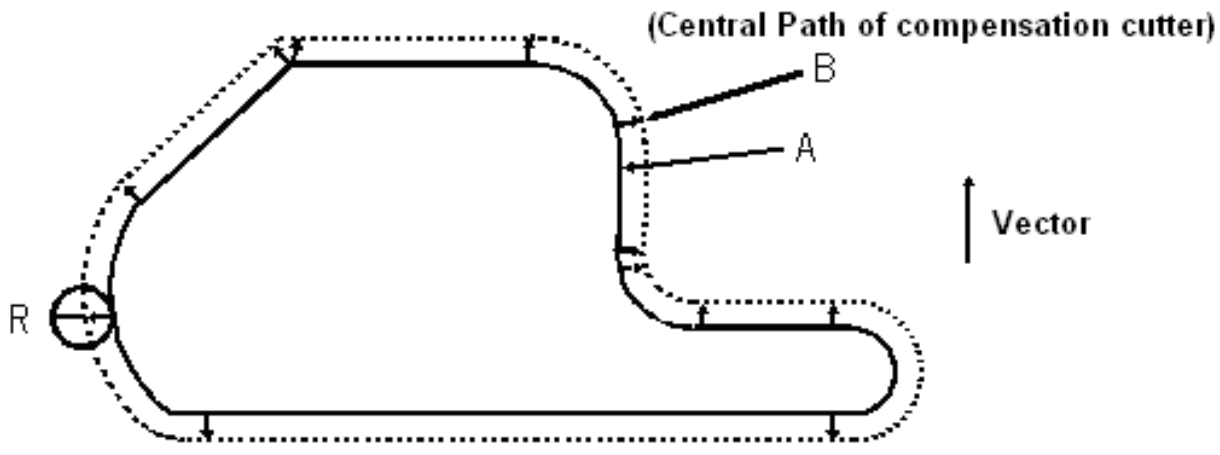
3.2.7 Cutter Compensation（G40、G41、G42、G43、G44、G49）

1）Cutter radius compensation

The cutter has a certain size (length and diameter). When the part with some shape is machined, the locus by which the cutter moves along will be subject to the nature of the cutter itself. If the data of the cutter's size are set in CNC in advance, the locus of the cutter will be automatically generated by CNC when the same program is used, even if cutters of different specification are employed. The data concerning the cutter size are called compensation amount (or offset).



As shown in the following figure, the cutter with radius R is used to cut the workpiece A, the central path of cutter is B, the distance between path B and A is R. The process that the cutter leaves the workpiece A for some distance is called "compensation". Programmers use the radius compensation mode to produce the machining programs. In actual machining, the radius of cutter will be measured and entered into CNC. The cutter path becomes the compensation path B.



Compensation and Vector

2）Compensation value (D Code)

Maximally, eighteen D00-D18 compensation values can be set in this System. In the program, the two numeric values after instruction D are the compensation amount. They must be set via the menu Cutter Compensation.

Set the amount of compensation are as follows:

	Mm input	Inch input
compensation value	0-±999.999mm	0-±999.999inch

3）Compensation vector

The compensation vector is of 2D nature, which equals the compensation value designated by code D. The calculation of compensation vector is accomplished within the control unit. In each program segment, its direction is modified according to the path of the cutter. This compensation vector is accomplished within the control unit so that how much compensation is needed for the cutter's move can be calculated. The compensation path (the central locus of cutter) equals the programming path plus or minus (subject to the compensation direction) the cutter radius.

Vector compensation is always concerned with cutting tools, in the preparation process, to understand the state vector is very important.

4）Plane selection and vector

The calculation for compensation can be executed within the plane selected by G17, G18 and G19. This plane is called compensation plane. For example, when XY plane is selected, （X，Y）or（I，J）will be used to execute the compensation and vector calculations in the program. The shaft which is not within the compensation plane will not be affected.

In the case of running three-shaft controller, only the cutter path projected onto the compensation plane can be compensated.

The compensation plane can be modified only after the compensation mode is cancelled. If it is modified in the compensation mode, the system will send out alarm signal and the running of the machine will be stopped.

G Code	compensation plane
G17	X-Y plane
G18	Z-X plane
G19	Y-Z plane

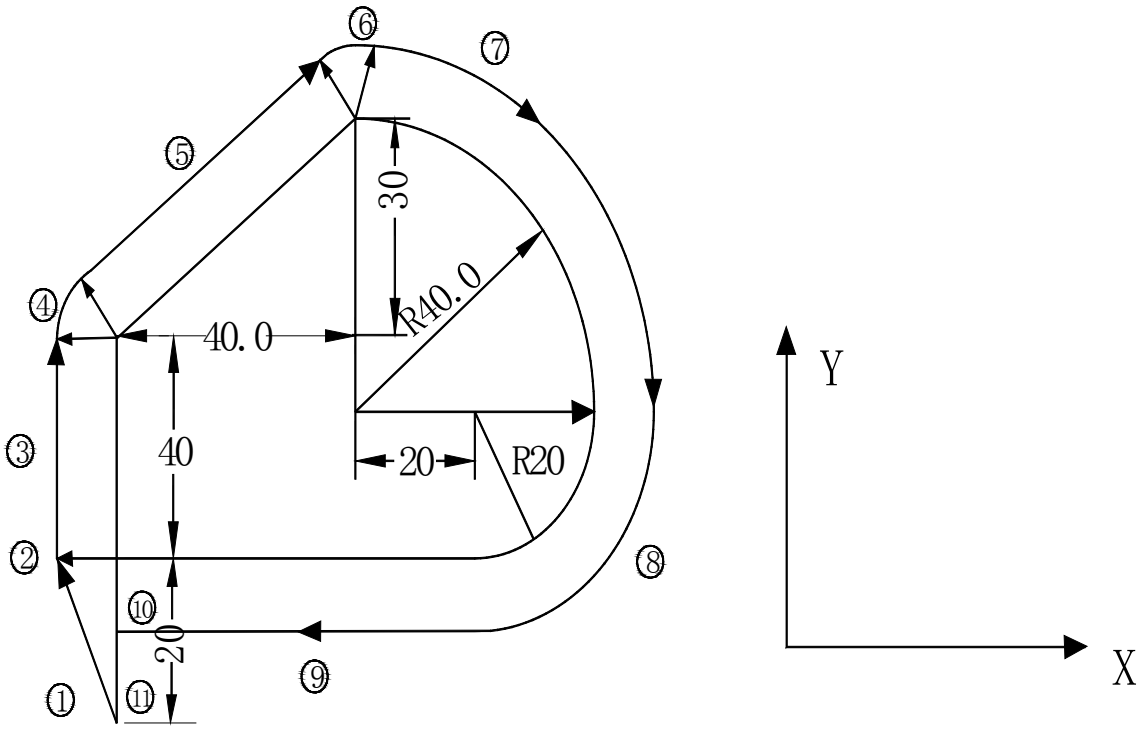
5）G40，G41 and G42

Use instruction G40, G41 and G42 to cancel and activate the compensation vector of the cutter radius. They are combined with instruction G00, G01, G02 and G03 to determine the value and direction of the compensation vector and moving direction of the cutter by defining a mode.

G Code	Function
G40	cancle the compensation of the cutter radius.
G41	left compensation of the cutter radius.
G42	right compensation of the cutter radius.

G41 or G42 allows the System to enter the compensation mode, whereas G40 allows the System to cancel that mode.

For example of compensation program:




```
O0007 ；
G0G40G49G80G90；
G0 X0 Y0；
N1   G91 G17 G00 G41 Y20.00 D07 ；
N2 G01 Y40.00 F25.00:
N3 X40.00 Y30.00:
N4 G02 X40.00 Y-40.00 R40.00:
N5 X-20.00 Y-20.00 R20.00:
N6 G01 X-60.00:
N7 G40 Y-20.00:
N8 M30
 %
```

Program segment (1) is used for start-up. In this program segment, instruction G41 changes the compensation canceling mode to compensating mode. At the end of this segment, the cutter center makes compensations by allowing the cutter radius to be vertical to the path direction of next program. The compensation value of cutter is designated by D07. That is to say, the compensation number is set as 7. G41 refers to the left compensation of cutter path.

6）Details of cutter radius compensation C

This part provides details of cutter radius compensation C.

a.Cancel mode

When the System is electrified/reset/executes instruction M02 and M30, the System will be in the cutter compensation mode.

The vector must be 0 in compensation mode, and the path of cutter center is consistent with programming path. The compensation mode G40 must be designated before the program is finished.

b. Compensation Start

In cancel mode, the System will enter the compensation mode when the program segment that satisfies the following conditions is executed:

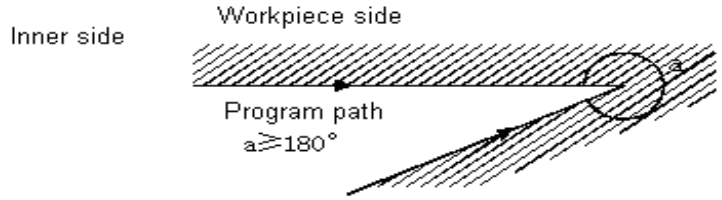
- Containing instruction G41 or G42, or the control section enters G41 or G42 mode.
- Offset number of cutter compensation is not zero.
- For movement of any axis (except I, J and K) on the instruction compensation plane, the movement value can't be zero.

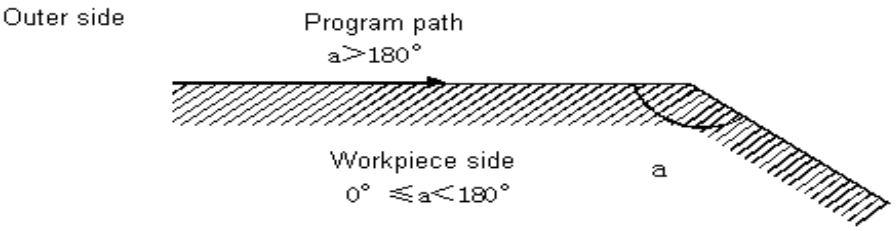
The program segment of compensation start should not have the arc instruction G02 and G03. Otherwise, the alarm (P/S34) will be activated. In compensation start segment, two program segments will be read. One is read and executed and the other enters the cutter compensation buffering area.

Under single program segment method, the second program segment is read and the first program segment is executed, and then stopped.

In continuous execution, normally two program segments are read in advance. Therefore, three program segments are available within CNC. One is the program segment being executed, and the next two program segments enter the buffering area

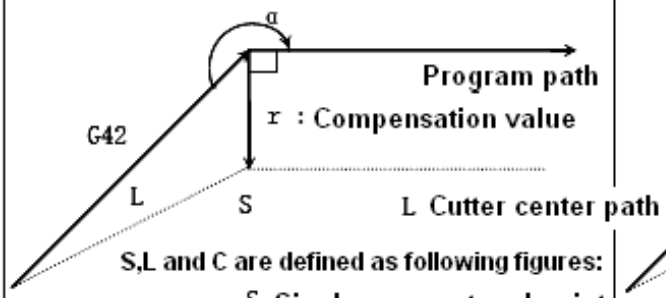
Note: In the descriptions below, the frequently seen terms, "inner side" and "outer side", are defined as: when the inclination of intersection of two moving program segments equals or greater than 180°, it is called "inner side", whereas the inclination is 0-180°, it is called "outer side" (see the following figures):





(a) Move along the inner side of the corner (α ≥180°)

(i) : beeline →beeline



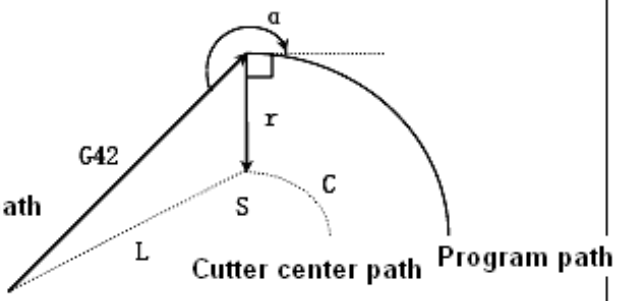
S,L and C are defined as following figures:

S: Single segment end point

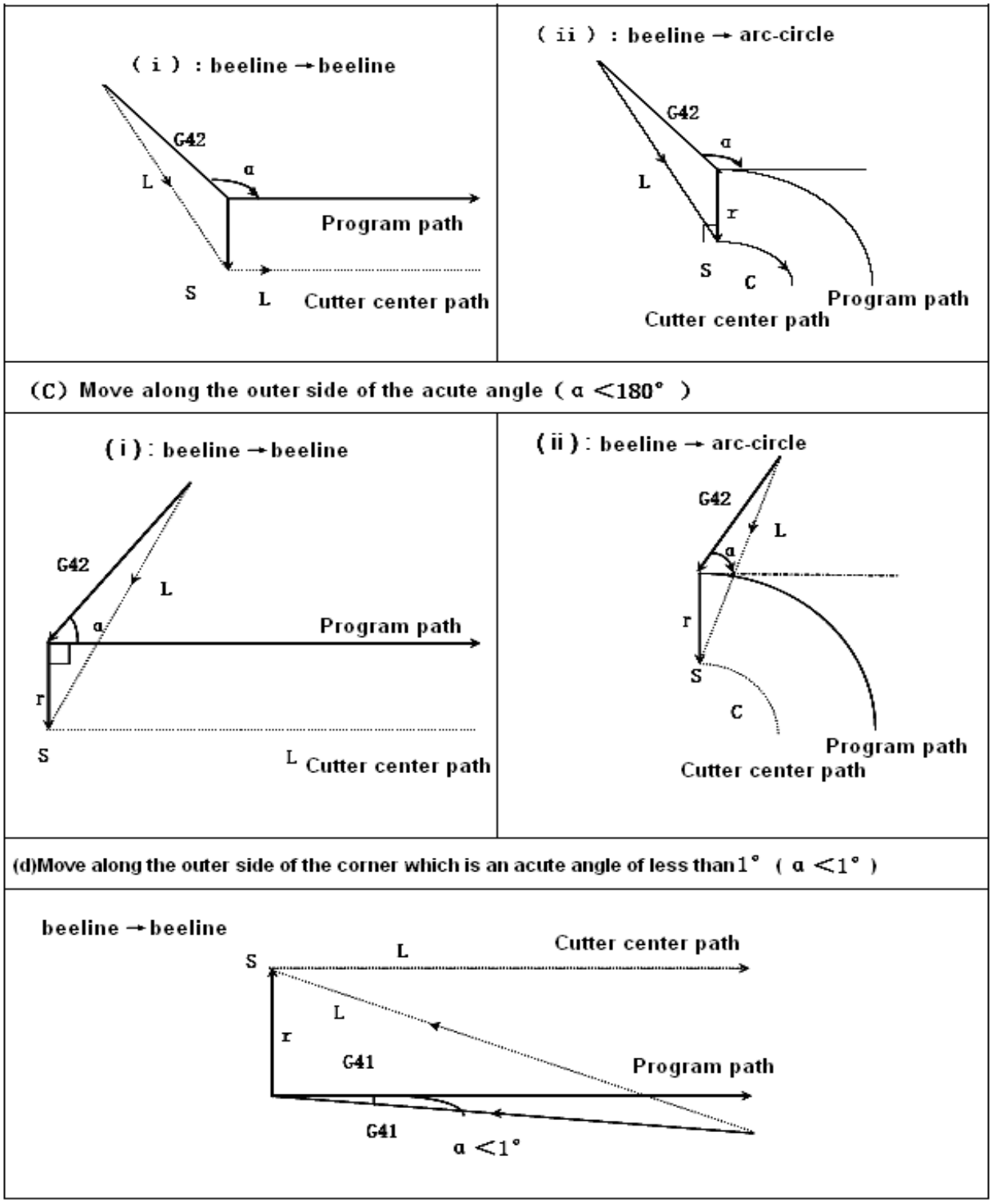
L: Beeline

C: arc-circle

(ii) : beeline→ arc-circle



(b)Move along the outer sider of the corner which is an obtuse angle. (180° > α ≥90°)

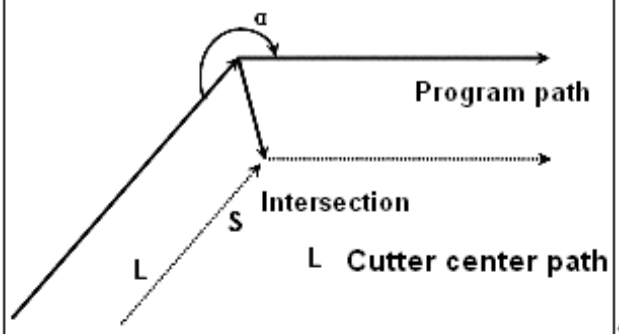


C. Compensation mode

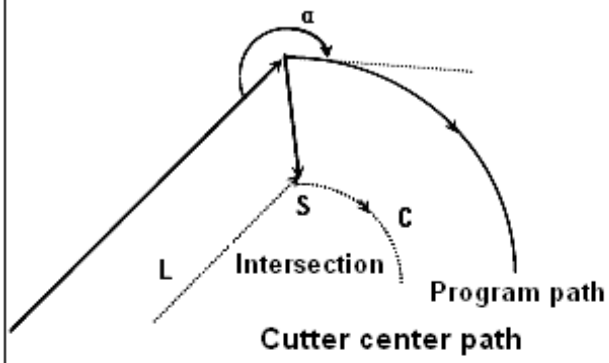
In compensation mode, if two or over two non-moving instructions are not consecutively designated (auxiliary function, pause, etc.), the compensation mode will be executed correctly. Otherwise, the part may be excessively cut or insufficiently cut. In executing the compensation mode, the compensation plane should not be modified. Otherwise, the alarm signal will be sent out and cutter stopped.

(a) Move along the inner side of the corner (α ≥180°)

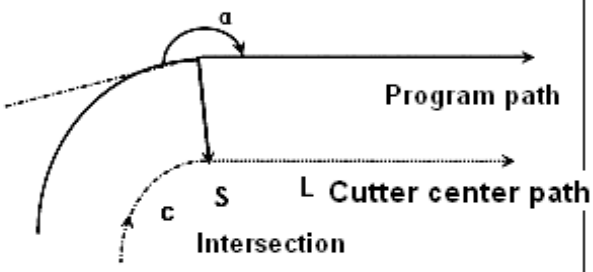
(i) : beeline → beeline



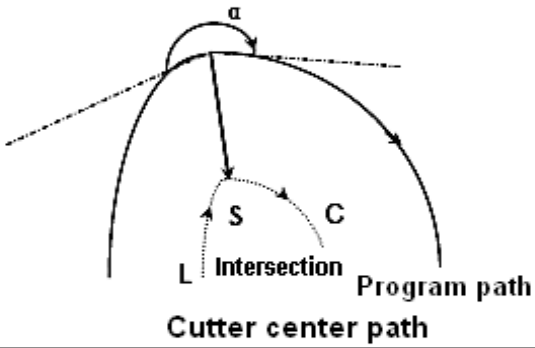
(ii) : beeline → arc-circle



(iii) : arc-circle → beeline

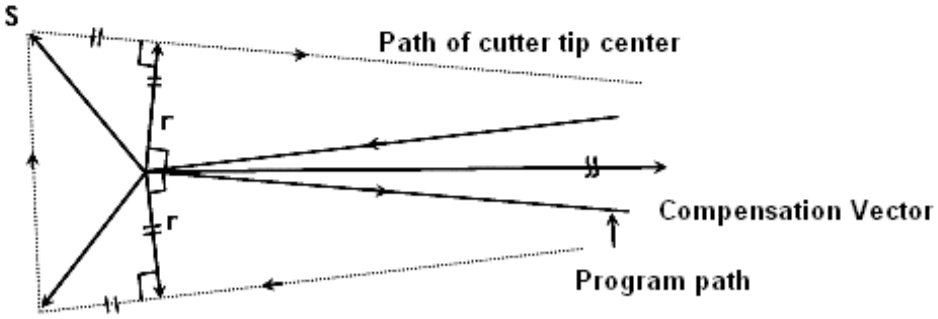


(iv) : arc-circle → arc-circle



(v) Machining for inner side of less than 1° and compensation vector amplifying.

(i) beeline → beeline



Take the following circumstances into consideration with the same method:

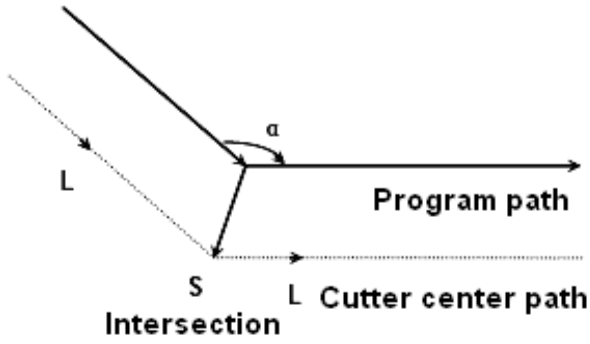
(ii) arc-circle → beeline

(iii) beeline → arc-circle

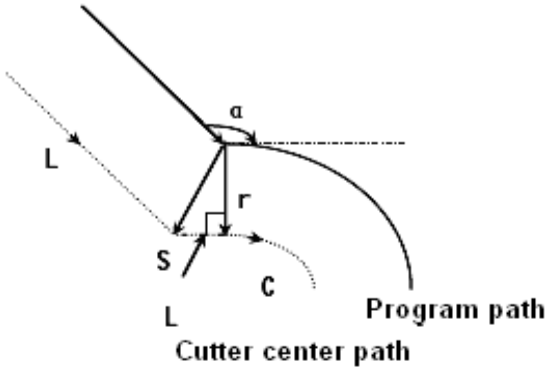
(iv) arc-circle → arc-circle

(b) Move along the outer sider of the corner which is an obtuse angle (180° > α ≥90°)

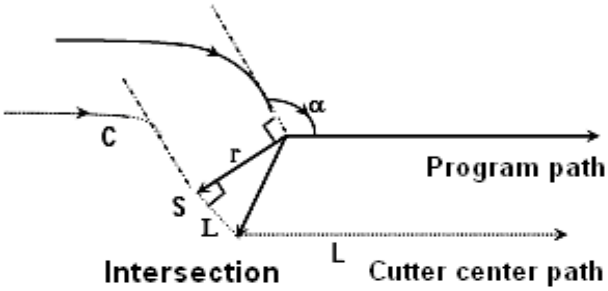
(i) : beeline →beeline



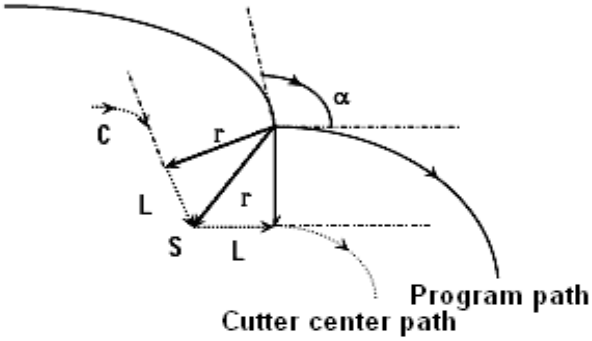
(ii) : beeline → arc-circle



(iii) : arc-circle →beeline

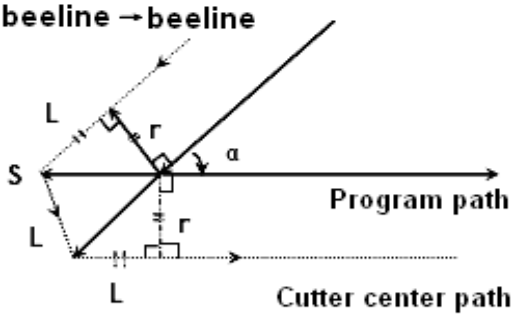


(IV) : arc-circle → arc-circle

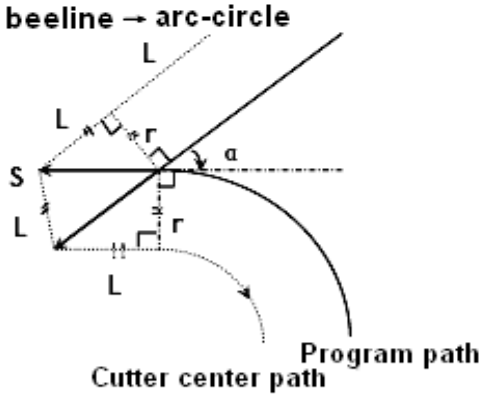


(c) Move along the outer sider of the corner which is an acute angle (α <90°)

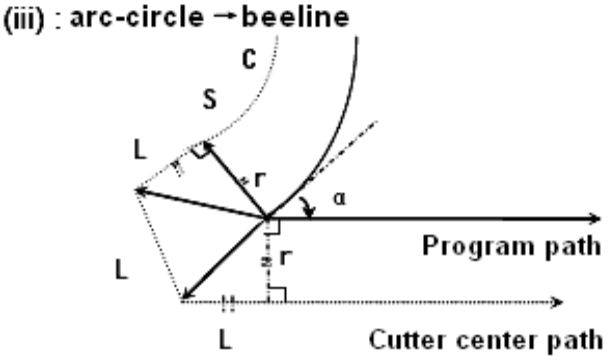
(i) : beeline →beeline



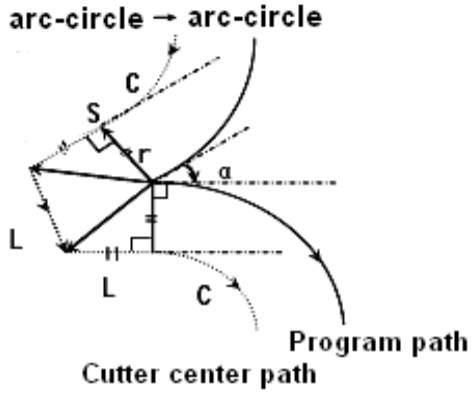
(ii) : beeline → arc-circle



(iii) : arc-circle →beeline

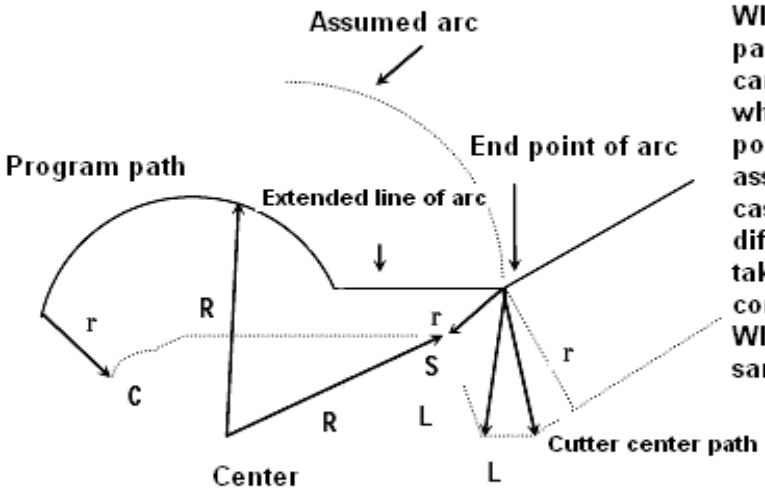


(iv) : arc-circle → arc-circle



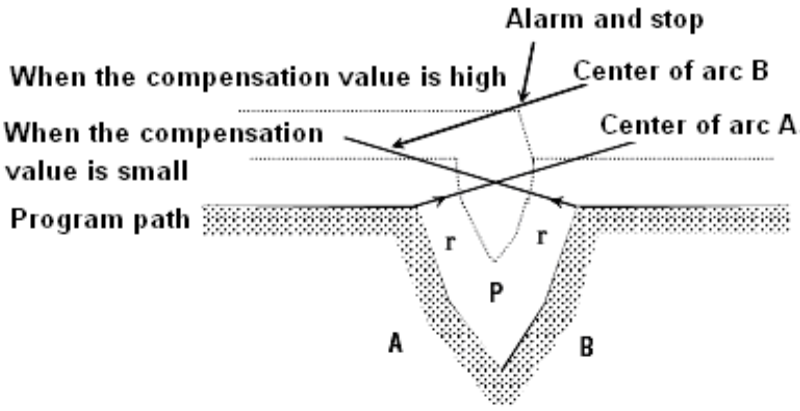
(d) Special cases

(i) The end point of program arc is not on the arc



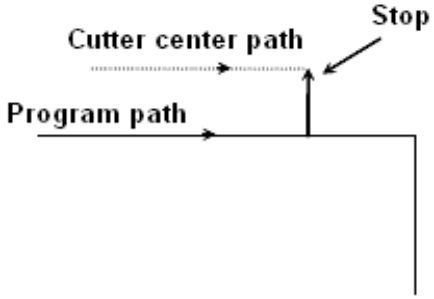
When the programming arc doesn't pass the end point, an extended line can be shown as in the left figure, which assumes an arc passes its end point. The compensation takes the assumed arc as its vector. In this case, the formed cutter center path is different from the offset path that takes the extended line into consideration.
When it moves from arc-arc, the same consideration can be taken.

(ii) No intersection is available



In the left figure, when cutter radius is a small value, the compensation path of arc will have intersection. However, when the radius value becomes greater, the intersection may not exist. The cutter will stop at the end point of the previous program segment and the system will display the alarm information.

(iii) The center of arc is coincide with the start point or end point



In left figure, the system will display the alarm information and the cutter will stop at the end point of the previous program segment .

(G41)

N5 G01 X1000;

N6 G02 X1000 I0 J0;

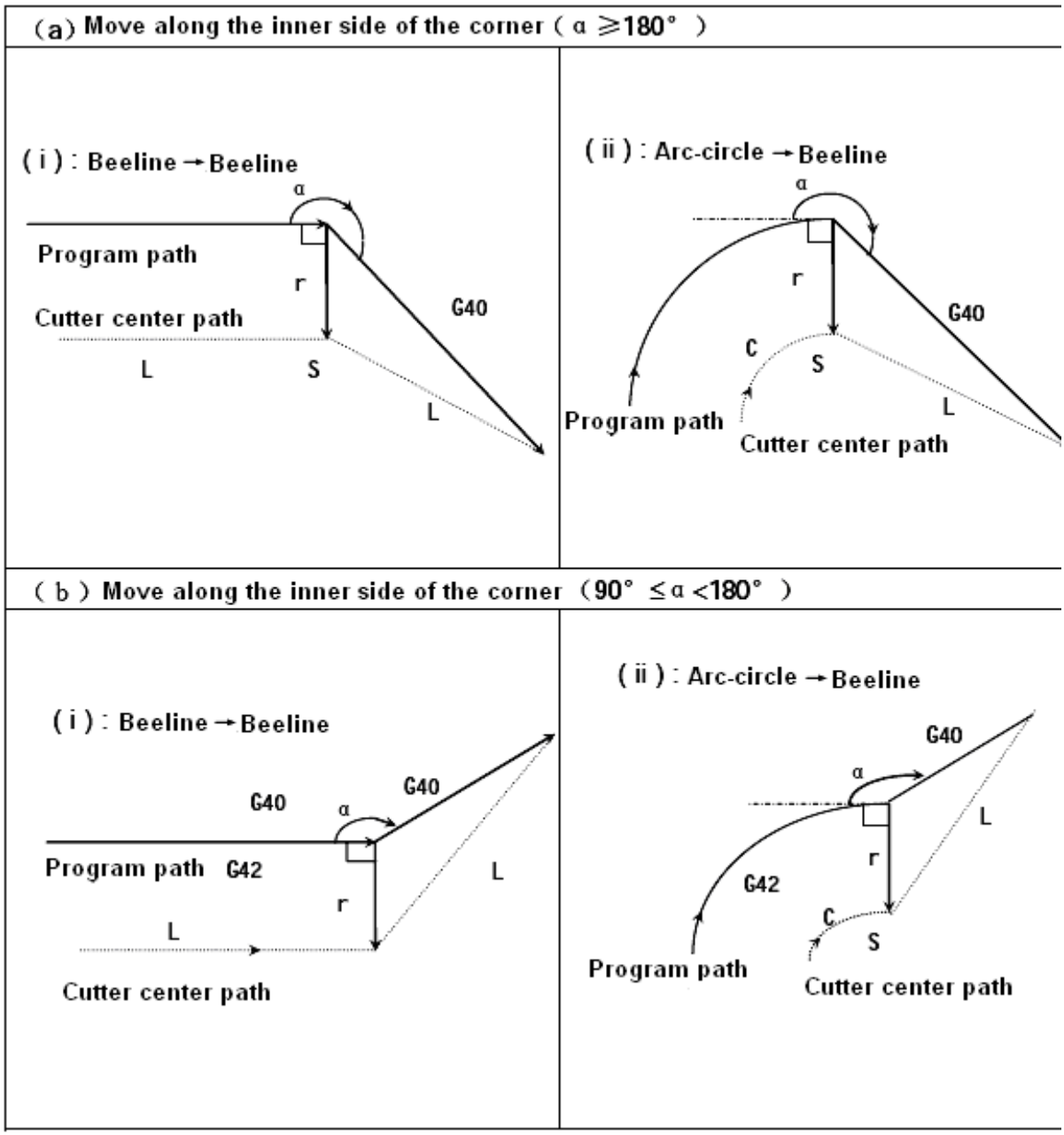
N7 G03 Y-1000 J-1000.;

d. Compensation Mode

In compensation mode, when the program segment satisfying any of the following conditions, the System will enter the compensation cancel mode. The action of this program segment is called "compensation cancel".

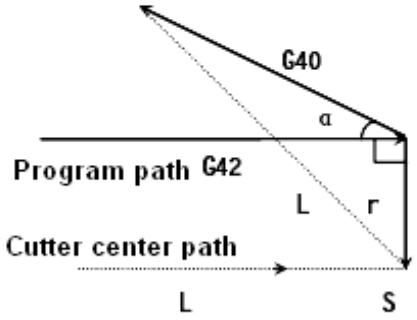
- Instruction G40
- The number of cutter radius compensation is 0.

When the compensation cancel mode is executed, the instructions for arc (G03 and G02) can't be used. Otherwise, the instruction arc will generate alarm (P/S34) and cutter will be stopped.



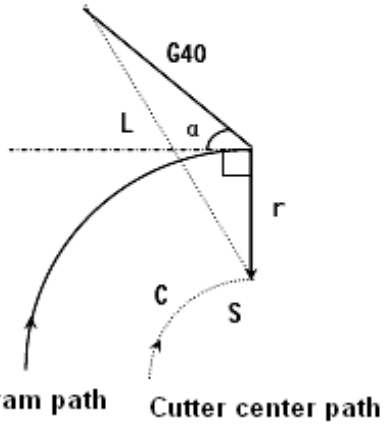
(c) Move along the outer side of the corner which is an acute angle (α < 90°)

(i) : beeline → beeline



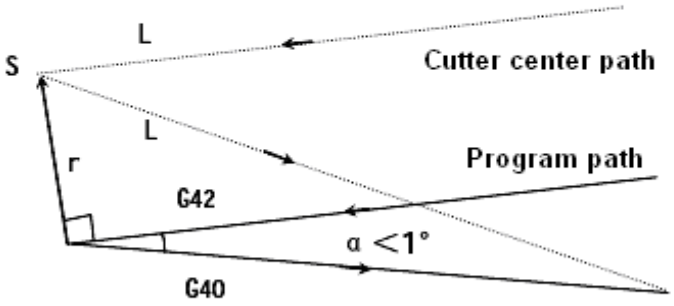
Cutter center path

(ii) : arc-circle → beeline



(d) Move along the outer side of the corner which is an acuter angle less than 1° (α <1° =

beeline → beeline

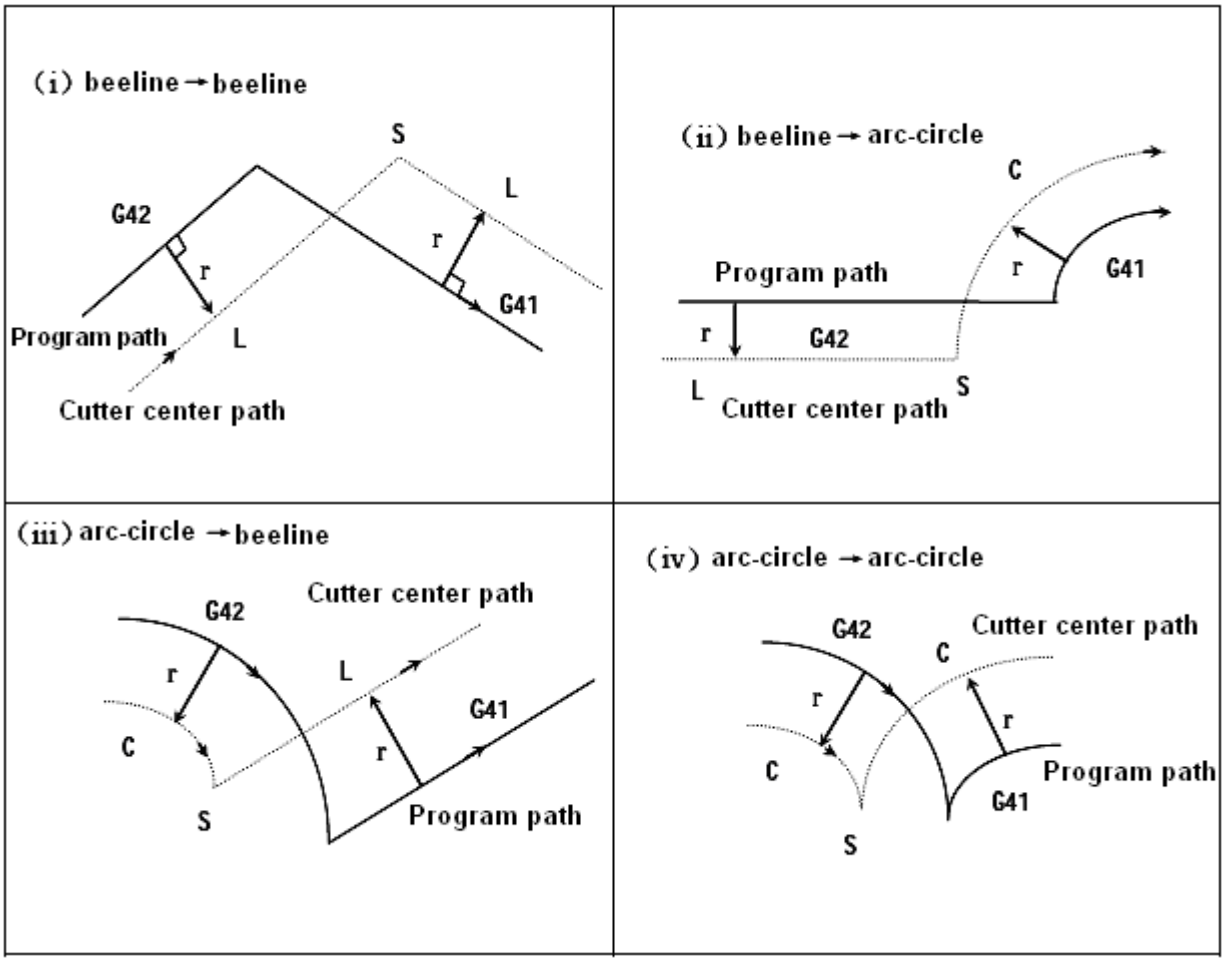


e. Change the compensation direction in the compensation mode

The G code (G41 and G42) for cutter radius compensation determines the compensation direction. The symbols of compensation are described as follows:

compensation symbol / G Code		
G41	left side compensation	right side compensation
G42	right side compensation	left side compensation

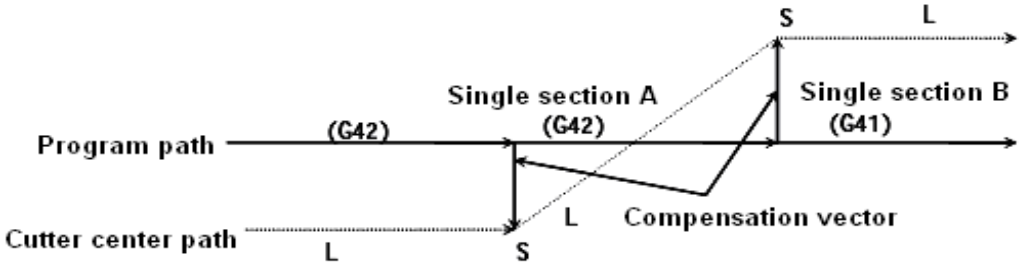
In special cases, the compensation direction can be modified in the compensation mode. However, such modification should not be executed in the start-up program segment and its follow-up program segments. Once the compensation direction is changed, the concept of inner and outer sides becomes ineffective. It is assumed the following compensation are positive values.

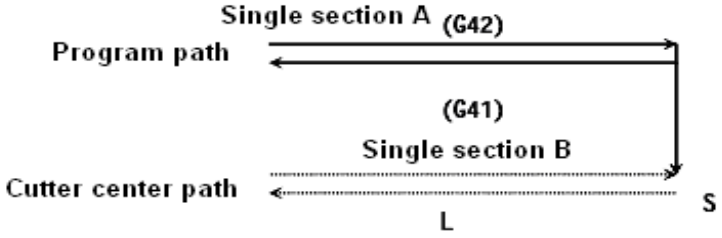


➢ When the compensation is carried out normally and there is no intersection

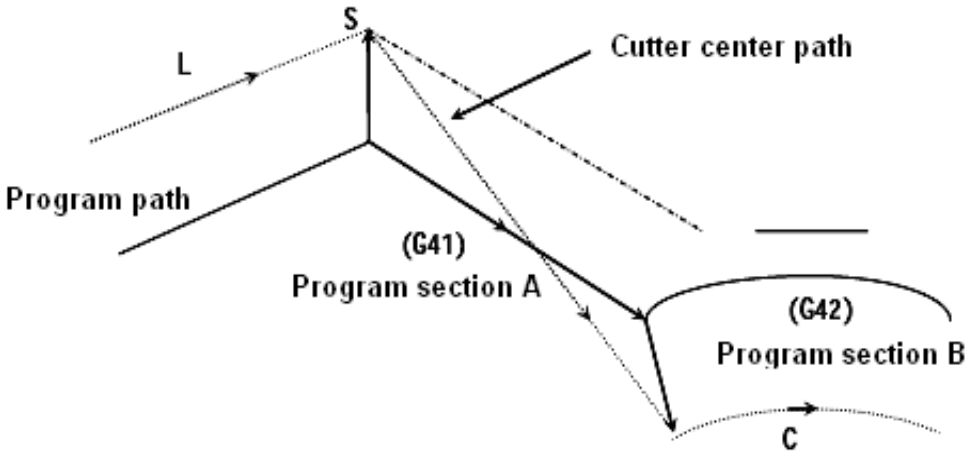
When G41 and G42 are used for changing the offset direction from program segment A to B, if the intersection of the compensation path is not needed, the vector can be made to be vertical to the program segment B from B's start point.

✧ linear----linear

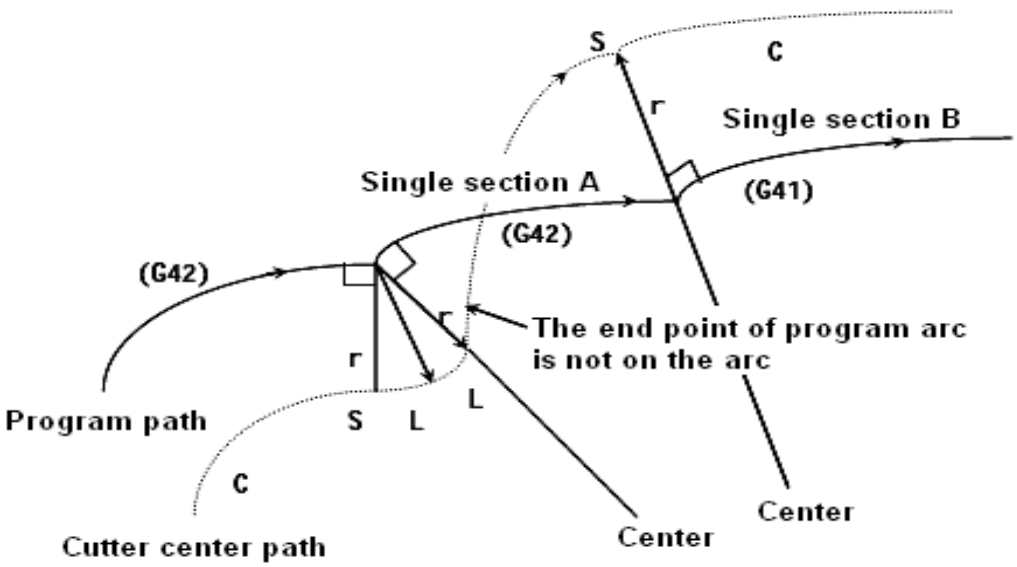




✧ linear----arc

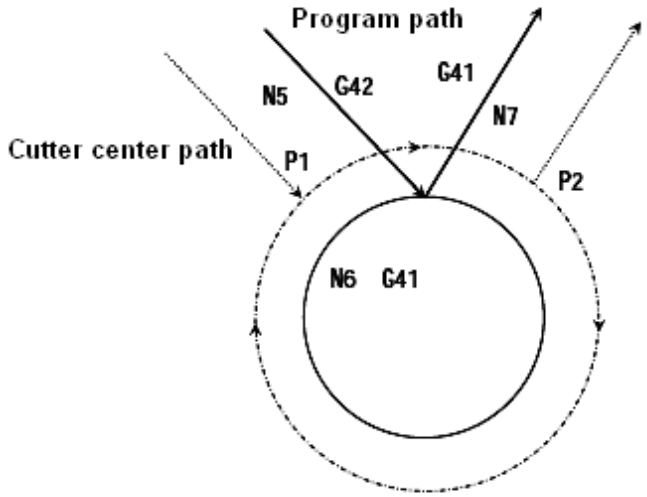


✧ arc----arc



➢ When the cutter center path for cutter radius compensation is more than one circle in length

Normally, this phenomenon won't occur. However, when G41 and G42 are modified, or I, J and K are used to instruct G40, the above situation may appear.



(G42)

N5 G02G91X5000Y-7000;

N6 G41G02J-5000;

N7 G42G01X5000Y7000;

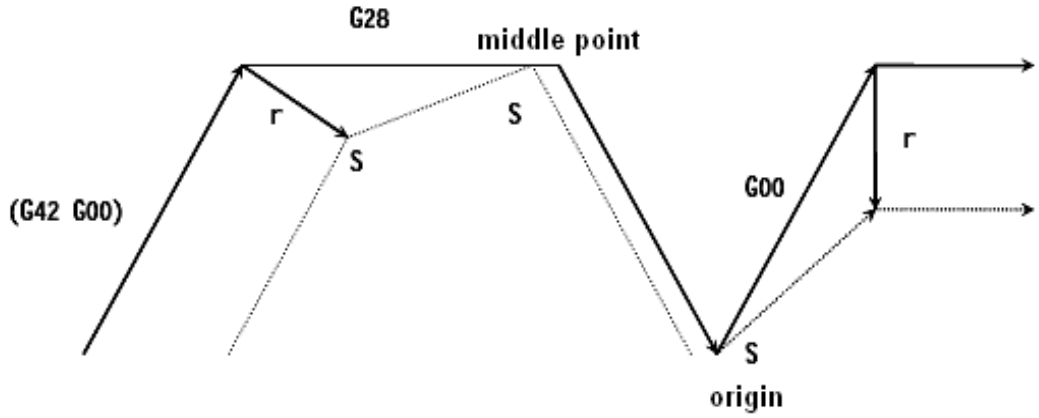
At the time, the cutter center path is not shaped as a circle, but a section of arc from P1 to P2.
In some cases, the alarm signal may be sent out because of the interference inspection. If it is expected that the cutter moves along the path of a full circle, the instructions must be executed segment by segment.

f. Temporary compensation cancel

In compensation mode, if the following instructions are executed, the compensation will be temporarily cancelled. After that, the System will automatically resume the compensation mode. For details of this operation, please refer to descriptions on compensation cancel and compensation start.

- G28 automatically returns to reference point

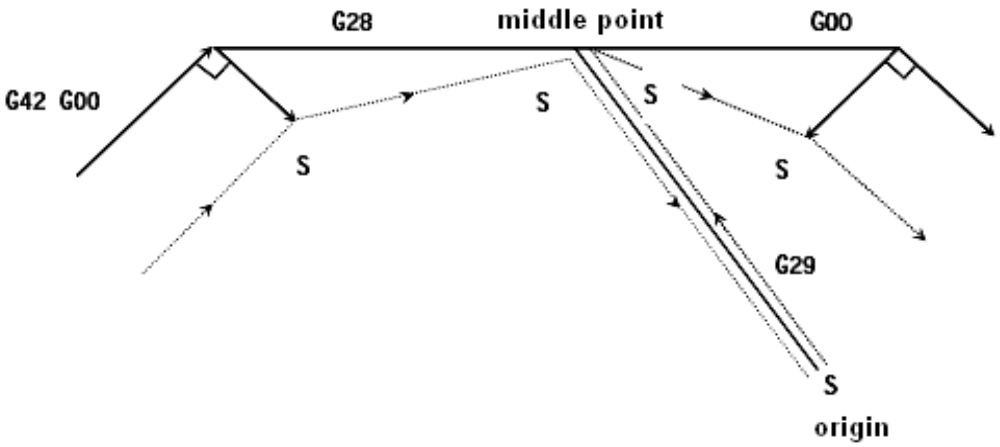
In compensation mode, if the instruction G28 is executed, the compensation will be cancelled at the middle point. The compensation mode will be automatically resumed after returning to the reference point.



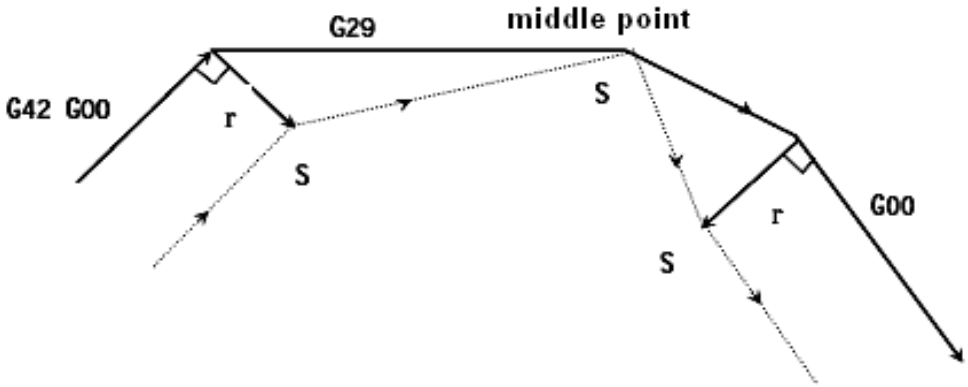
- G29 automatically returns from the reference origin

In compensation mode, if the instruction G29 is executed, the compensation will be cancelled at the middle point. The compensation mode will be automatically resumed in the next program segment.

When instruction is immediately executed after G28.



When instruction is not immediately executed after G28.

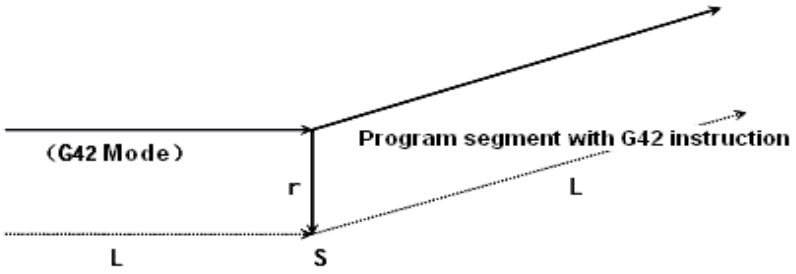


g. G code for cutter radius compensation in compensation mode

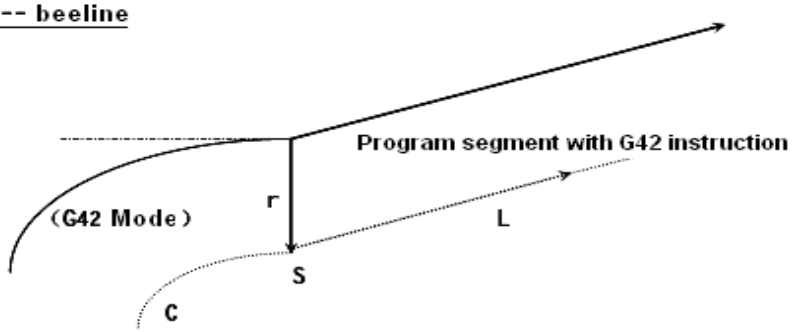
In compensation mode, when the G code (G41 and G42) for cutter radius compensation is designated, there will be a vector vertical to the previous program segment and relative to the moving direction. This vector is irrelevant to the machining inner and outer sides. However, if this G code is designated in the arc instructions, the correct arc can't be obtained.

If the cutter radius compensation G (G41 and G42) changes its compensation direction, please refer to (5).

beeline-----beeline



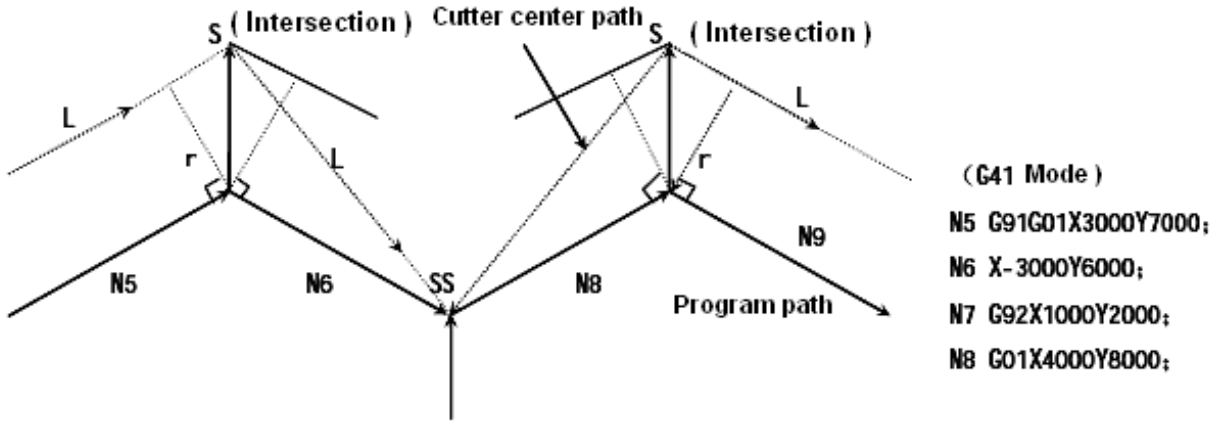
arc-circle ----- beeline



h. Instruction temporarily cancelling compensation vector

In compensation mode, if G92 (absolute coordinate programming) is designated, the compensation vector will be temporarily cancelled. After that, this vector will be automatically resumed.

At the time, unlike the compensation mode, the cutter will move from the intersection to the point which cancels the compensation vector. Once the compensation mode is resumed, the cutter will directly move to the intersection.



Note: SS indicates the point where the single segment mode cutter stops twice.

i. Program segment where cuter doesn't move

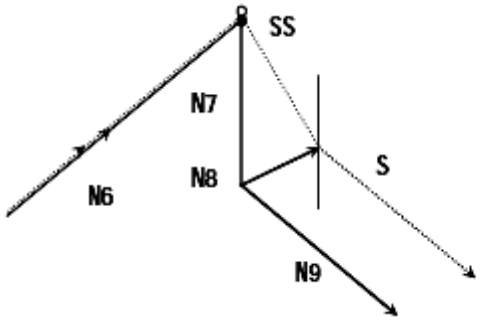
In the following program segments, the cutter won't move. In these segments, the cutter won't move even if there is an intersection for cutter radius compensation mode.

(1)M05：..... M Code input
(2)S21：..... S Code input
(3)G04 X10000：..... pause
(4)(G17)Z100：no movement instruction on the compensation plane
(5)G90：..... Only G code is available.
(6)G01 G91 X0：.....Movement is zero.

} Not move.

- Instruction for compensation start

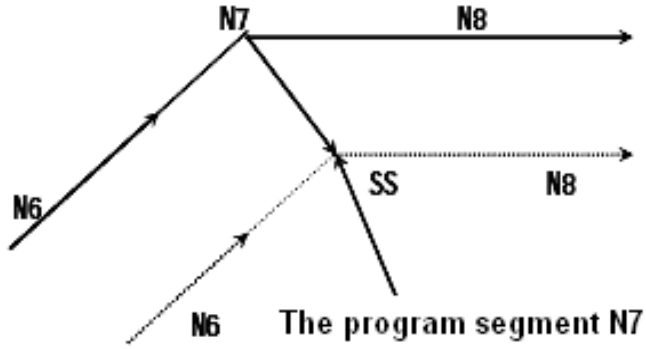
If the instruction for compensation start is executed without the movement of cutter, no compensation vector will be generated.




```
G40 G91 . . . . . .
.
.
.
N6 X1000.0 Y1000.0;
N7 G41 X0 ;
N8 X-1000.0 ;
```

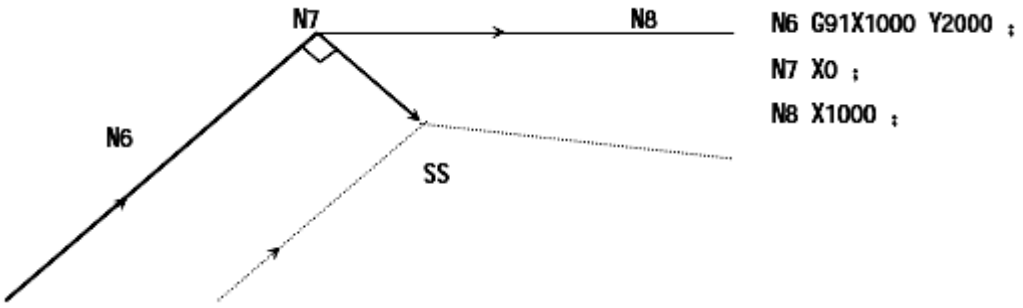
- Instruction for compensation mode

In compensation mode, if only the instruction for the program segment, which does not move the cutter, is executed, the vector and the cutter center path will remain unchanged as the time without this program segment. (Please refer to (3) for compensation mode) at the time, the program segment for cutter moving is executed at the stop point of single program segment.

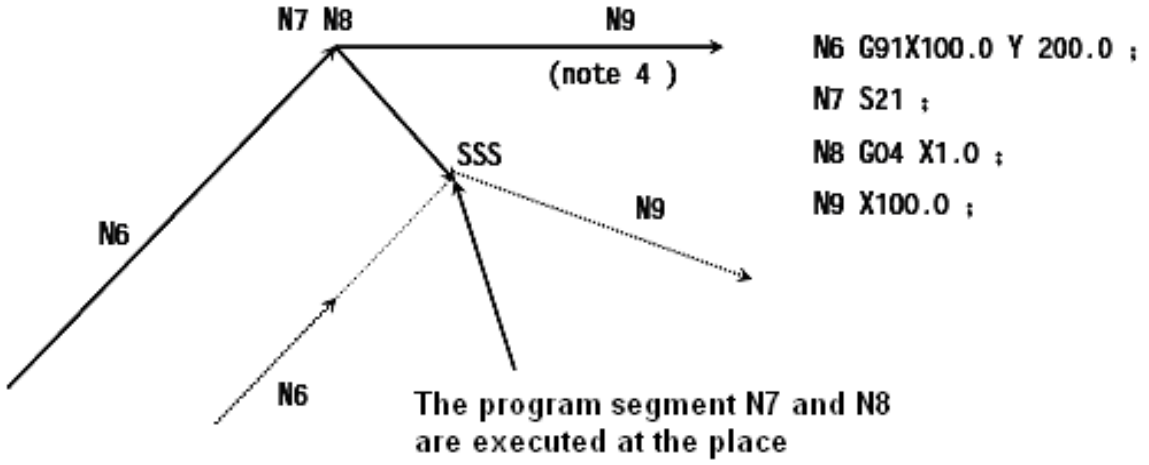



```
N6 G91X100.0 Y200.0 ;
N7 G04 X100.0 ;
N8 X100.0 ;
```

However, when the movement of the program segment is zero, even if only one program segment is designated, the cutter will move like the time there is no movement instruction. For details, please refer the following descriptions.



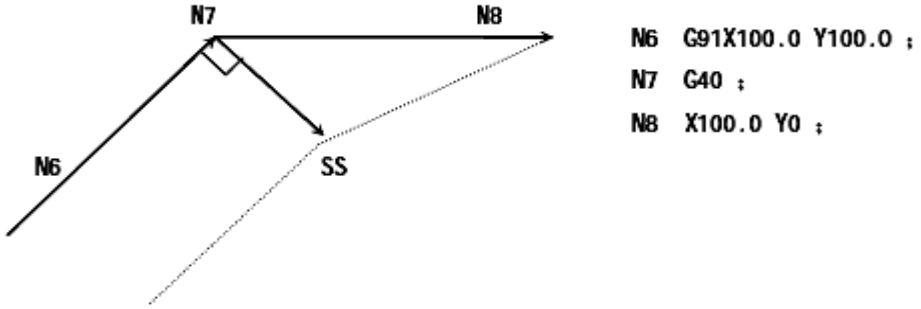
Two program segments without cutter movement can't be executed consecutively. If executed in that way, a vector, which takes the length as the compensation value and whose direction is vertical to the movement direction of the previous program segment, will be generated. This will lead to over-cutting.



Note: SSS indicates the program segments are used for operating the cutter thrice.

- Instruction at the same time as compensation cancel

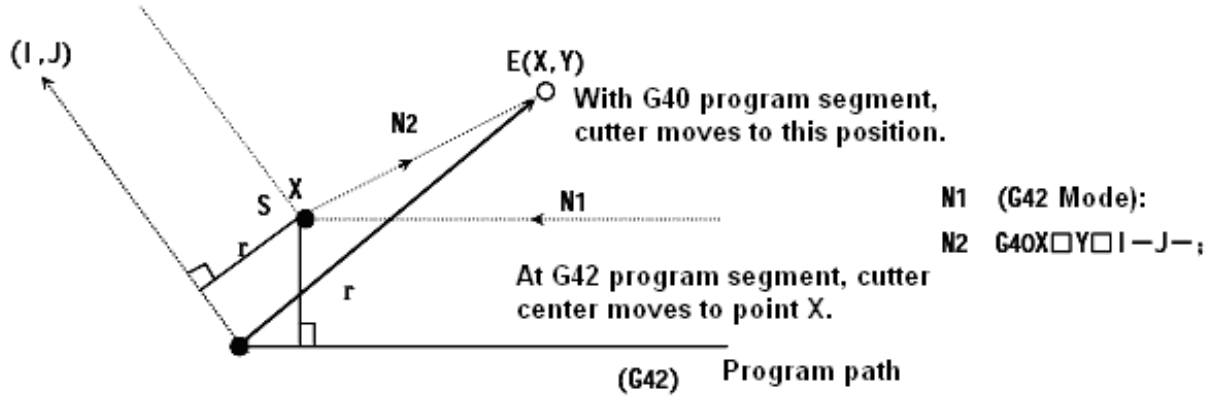
When the program segment is executed at the same time as compensation cancel but without cutter movement, a vector, which takes the length as the compensation value and whose direction is vertical to the movement direction of the previous program segment, will be generated. This vector will be cancelled at the next movement instruction.



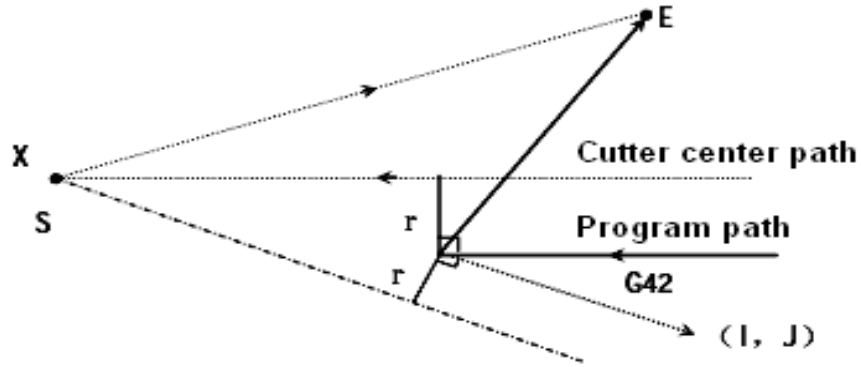
j. On the compensation plane, this program segment include G40 and I—J—K instructions.

- Previous program segment as G41 or G42

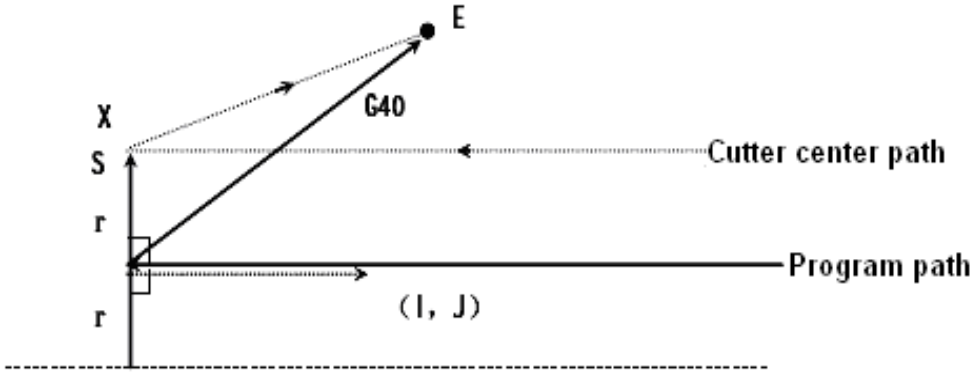
At the time, suppose that CNC sends out the instruction that a movement along the direction of I, J or K is made from the previous program segment.



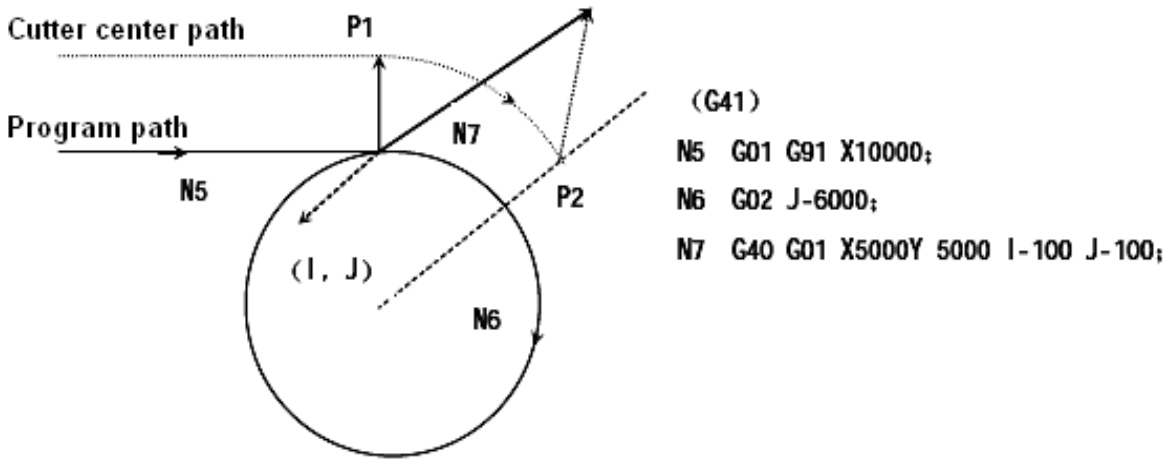
Note: The obtaining of cutter intersection by CNC is irrelevant to the inner and outer sides of the designated machining.



When the intersection can't be obtained, the end point cutter of the previous program segment moves to the position vertical to the previous program segment.



- Cutter center path is longer than a circle.



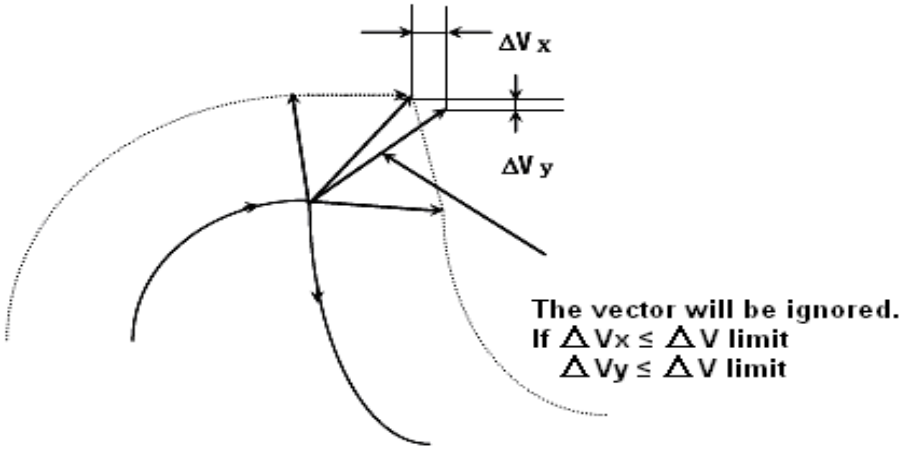
In above figure, the cutter center path doesn't move along the circle, but along the arc from P1 to P2.

In some cases, the alarm signal (P/S41)may be sent out because of the interference inspection. The related explanation will be followed up. (If it is expected to move along the circle, the arc instructions must be executed segment by segment.)

k. Corner Movement

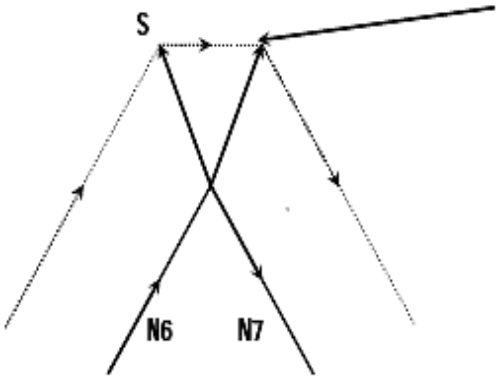
If more than two vectors are generated at the end of the program segment, in other words, the cutter moves from one vector to another, this movement is called corner movement.

If these vectors almost have the same value, the corner movement will not be executed. The latter vector can be ignored.



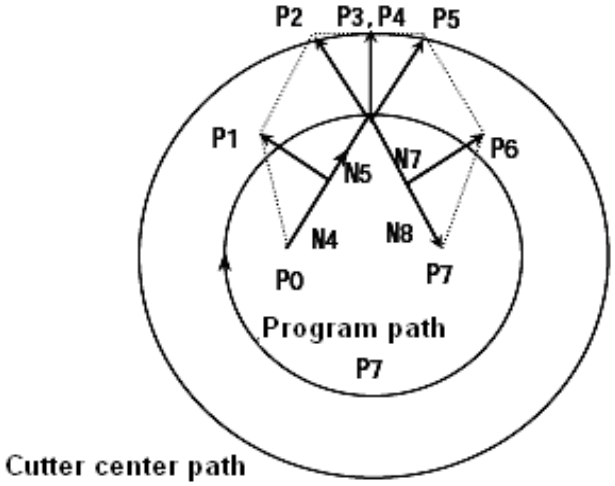
If △VX≤△V limit and △VZ≤ △V limit, the latter vector will be ignored. △V limit uses the parameter.

If these vectors are inconsistent, a movement along the corner will be generated. This movement belongs the latter program segment.



These movements belong to the program segment N7. Therefore, the feedrate equals that of the program segment N7. If program segment N7 is of G00 mode, the cutter will move at the fast feedrate. If of G01, G02 and G03 modes, it will move at the cutting feedrate.

However, if the path of the next program segment exceeds the length of a half circle, the abovementioned process will not be carried out. The reasons can be seen as follows:



N4 G41G91X1500Y200(
N5 X1500Y2000;
N6 G02J-6000;
N7 G01X1500Y-2000;
N8 G40X1500Y-2000;

If the vector is not ignored, the cutter path can be described as follows:

P0→P1→P2→P3（arc-circle）→P4→P5→P6→P7

However, if the distance between P2 and P3 is ignored, P3 will be ignored. The cutter path can be described as follows:

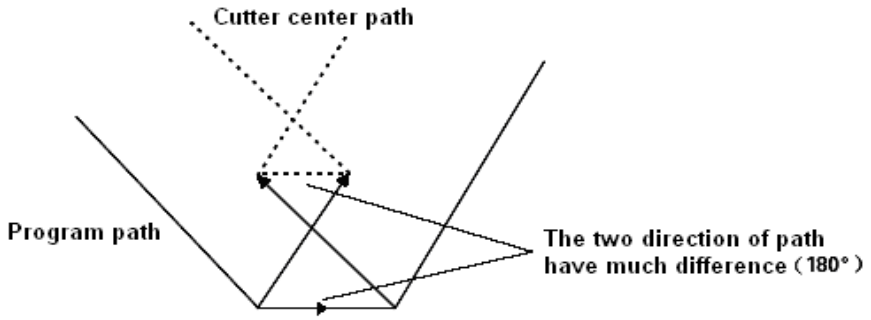
P0→P1→P2→P4→P5→P6→P7 The arc cutting of program segment N6 is ignored.

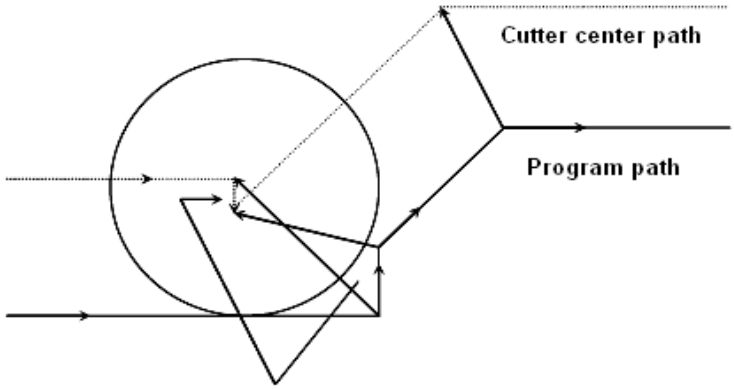
I. Interference inspection

The excessive cutting of cutter is called "interference". The interference mode can examine the whether the cutter cut excessively. However, this function can't inspect all the interferences. The interference inspection mode can be activated even if there is no excessive cutting.

- **Preconditions of interference:**
  - The direction of cutter path differs from that of program path. (The inclination is between 90° and 270°).
  - When the arc machining is being carried out, there should a substantial difference between the inclination of the start point and end point of cutter center path and that of the start point and end point of the program path.

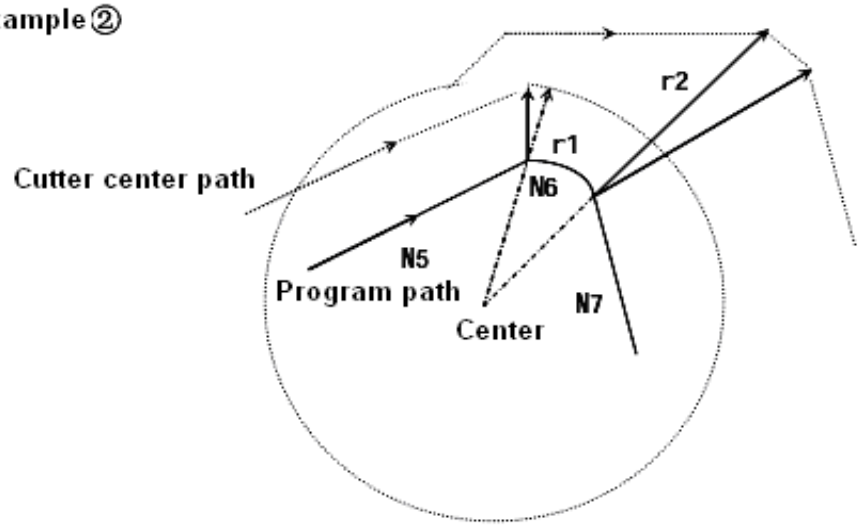
example ①





The two direction of path have much difference（180° ）

example②



(G41)

N5 G01 G91 X8000 Y2000 D01;

N6 G02 Y-1600 X3200 12000 J-8000 D02;

N7 G01 X2000 Y-5000:

（H01 Tool radius compensation amount r1=2000）

（H02 Tool radius compensation amount r2=6000）

In above examples, the arc of program segment N6 is within the first quadrant. But after cutter compensation, the arc is located in the fourth quadrant.

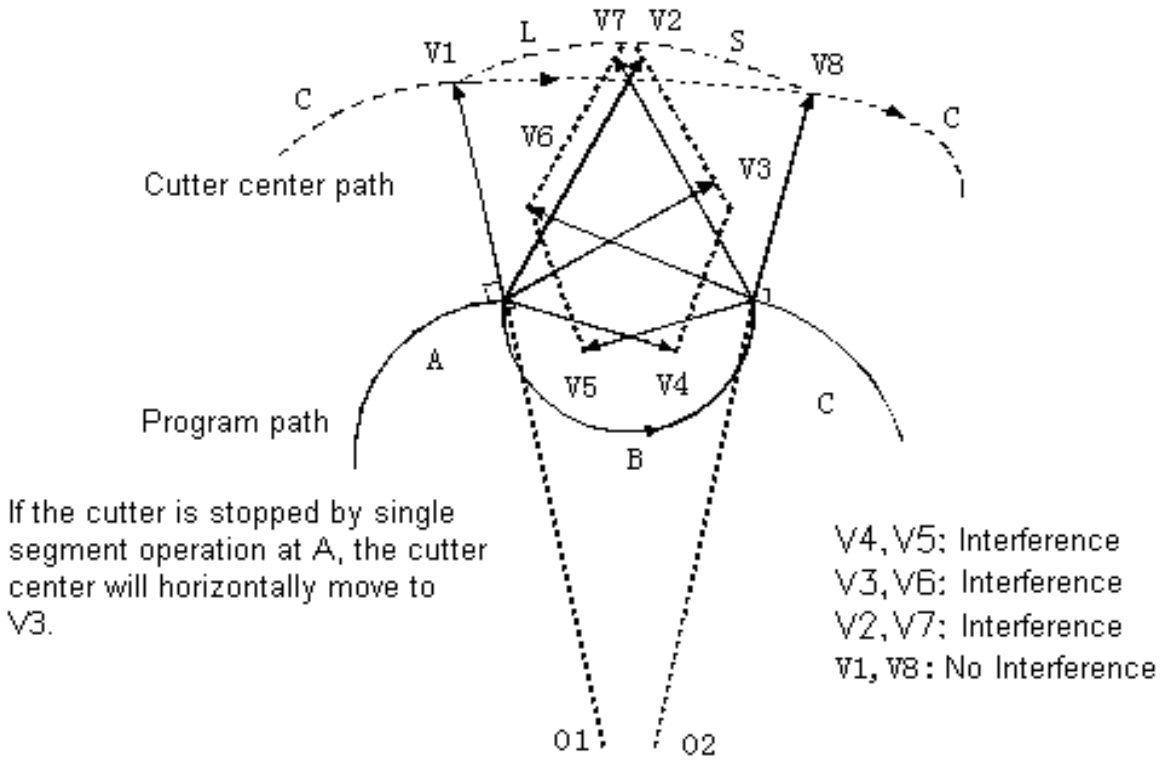
- **Pretreatment of interference**
  - ✧ Interference caused by the movement of vetor

When the program segment A, B and C for cutter compensation are executed, vector V1, V2, V3 and V4 will be generated between A and B, and vector V5, V6, V7 and V8 will generated between B and C. The closest vector should be inspected. If there is an interference, it will be automatically eliminated. If the vector to be ignored is located at the last part of the corner, the interferences can't be eliminated.

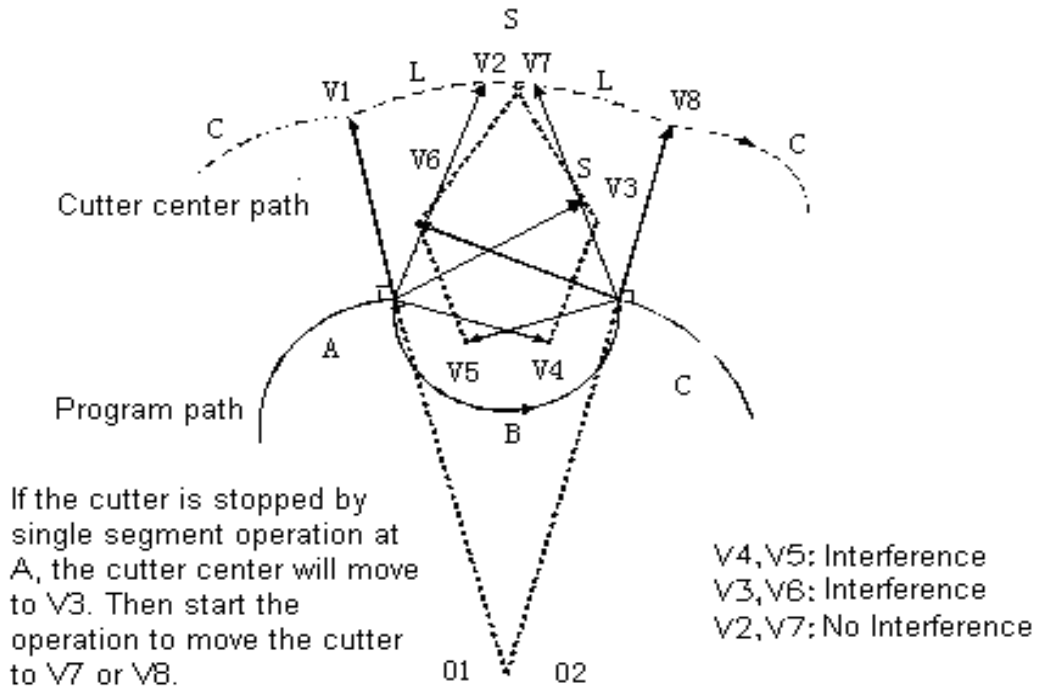
Interference inspection:
Between V4 and V5—interference—V4, V5 eliminated
Between V3 and V6—interference—V3, V6 eliminated
Between V2 and V7—interference—V2, V7 eliminated
Between V1 and V8—interference—V1, V8 can't be eliminated

In inspecting, if some vector has no interference, the follow-up vectors won't be inspected. If the program segment B is of arc movement, the vector interference will cause linear movement.

(Example 1) Cutter's linear movement from V1 to V8

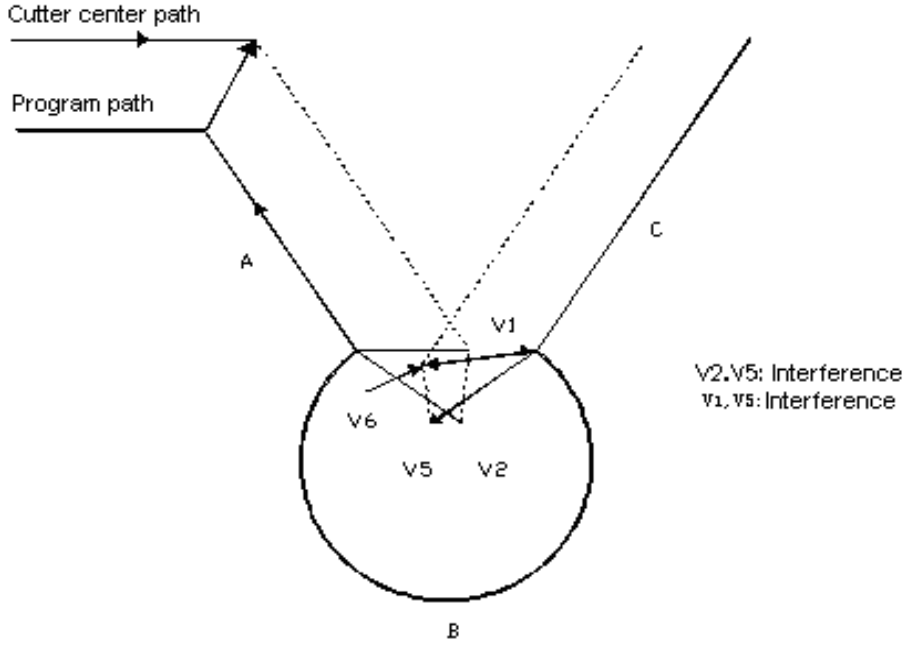


(Example 2) Cutter's linear movement is send as follows:
Cutter path: V1→V2→VY→V8



- ✧ If the interference still happens after the treatment (1), the cutter will be stopped and the alarm will be generated. If the interference happens after the treatment (1) or there is only one group of vector after inspection starts, and this vector has interference, the cutter will be stopped immediately after the previous program segment is executed, and the alarm information will be displayed (P/S41)

(If the single program segment is used for execution, the cutter will be stopped when the program segment is finished.)

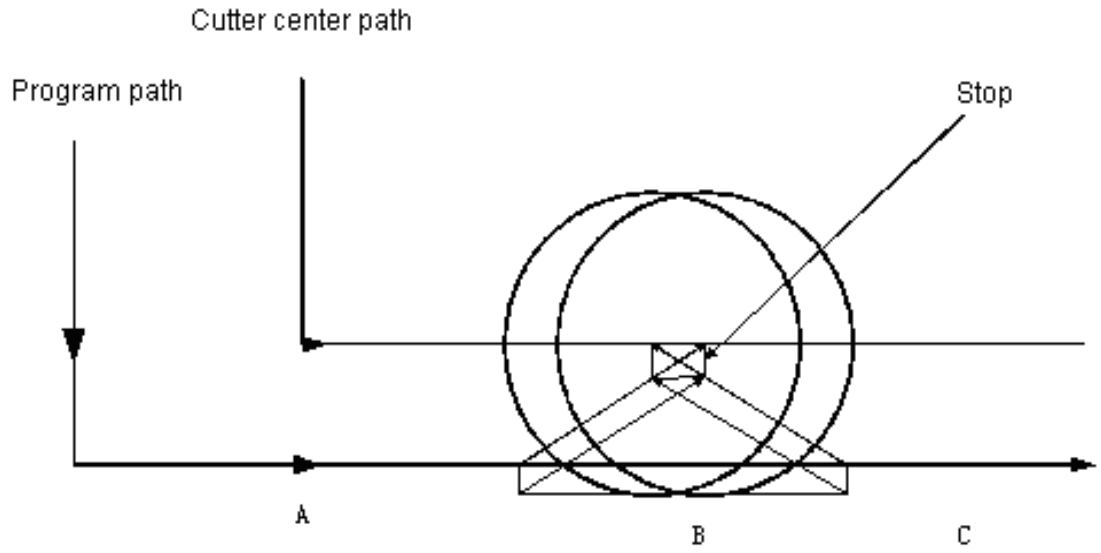


The interference ignores the vector V2 and V5. But interference will happen between the vector V1 and V6. The alarm information will be displayed and cutter stopped immediately.

- ➢ No interference actually happens., but interference inspection is performed.

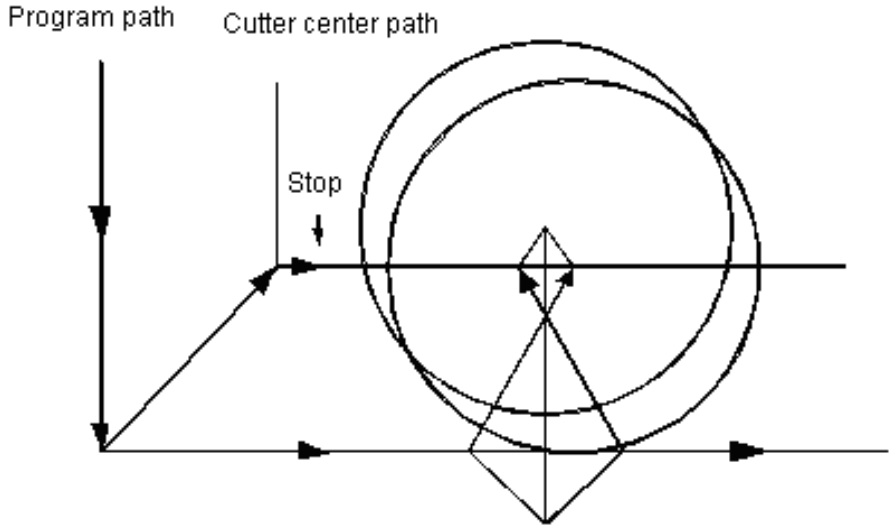
See the following example:

✧ The depth of the concave is less than the compensation value.



No interference actually happens. However, as it is the program segment B, the direction of program is opposite to the path of the radius compensation. The cutter will be stopped and alarm information displayed.

✧ The depth of groove is less than the compensation value

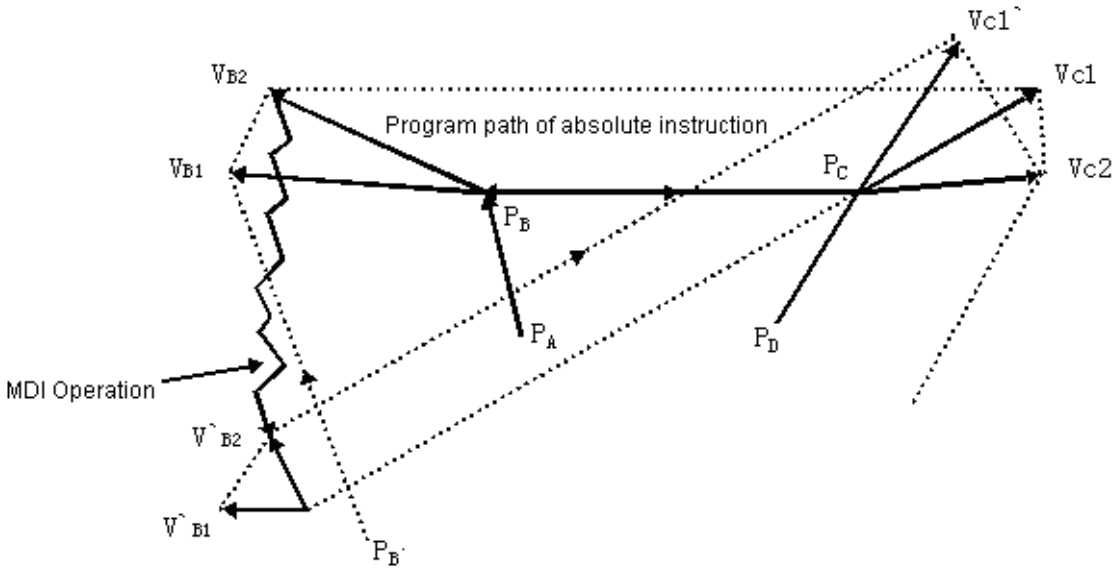


Like the example (1), the direction of cutter path is opposite to that of program path

m.Compensation can't be conducted by entering instruction from MDI

During the automatic running of the NC program made by absolute instructions, when the single segment is used for temporary stop, after the MDI operation is interpolated and the auto running is started again, the cutter path can be described as follows:

At the time, the vector of the next program segment is transmitted, and other vectors will be generated according to the next two program segments. Therefore, compensation after point Pc can be performed correctly.



When point Pa, Pb and Pc are programmed with absolute instructions, the single segment will be used for stopping after the program segment is executed from Pa to Pb. The cutter is moved by inserting MDI. The vector Vb1 and Vb2 are transmitted to V ` b1 and V ` b2, and the vector Vc1 and Vc2 between program segment Pb→Pc and Pc→Pd will be re-calculated.

However, as vector Vb2 is not calculate again, the compensation after point Pc can be executed correctly.

n. Manual operation

For the manual operation in cutter tip radius compensation, please refer to the manual part in the Operation chapter.

o. If the compensation for cutter length is executed in the cutter radius compensation, the compensation for cutter radius is considered as the compensation change.

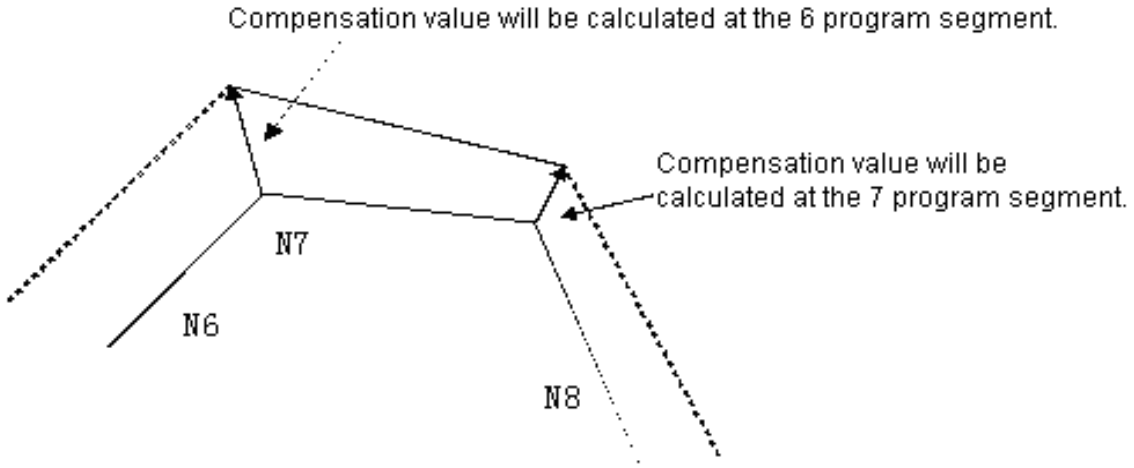
p. Precautions on compensation

Instruction compensation

D code is used for designating the number of compensation value. Once designated, H code will remain effective till another H code is designated or compensation is cancelled. In addition to designating compensation value for cutter radius, H code is also used for the value of cutter offset.

Modifying compensation

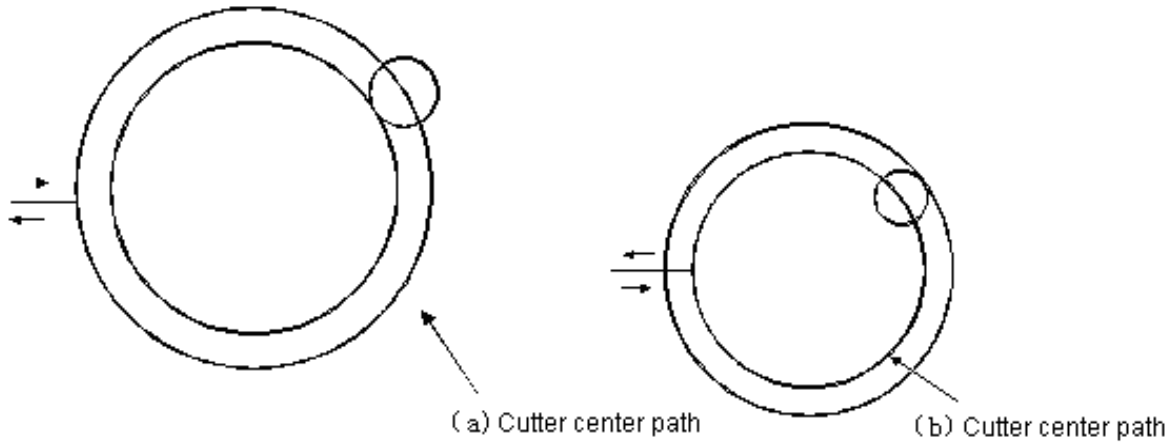
Normally, when the cutter is changed, the compensation value must be modified in the cancel mode. If the compensation value is modified in the compensation mode, the new compensation value will be calculated at the end of the program segment.



➢ Positive and negative compensations and cutter center path

If the compensation is a negative value (-), G41 and G42 in the program will be exchanged mutually. If the cutter center moves along the outer side of the workpiece, it will move along the inner side. Vice versa.

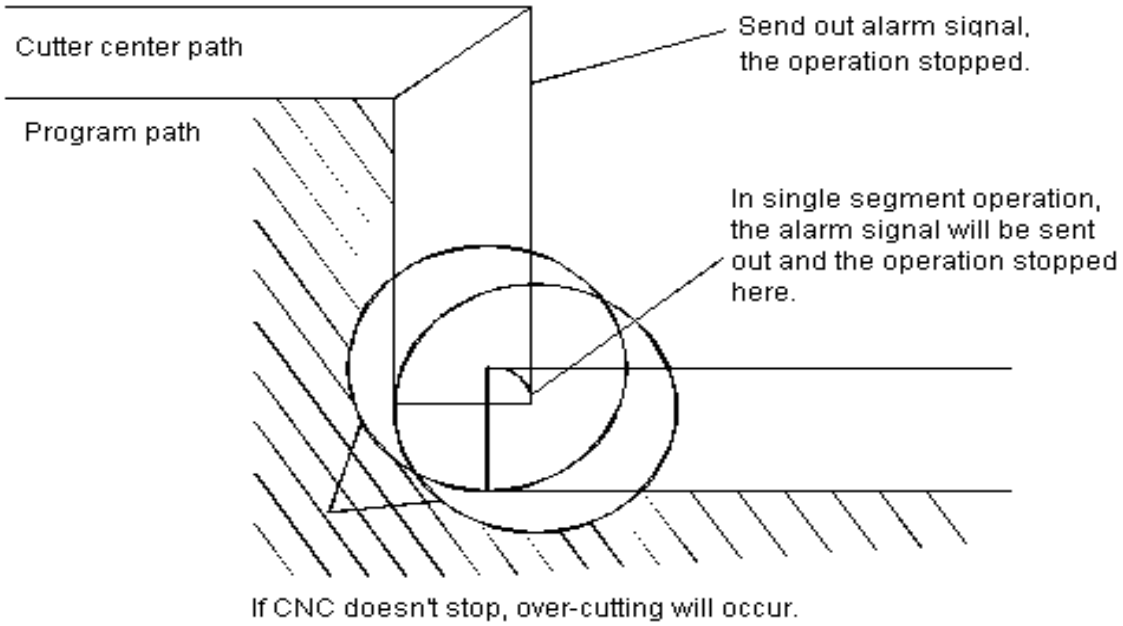
As shown in the following example, the compensation is normally set as positive in preparing the program. When the cutter path is programmed as Figure (a), if the compensation value is negative (-), the cutter center will move in a path shown in Figure (b). Vice versa. Therefore, the part can be cut into a male or female shape in the same program, and the gaps between them can be adjusted by the selecting the compensation. (Suitable for compensation start and the type A canceling.)



➢ Using cutter radius to compensate excessive cutting

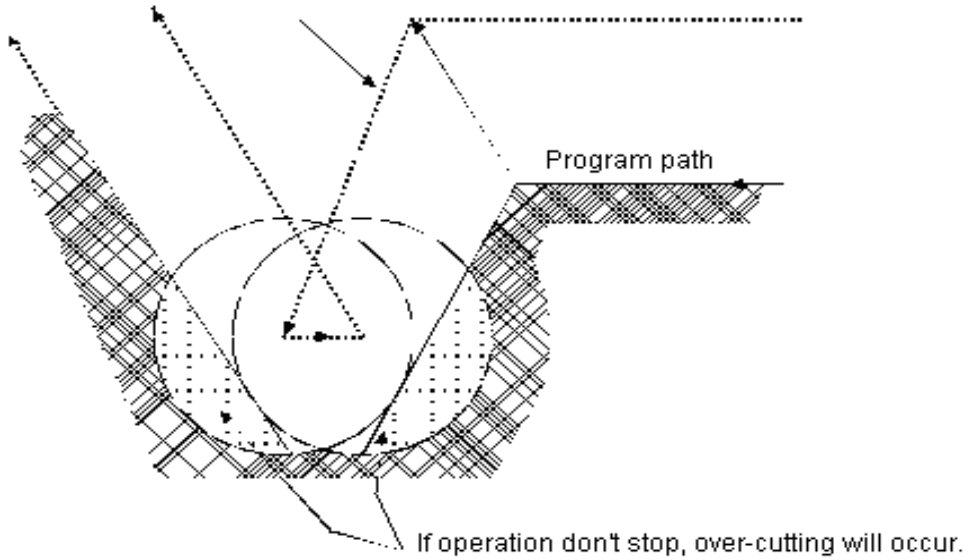
Machining with arc's inner side of small cutter radius

When the radius of corner is smaller than the cutter radius, the inner side compensation of cutter will cause over cutting, and the system will alarm. CNC will stop at the start position of the single-segment program.



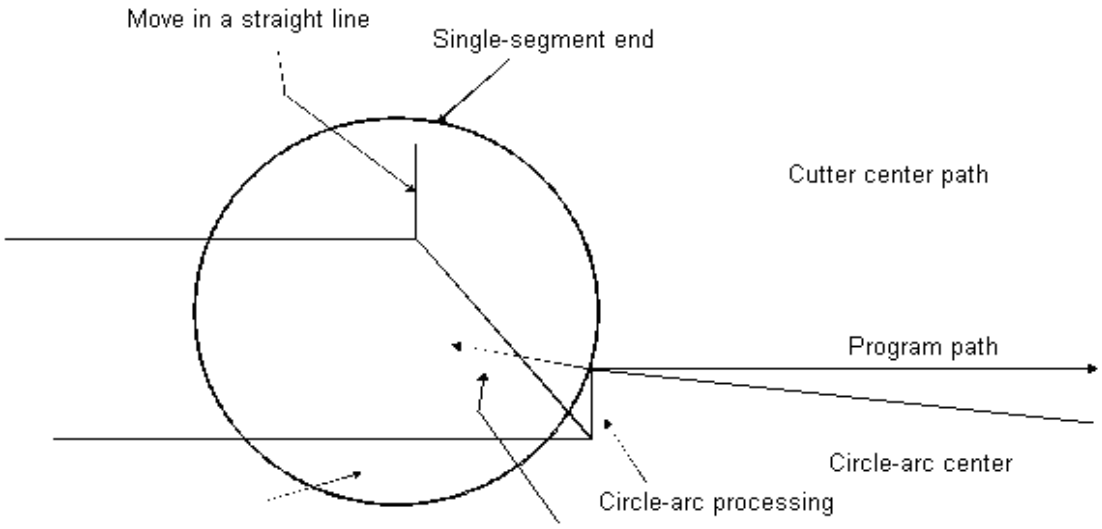
✧ Groove machining with size smaller than cutter radius

As the cutter center path is forced to move reversely to the program path due to the cutter radius compensation, over-cutting will occur.



✧ Segment-difference machining with size smaller than cutter radius

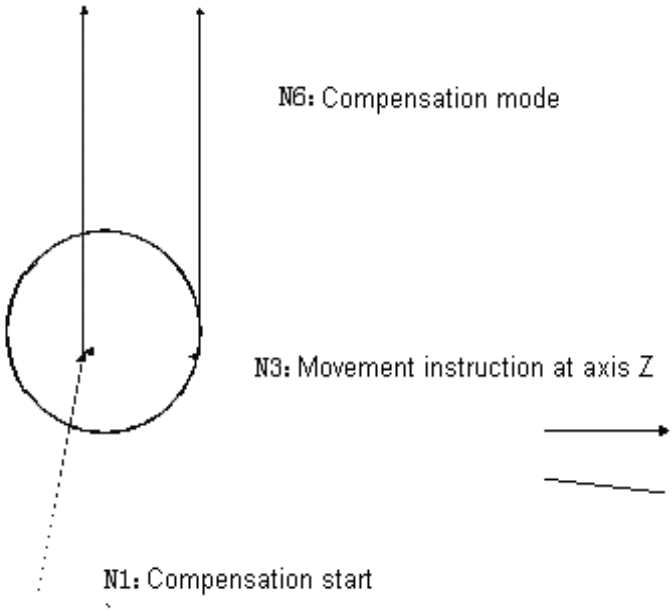
If there is segment difference smaller than the cutter radius in the program, and this segment difference is machined by the arc instruction, the cutter center path as normally compensated will have the direction opposite to that of the program. At the time, the compensated vector is ignored and the cutter moves to the second vector in a linear fashion. The execution of single-segment program stops here. If the machining is not conducted under the single-segment mode, the auto running will continue. If the segment difference is a straight line, no alarm signal will be sent out and the cutting be correct. However, the uncut part will remain.

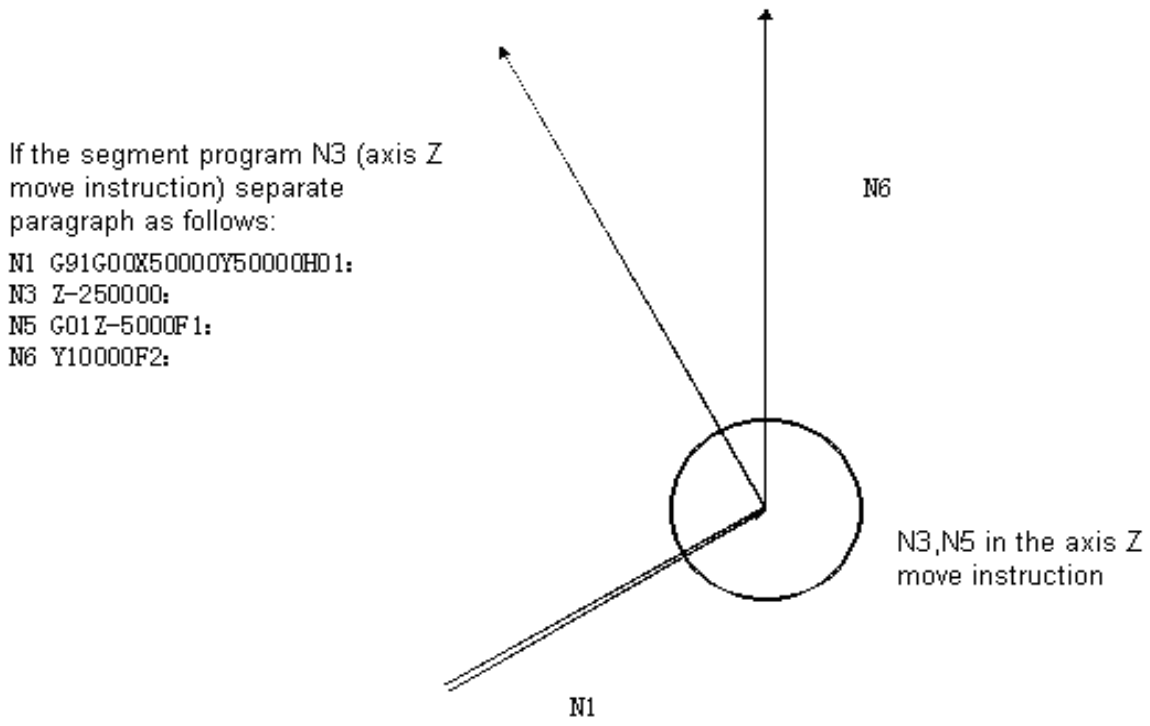


- ✧ If the initial vector of cutter is not ignored, over-cutting will occur.
- ✧ Normally, when the machining process begins, the cutter will move along axis Z some distance away from the workpiece after the cutter radius is effectively compensated. In aforesaid case, you should refer to the procedure below if the movement along axis Z is divided into fast feed and cutting feed:

```
N1 G91G00X50000Y50000H01:
N3 G01Z-30000F1:
N6 Y10000F2:
```

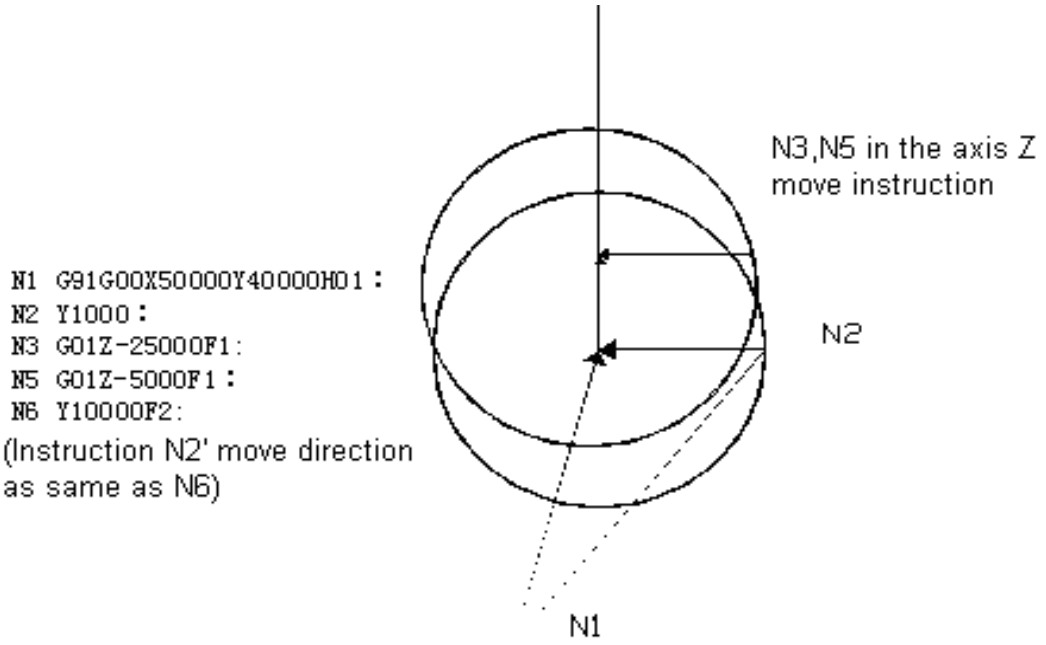
When N3 is executed, N6 enters the buffering area. Use the relationship between them to correctly execute the compensation, as shown in the right drawing.





If the selected plane doesn't include the two program segments with movement instruction, N6 can't enter the buffering area, and the cutter center path will be calculated by N1, as shown in above drawing. If the compensation vector is not calculated at compensation start, over-cutting will consequently occur. Thus the abovementioned example must be modified as follows:

When N1 is executed, program N2 and N3 will enter the buffering area. Use the relationship between N1 and N2 to execute the correct compensation.



Length Compensation G43 G44 G49

G43 G43
Z_H_ or H_
G44 G44

According to above instruction, move the end position of axis Z instruction for one more offset, and set the difference of the assumed cutter length and the actual value in machining to the offset memory. Therefore, the program doesn't need to be modified. To use cutters with different lengths, you only need to change the compensation value of the cutter.

G43, G44 designate a different direction of migration, The offset number is designated by H code.

- Migrate direction
  G43：Positive offset
  G44：Negative offset

No matter it is a absolute instruction or incremental instruction, when at G43, you should add the offset designated by H code (set in the offset memory) to coordinate value of the end point of the axis Z's movement instruction; when at G44, you should deduct the offset designated by H code. Then use the coordinate value of the calculated results as that of the end point.

When the movement of axis Z is omitted, it can be considered as the following instruction. If the offset is a positive value, instruction G43 serves as an offset moving positively, whereas instruction G44 serves as an offset moving negatively.

G43
G91 H_
G44

When the offset is a negative value, the movement is reverse.

G43 and G44 are of mode status G code, which remain effective before they meet other G code in the same group.

- **Designation of offset**

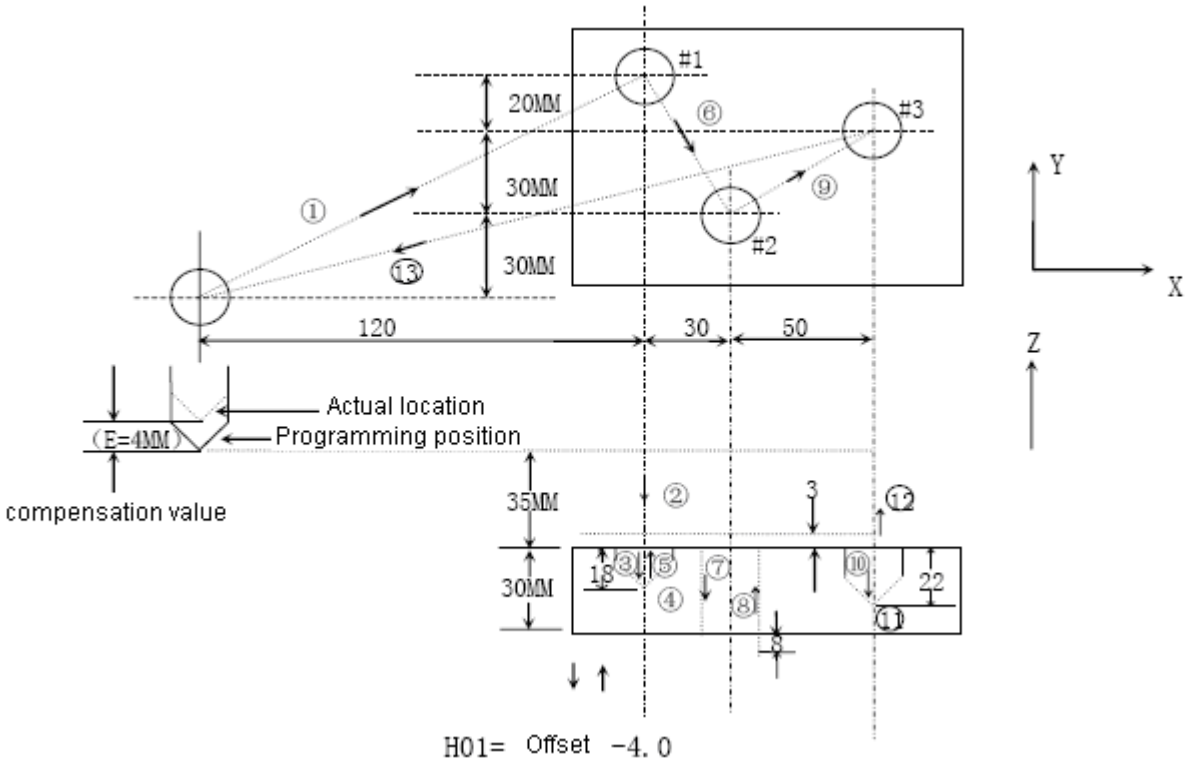
The offset number is designated by H code. The offset corresponding to the offset number is added to or deducted by the value of movement instruction at axis Z to produce the new movement instruction at axis Z. The offset number can be designated from H00-H18.

Enter cutter compensation menu, and preset the offset onto the corresponding offset number in the offset memory.

	Mm input	Inch input
Offset	0～±999.999	0～±99.9999

The offset number 00 means the corresponding offset of H00 is 0. The offset H00 corresponds can't be set.

- Cancel the cutter length compensation; Use G49 or H00 to cancel the cutter compensation. Once the instruction G49 or H00 is executed, the compensation will be cancelled immediately.
- Examples of cutter length compensation.
- Cutter length compensation (machining hole #1, #2 and #3).




```
N1 G91 G00 X120.0 Y80.0；·····（1）
N2 G43 Z-32.0 H01；·····（2）
N3 G01 Z-21.0；·····（3）
N4 G04 P2000；·····（4）
N5 G00 Z21.0；·····（5）
N6 X30.0 Y-50.0；·····（6）
N7 G01 Z-41.0；·····（7）
N8 G00 Z41.0；·····（8）
N9 X50.0 Y30.0；·····（9）
N10 G01 Z-25.0；·····（10）
N11 G04 P2000；·····（11）
N12 G00 Z57.0 H00；·····（12）
N13 X-200.0 Y-60.0；·····（13）
N14 M30；
```

Note: When the offset number is changed to modify the offset, it only means the offset becomes a new one. It does not mean that the new offset is added to the old one.

H01·····Offset 20.0
H02·····Offset 30.0
G90 G43 Z100 0 H01·····Z Moves to 120.0

G90 G43 Z100 0 H02·····Z Moves to 130.0

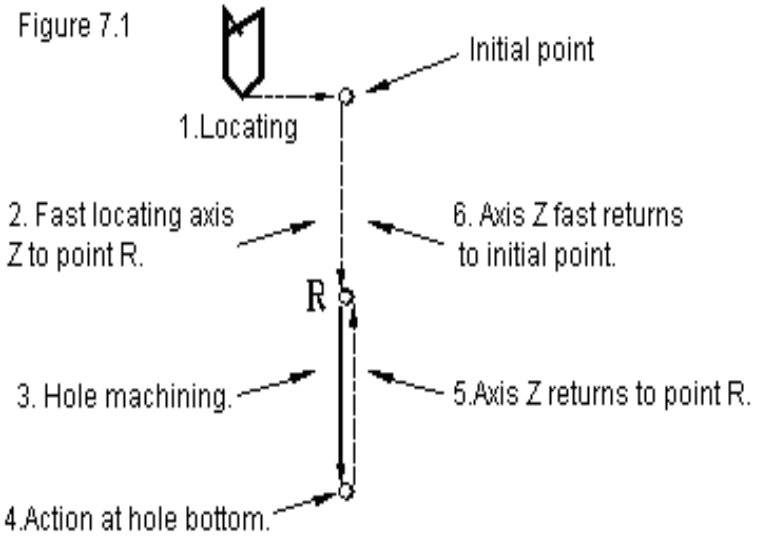
3.2.8 Hole machining cycle（G73～G89）

If the fixed cycle function for hole machining is used, the functions, which are accomplished by several program segments if other methods are involved, can be performed within one program segment. The Table 7.1 provides all fixed cycles for hole machining. Basically, to accomplish one fixed hole machining cycle, the following six procedures should be performed:

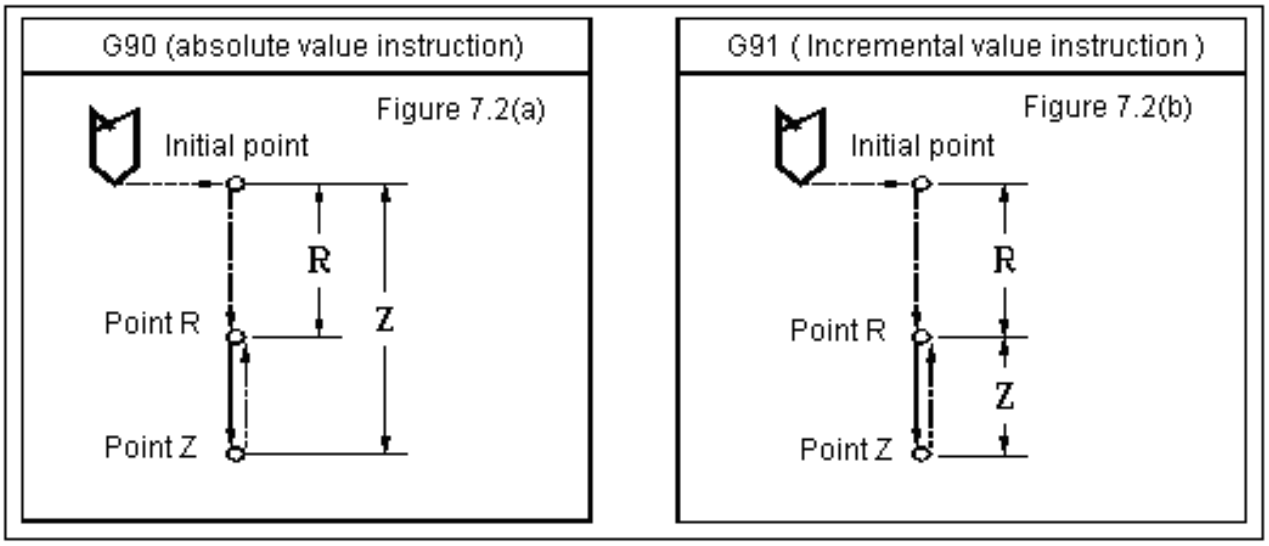
1. Fast locating of axis X and Y.
2. Fast locating axis Z to point R.
3. Hole machining.
4. Action at hole bottom.
5. Axis Z returns to point R.
6. Axis Z fast returns to initial point.

Table 7.1 Fixed Hole Machining Cycle

G Code	Machining (negative direction at axis Z)	Action at hole bottom	Returning (positive direction at axis Z)	Application
G73	Time by time, cutting feed	–	Fast locating feed	High-speed deep hole drilling
G80	–	–	–	Canceling fixed cycle
G81	cutting feed	–	Fast locating feed	Regular drilling cycle
G82	cutting feed	Pause	Fast locating feed	Drilling or coarse boring
G83	Time by time, cutting feed	–	Fast locating feed	Deep-hole drilling cycle
G84	cutting feed	Pause—Spindle on CCW	cutting feed	Right-thread tapping
G85	cutting feed	–	cutting feed	Boring cycle
G86	cutting feed	Spindle off	Fast locating feed	Boring cycle
G88	cutting feed	Pause—Spindle off	Manual	Boring cycle
G89	cutting feed	Pause	cutting feed	Boring cycle



he instruction G90/G91 and G98/G99 can affect the execution of the instruction for fixed hole machining cycle. Figure 7.2(a) and Figure 7.2(b) shows the influence posed by G90/G91 to the instruction for fixed hole machining cycle.

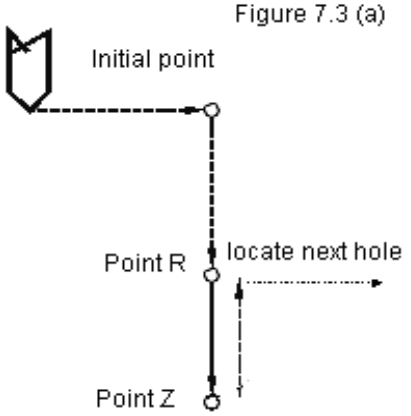


G98/G99 determines whether the cutter returns to point R or the initial point after the hole machining is finished in the fixed cycle. Under G98 mode status, axis Z will return to the initial point after hole machining. Under G99 mode status, it will return to point R.

Normally, if the hole being machined is on a perfectly flat plane, we can use the instruction G99. This is because the system will position the next hole after returning to point R under G99 mode status. As in the regular programs point R is very close to the surface of the workpiece, G99 will save the time of machining the parts. However, if there is protruded areas or bars on the surface of workpiece, the cutter may collide with the workpiece when G99 is used. In this case, G98 should be used, by which the next hole will be located after axis Z returns to the initial point. Thus this practice could be safer. Please refer to Figure 7.3(a) and Figure 7.3(b).

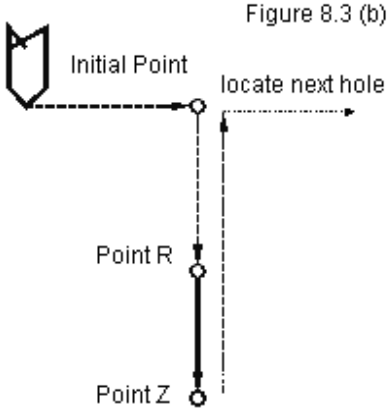
G99 (Return to point R)

Figure 7.3 (a)



G98 (Return to inital point)

Figure 8.3 (b)



The parameters of hole to be machined are provided after G73/G74/G76/G81～G89, with format as follows:

G××X___ Y___ Z___ R___ Q___ P___ F___ K___;

G×× : Hole machining method

X___ Y___ Z___ :Parameters for position of the hole to be machined

R___ Q___ P___ F___ : Machining parameter of the hole

K___ : Repeat times

Hole machining method: G	See Table 7.1
Parameters for position of the hole to be machined: X, Y	When the position of the hole to be machined is designated by incremental or absolute value method, the locus by which the cutter moves along the hole and cutter's speed are the same as G00.
Parameters for position of the hole to be machined: Z	The position of the hole bottom along axis Z is designated by absolute value method, whereas the distance between point R and the hole bottom is designated by incremental value method.
Machining parameter of the hole: R	The position of point R along axis Z is designated by absolute value method, whereas the distance between the initial point and point R is designated by incremental value method.
Machining parameter of the hole: Q	Used for designating the feed of each time in the deep-hole drilling cycle G73 and G83, and the offset in fine boring cycle G76 and reverse boring cycle G87 (always incremental instruction, regardless of G90 or G91 mode status)
Machining parameter of the hole: P	Used for designating the pause time in the fixed cycle where the hole bottom has pause, with unit as second.
Machining parameter of the hole: F	Used for designating the cutting feedrate in the fixed cycle. In the fixed cycle, the movement from the initial point to point R and point R to initial point is carried out at the fast feedrate, and movement from point R to point Z is carried out at the cutting feedrate designated by F. However, the movement from point Z to point Z can be carried out either at the rate designated by F or

	at the fast feedrate, depending on the nature of the fixed cycle.
Repeat times: K	Used for designating the repeat times of the fixed cycled at the current locating point. If K is not executed, NC will consider K=1. If K=0, there will be no execution at the current point in the fixed cycle.

As the hole machining designated by G×× is of the mode status, the current mode status will remain unchanged if it not is modified or the fixed cycle is not cancelled. The fixed cycle can be canceled by using G80 or instruction G of group 01. The machining parameter of the hoe is of the mode status too, and it will also remain unchanged before it is modified or the fixed cycle is canceled, even if the mode status for hole machining is changed. Any machining parameter of the hole can be designated or modified when a fixed cycle is instructed or at any time the fixed cycle is executed. The repeat times are not a value of mode status, and it is only provided when repetition is needed. The feedrate is a value of mode status, which will exist even if the fixed cycle is canceled. If NC system is reset in the process of executing a fixed cycle, the mode status of hole machining, machining parameter of the hole and repeat times will all be canceled.

The following example will help you better understand the aforesaid contents:

Item No.	**Program content**	**Notes**
1	S____ M03	Provide the rotation speed and instruct the spindle to rotate in positive direction.
2	G81X__Y__Z__R__F__K__	Fast position to the designated points of X and Y, and machine the part according to the parameters provided by Z, R and F and with the method provided by G81. Then repeat the process for K times. At the beginning of executing the fixed cycle, Z, R and F are the necessary machining parameters of the hole.
3	Y__	Axis X remains unmoved, and axis Y is fast located to instructed point for machining. The hole machining parameter and method the keep the mode status value as 2. K value of 2 is ineffective here.
4	G82X__P__K__	Hole machining method is modified, and hole machining parameter Z, R and F keep their respective mode status values. Provide the value of hole machining parameter P and designate to repeat K times.
5	G80X__Y__	Fixed cycle is canceled, and all hole machining parameters are canceled except F.
6	G85X__Y__Z__R__P__	As the fixed cycle is canceled when 5 is executed, the necessary hole machining parameters, except F, must be provided again, even if these parameters are unchanged when compared to the original values.
7	X__Z__	Axis X is located to the instructed point for machining the hole. The hole machining parameter Z is modified in this program segment.

8	G89X__Y__	Position to XY's instructed point for hole machining. The hole machining method is modified as G98. R and P are designated by 6 and Z by 7.
9	G01X__Y__	The mode status of fixed cycle is canceled. All hole machining parameters, except F, are canceled.

The following methods are used for indicating the feed of each segment in the figures below:

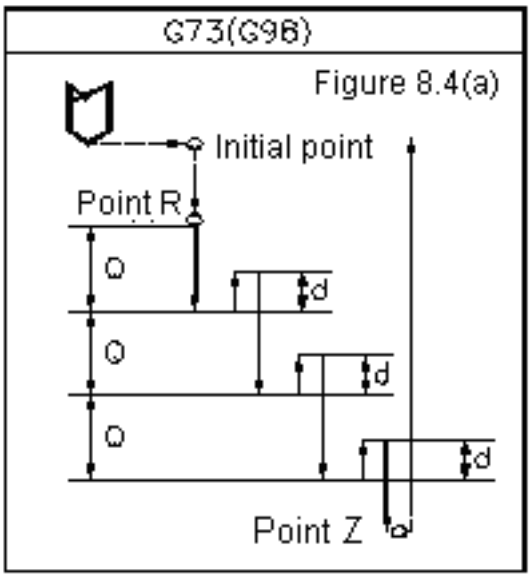
To indicate the movement with the fast feerate: ——→

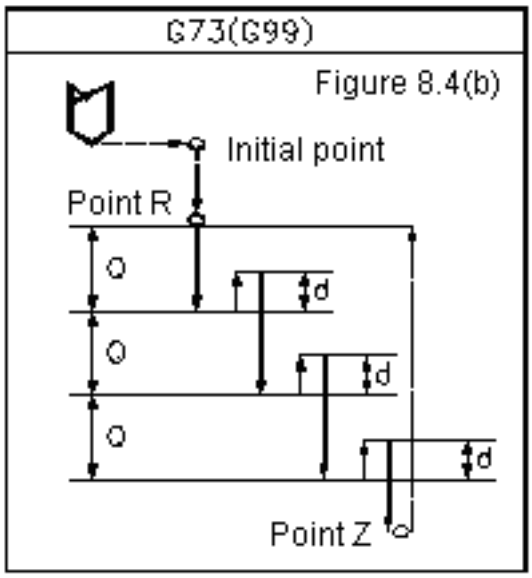
To indicate the movement with the cutting feerate: →

To indicate the manual feed: ----------→

➢ G73（High-speed drilling cycle）

Format：G73 X_ Y_ Z_ R_ Q_ F_





In the high-speed drilling cycle for deep holes, the feed from R to Z is accomplished section by section. After each section of cutting feed is finished, axis Z will lift upward for some distance, then the cutting feed of the next section will be performed. The distance d, by which the axis Z lifts upward, is provided by 531# parameter. The depth of feed is provided by the hole machining parameter Q each time. This fixed cycle is mainly used for machining holes with small radius-depth ratio (likeΦ5, depth of 70). The action that axis Z lifts upward each time the cutting feed of each section is finished plays a role of breaking chips.

➢ G74（Back whorl tapping cycle）

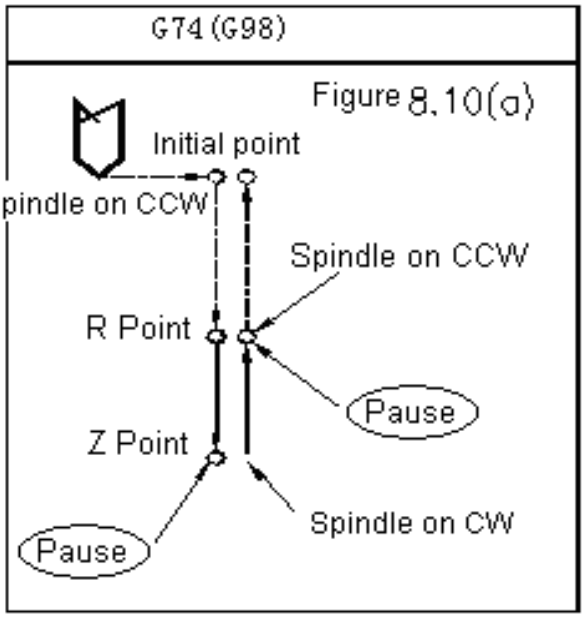
Format G74 X_ Y_ Z_ R_ F_(D_)

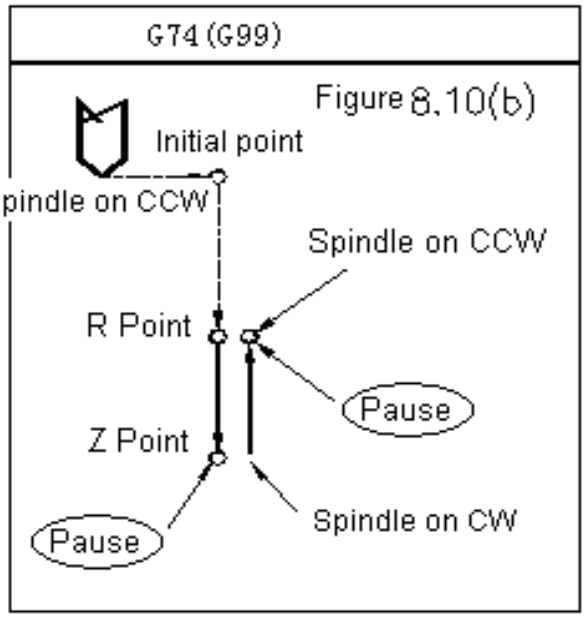
X_Y_: whorl position

Z_: whorl depth

R_: initial point of the feed and feed withdrawal

F_(D_): calculate the feed speed according to the pitch, or give the pitch distance with D_ directly.





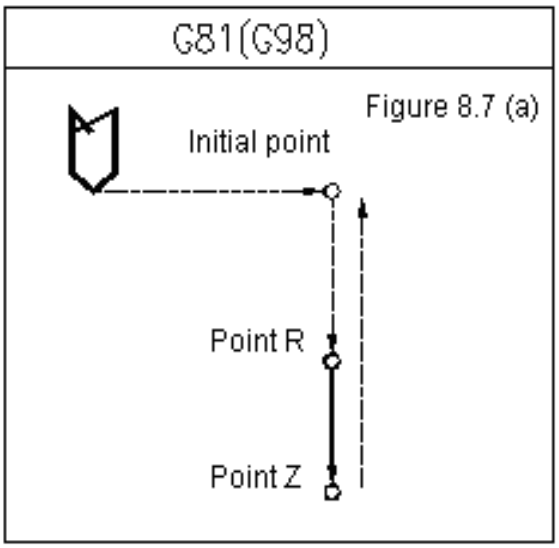
Notice: in the cycle of G74 and G84, the function of the feed rate switch and feed holding switch will be neglected, namely the feed rate will be keep at 100％, and it can not stop before a fixed cycle has been executed, the main shaft should be ordered to to rotate around the tapping direction before the cycle.

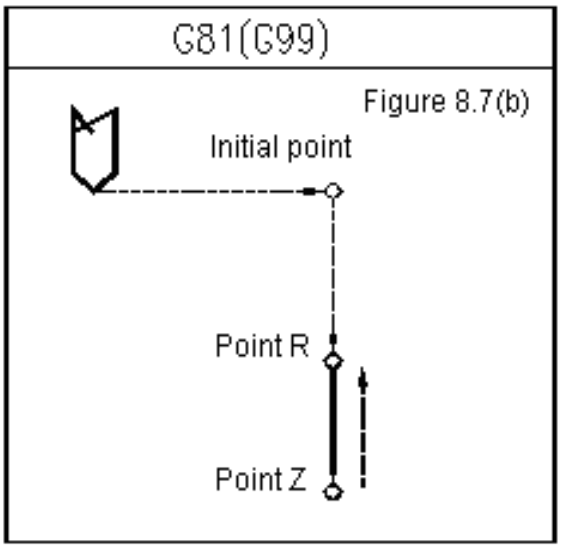
➢ G80（Cancel the fixed cycle）

After instruction G80 is executed, the fixed cycle will be canceled by this instruction, and all hole machining parameters of R and Z, except F, will be canceled. G code of another group 01 can play the same role.

➢ G81（Drilling Cycle）

Format G81 X_ Y_ Z_ R_ F_

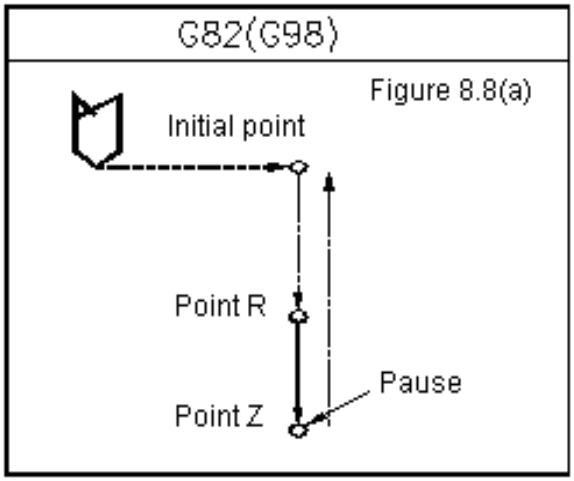


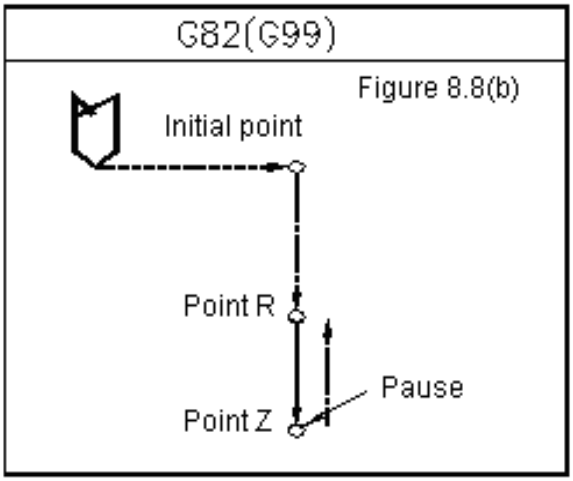


G81 is the simplest fixed cycle, whose execution process can be described as: after X, Y locating, axis Z fast moves to R, and moves to Z with F rate, then fast returns to initial point (G98) or R (G99). There is no action at the hole bottom.

➢ G82（Drilling cycle,Boring cycle）

Format G82 X_ Y_ Z_ R_ P_F_



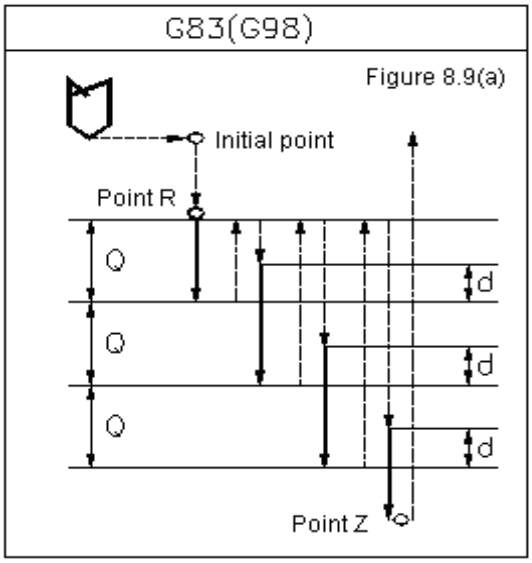


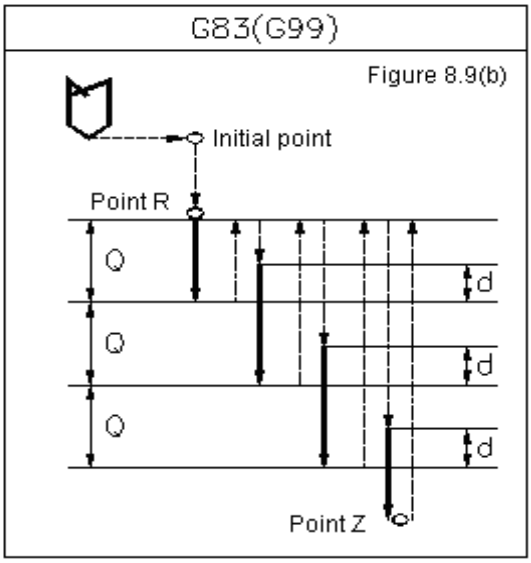
The fixed cycle of G82 has an action of pause at the hole bottom. Other procedures are the same as G81. The pause at the hole bottom can improve the precision on hole's depth.

➢ G83（Deep-hole Drilling Cycle）

Format G83 X_ Y_ Z_ R_ Q_ F_

Similar to G73, under instruction G83, the feed from R to Z is also accomplished section by section. Unlike G73, axis Z returns to R after the feed of one section is finished. Then it moves at fast feedrate to the position, which keeps a distance of d to the start point of the next feed section, and starts the movement for the feed of next section. The feed distance for each section is given by the machining parameter Q, which is a positive value permanently. The vale of d is provided by the parameters of 532＃machine tool. Please refer to Figure 8.9:

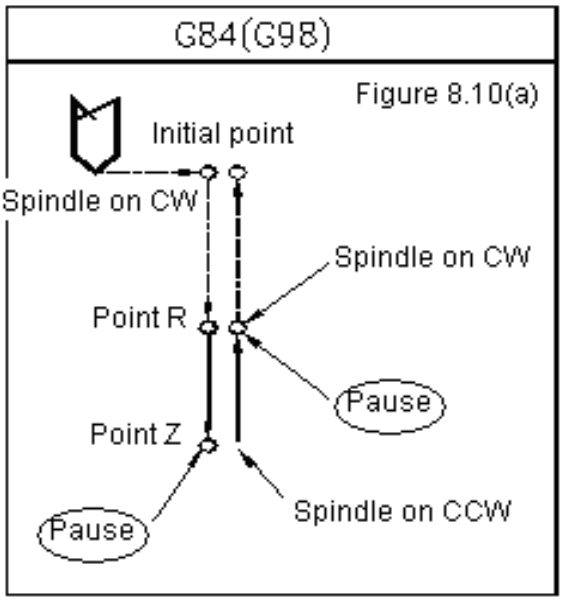


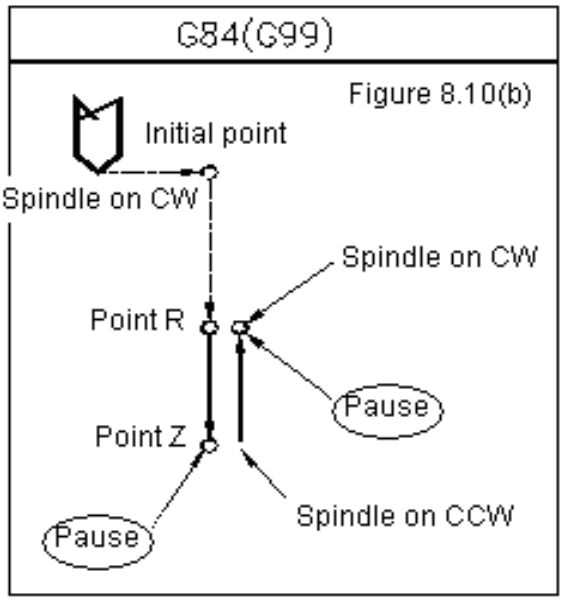


➢ G84（Tapping Cycle）

Format G84 X_ Y_ Z_ R_ F_(D_)

X_Y_: whorl position
Z_: whorl depth
R_: initial point of the feed and feed withdrawal
F_(D_): calculate the feed speed according to the pitch, or give the pitch distance with D_ directly.



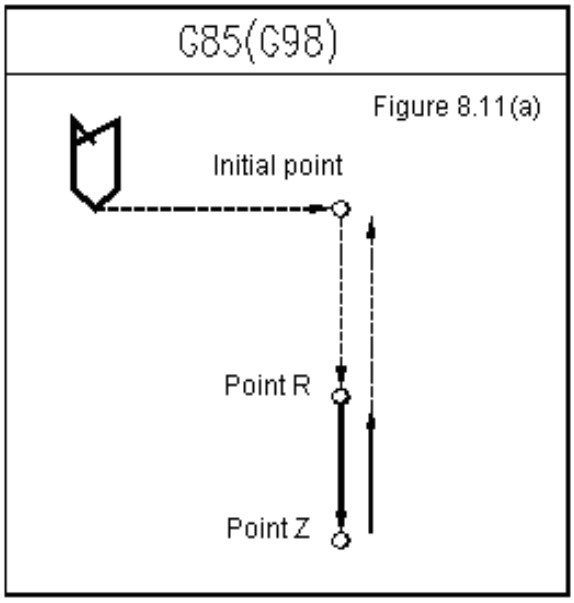


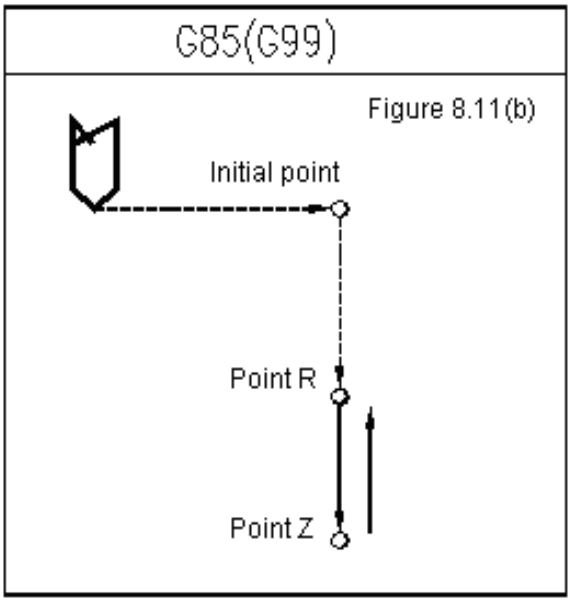
Notice: In the cycle of G74, G84, the function of feed rate switch and feed holding switch will be ignored, namely feed rate is kept at 100%, it can not stop before a fixed cycle is finished, you should command main shaft to rotate along the tapping direction before the cycle.

7）G85（Boring Cycle）

Format G85 X_ Y_ Z_ R_ F_

This is a very simple fixed cycle, whose execution process can be described as: after X, Y locating, axis Z fast moves to R, and moves to point Z with rate designated by F, then fast returns to R. If it is under G98 mode status, it will fast return to the initial point after returning to R.

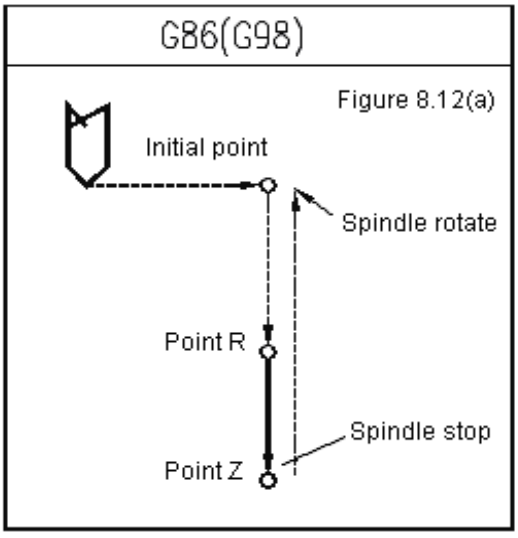


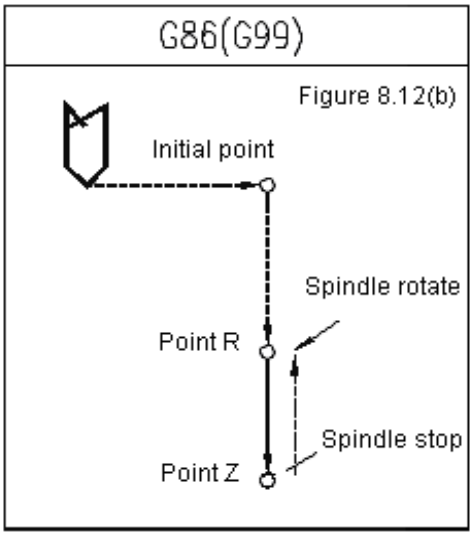


➢ G86（Boring Cycle）

Format G86 X_ Y_ Z_ R_ F_

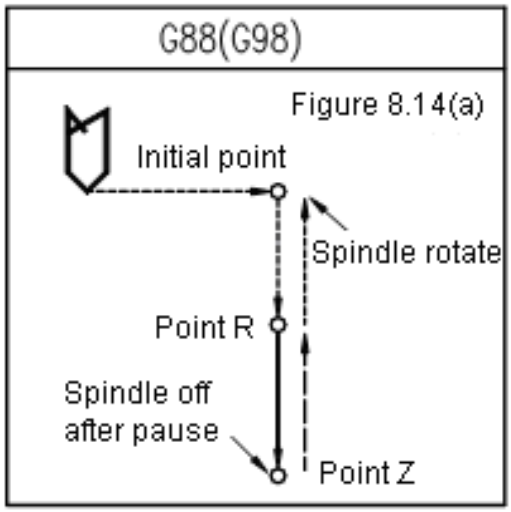
The execution process of this fixed cycle is similar with G81. The difference between them is that in G86 the spindle will be stopped after the cutter moves to the hole bottom. It will make the spindle rotate with the original direction and speed after the cutter returns to R and the initial point.

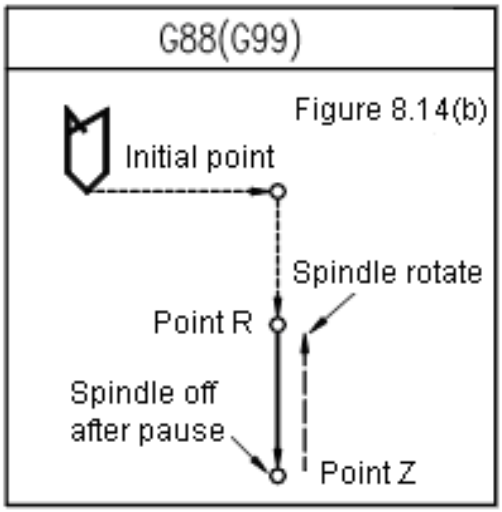




➢ G88（Boring Cycle）

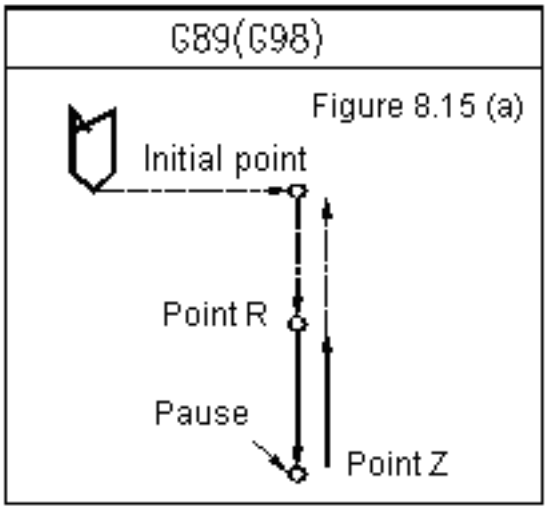
Manual return is available in fixed cycle G88, which is used for boring in the cycle (see the Figure below):

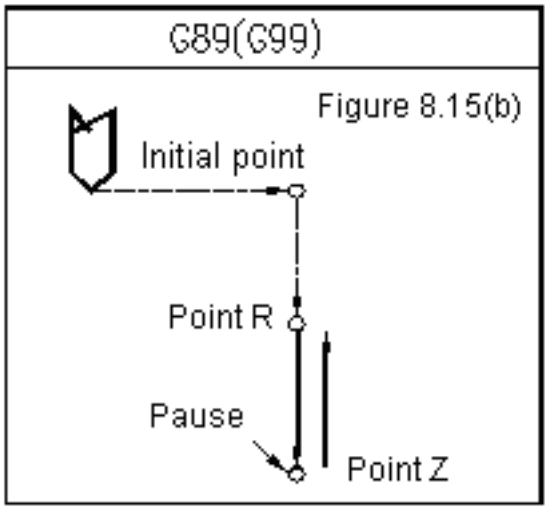




➢ G89（Boring Cycle）

In this fixed cycle, the pause of hole bottom is added on the basis of G85. Please refer to Figure 8.15:





➢ Precautions on fixed cycle for hole machining

a. In programming, it should be noted that the spindle must be instructed to rotate by using S and M code before the instruction for fixed cycle is executed.

M03 ； spindle on CW

.

.

G□□..... ； correct

.

.

M05 ； Spindle off
G□□……； Incorrect (instruction M03 or M04 is needed before this program segment)

b. Under the mode status of fixed cycle, the program segment including X, Y, Z and R will execute the fixed cycle. If a program segment doesn't include any of the aforesaid addresses, this program will not execute the fixed cycle, except the address X in G04. Besides, the address P in G04 will not change the P value in the hole machining parameters.
； (hole not machined)
F__； (hole not machined, F value upgraded)
M__； (hole not machined, only execute auxiliary functions)
G04 P__； (hole not machined, use G04 P_ to change the hole machining parameter P)

c. The hole machining parameter Q and P must be designated in the executed program segment in the fixed cycle. Otherwise, the instructed Q and P values will be ineffective.

d. In executing the fixed cycle with spindle control (such as G76 and G84), the spindle may have not reached the instructed speed when the cutter starts cutting. In this case, the pause instruction G04 should be added between the operations for hole machining.

e. As we have discussed, the G code in group 01 can also play a role of cancelling the fixed cycle. Therefore, the instruction for fixed cycle and the G code of group 01 should not be written in the same program.

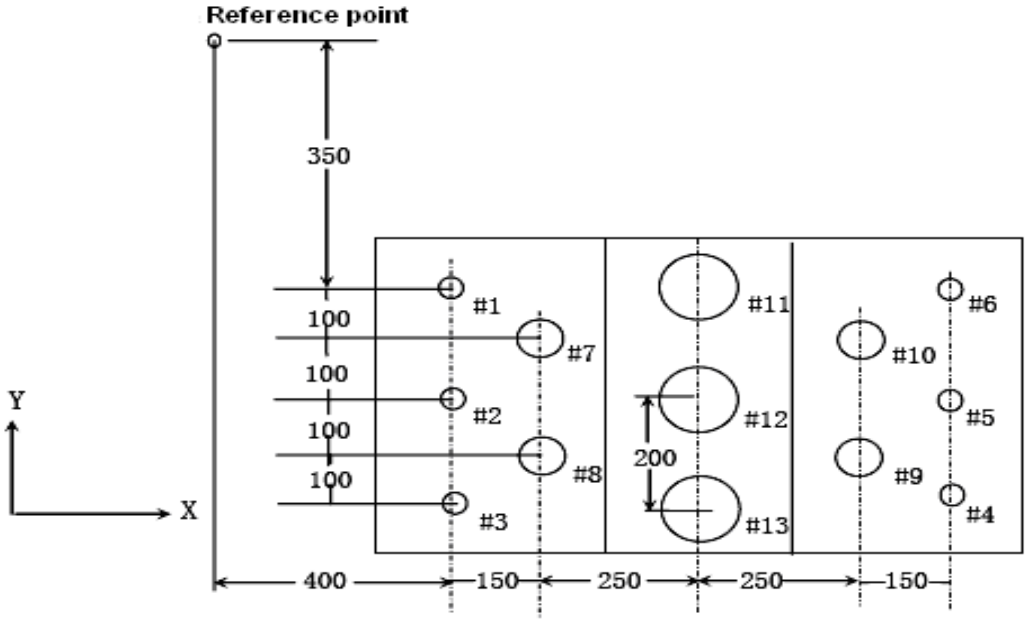
f. If an M code is instructed in the program for executing the fixed cycle, this M code will simultaneously executed as the fixed cycle. The signal that indicates that the instruction M has been executed will be sent out after axis Z returns to R or the initial point. When parameter K is used for repeatedly executing the fixed cycle, the M code will be executed at the first time the fixed cycle is executed.

g. Under the fixed cycle mode, the instruction G45-G48 for cutter offset will be ignored (not executed).

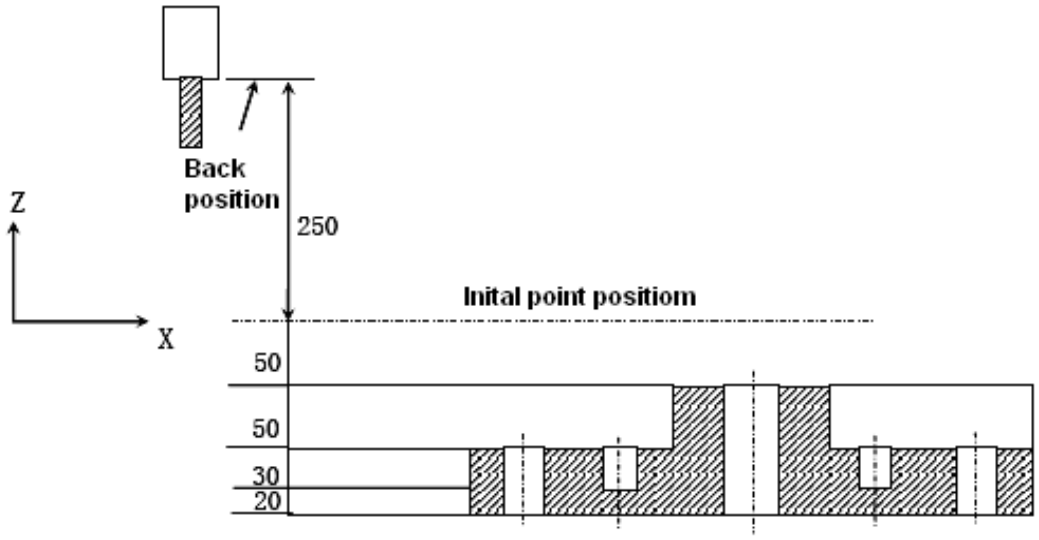
h. When the switch for single program segment is set at the upper position, the fixed cycle will stop after axis X and Y locating, fast feeding to R and returning from hole bottom (to R or initial point). In other words, to complete the machining on one hole, the start-up button for cycle needs to pressed thrice. In these three stops, the first two keep the system to be in feed hold state, and the last one make the system to be in stop state.

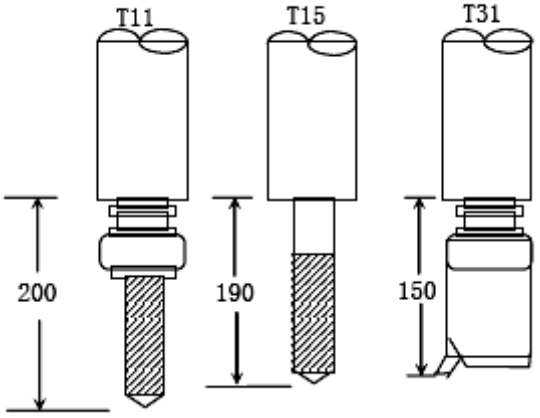
i. In executing G74 and G84 cycles, if the button for feed hold is pressed between the two steps, namely axis Z moves from point R to point Z and moves from point Z to point R, the indicator for feed hold will be illuminated immediately. However, the action of the machine tool won't be stopped immediately, and only when axis Z returns to R can the system enter the feed hold state. In addition, in the G74 and G84 cycles, the switch for feed percentage is ineffective and it remains 100%.

➢ Example for cutter length compensation and fixed cycle



1～ 6. . . Drilling hole Φ10
7～10. . . Drilling hole Φ20
#11～13. . . Boring hole Φ95 (50mm deep)





The value of offset number 11 is 200.0, of 15 is 190.0 and of 31 is 150.0. The offsets are set respectively. The program is shown as follows:

```
N001 G92 X0 Y0 Z0  ;       set the reference point of coordinate system.
N002 G90 G00 Z250.0 T11 M6;    change cutter.
N003 G43 Z0 H11  ;         At the initial point, cutter length compensation oF plane.
N004 S30 M3  ;             Spindle starts up.
N005 G99 G81 X400.0 Y-350.0
Z-153.0 R-97.0 F120.0  ;    Machine hole #1 after locating.
N006 Y-550.0  ;          Machine hole #2 after locating, return to plane of R.
N007 G98 Y-750.0  ; Machine hole #3 after locating, return to plane of initial point.

N008 G99 X1200.0 ;         Machine hole #4 after locating, return to plane of R.
N009 Y-550.0  ;           Machine hole #5 after locating, return to plane of R.
N010 G98 Y-350.0  ; Machine hole #6 after locating, return to plane of initial point.

N011 G00 X0 Y0 M5  ;         Return to reference point, spindle off.
N012 G49 Z250.0 T15 M6  ;    Cancel cutter length compensation, change cutter.
N013 G43 Z0 H15  ;     On the plane of initial point, cutter length compensation.

N014 S20 M3  ;             Spindle starts up.
N015 G99 G82 X550.0 Y-450.0  ;
Z-130.0 R-97.0 P30 F70;  Machine hole #7 after locating, return to plane of R.
N016 G98 Y-650.0  ;  Machine hole #8 after locating, return to plane of initial point.

N017 G99 X1050.0  ;      Machine hole #9 after locating, return to plane of R.
N018 G98 Y-450.0  ;  Machine hole #10 after locating, return to plane of initial point.

N019 G00 X0 Y0 M5  ;         Return to reference point, spindle off.
N020 G49 Z250.0 T31 M6  ;  Cancel cutter length compensation, change cutter.
N021 G43 Z0 H31  ;     Cutter length compensation at the plane of initial point.
N022 S10 M3  ;             Spindle starts up.
N023 G85 G99 X800.0 Y-350.0  ;
Z-153.0 R47.0 F50  ; Machine hole #11 after locating, return to plane of R.
N024 G91 Y-200.0  ;  Machine hole #12 and #13 after locating, return to plane of R.
Y-200.0 ;
N025 G00 G90 X0 Y0 M5  ;  Return to reference point, spindle off.
N026 G49 Z0  ;           Cancel cutter length compensation.
N027 M30  ; %              Program stop.
```

3.3 Assistant Function（M，S，T）

In this System, S code is used for programming the spindle speed, T code for cutter selection, and other programmable miscellaneous functions are realized via M code.

3.3.1 M Code

M Code List：

M Code	Function
M01	Program stop
M03	Spindle on CW
M04	Spindle on CCW
M05	Spindle stop
M06	Change cutter command
M08	Open cooling
M09	Close cooling
M32	lubrication open
M33	lubrication close
M30	Program finished and return to program header
M98	Invoke subprogram
M99	Subprogram finished and return/repeated execution
M56	Output NO.2 interrupt port is high electric level
M57	Output NO.2 interrupt port is low electric level
M58	Output NO.3 interrupt port is high electric level
M59	Output NO.3 interrupt port is low electric level
M10	Output NO.6 interrupt port is high electric level
M11	Output NO.6 interrupt port is high electric level
M20	Output NO.7 interrupt port is high electric level
M21	Output NO.7 interrupt port is low electric level
M12	Output NO.8 interrupt port is high electric level
M13	Output NO.8 interrupt port is low electric level
M14	Output NO.9 interrupt port is high electric level
M15	Output NO.9 interrupt port is low electric level
M16	Output NO.10 interrupt port is high electric level
M17	Output NO.10 interrupt port is low electric level
M18	Output NO.11 interrupt port is high electric level
M19	Output NO.11 interrupt port is low electric level
M40	Output NO.12 interrupt port is high electric level
M41	Output NO.12 interrupt port is low electric level
M42	Output NO.13 interrupt port is high electric level
M43	Output NO.13 interrupt port is low electric level
M44	Output NO.14 interrupt port is high electric level
M45	Output NO.14 interrupt port is low electric level
M46	Output NO.15 interrupt port is high electric level
M47	Output NO.15 interrupt port is low electric level
M48	Output NO.16 interrupt port is high electric level
M49	Output NO.16interrupt port is low electric level
M50	Output NO.17 interrupt port is high electric level
M51	Output NO.17 interrupt port is low electric level
M66	Output NO.20 interrupt port is high electric level
M67	Output NO.20 interrupt port is low electric level

M64	Output NO.21 interrupt port is high electric level
M65	Output NO.21 interrupt port is low electric level
M62	Output NO.22 interrupt port is high electric level
M63	Output NO.22 interrupt port is low electric level
M60	Output NO.23 interrupt port is high electric level
M61	Output NO.23 interrupt port is low electric level
M88 Pn Lm	Inspect waiting input IO（IN n）whether the level signal m（high or low）
M89 Pn Lm Qt	Output OUT n, level is m，t millisecond delay to output

In machine tools, the roles of M code can be classified as two types: One is used for controlling the execution of the program and the other is used for controlling the action execution of the spindle, ATC device, cooling system and other auxiliary equipment.

Used M codes for program control

The M codes for program control include M00, M30, M98 and M99, whose functions are respectively described as follows:

M00……Program stop. When NC receives M100, the program execution will be interrupted. The program execution will be resumed after resetting and pressing start-up button.

M30……Program end, and return to program header.

M98……Invoke subprogram.

M99……Subprogram end, and return to main program.

Other M Code

M03……spindle on cw. Use this instruction to allow the spindle to rotate counter-clockwise at the current designated speed (CWW).

M04……Spindle on cww. Use this instruction to allow the spindle to rotate clockwise at the current designated speed (CW).

M05……spindle stop.

M06……Change cutter. M06 T02 is used for changing to cutter 2#.

M08……open cooling.

M09……close cooling.

M32……lubrication open.

M33……lubrication close.

M88……specified input IO to carry out level judgement, continue carrying out if it is the same or wait always. If the level signal is not specified, then default it as low level signal. For instance, M88 P0 L1 waiting INO is high level, of wait always.

M89……specify output IO as the specified level judgement, if there is no specified level signal, default is as the low level, if the Q value is specified, then this operation should has Q millisecond delay before output the IO signal. For instance, M89 P5 L0, specify OUT5 output low level.

Notice:

- when the move instruction and M is in the same programm segment, then the M instruction will be carried out preferentially.
- If there are more than one M code in the program, then there is only one is in effect, that is the last defined M code is in effect.

3.3.2 S Code

The speed of spindle is sent out via S code, which is of the mode status. In other words, it will remain effective after the speed is set till the mode status value of another S code changes.

The maximum value of the S instruction is limited by the main shaft maximum rotation speed of the parameter P5.020

The S instruction has three kinds of output mode, they are limited by the parameter P2.049(axis number of the main shaft specified interface), P1.061(frequency conversion control mode), as follows:

Set the P2.049 as nonzero value:

It means current main shaft is the AB phase pulse control mode, then the S value is depending on the setting of the main shaft coder to decide the pulse frequency.

Set the P2.049 as nonzero value，set the P2.061 as 1:

The control mode of frequency conversion gear position and the communication of the frequency conversion, utilize four IO ora（OUT23 ~ OUT20)gear position. Four gear position consist of sixteen coding, namely the S instruction value is S00 ~ S15.

Set the P2.049 as nonzero value, set p2.061 as 0:

The control mode of frequency conversion analog, according to the ratio between S value and the maximum rotation speed of parameter p5.020 then time 10v, get the simulation voltage should be output; The MO3 or MO4 should be executed after the S instruction has been specified, then output the analog.

3.3.3 T Code

The random selection mode is adopted in the cutter warehouse of this machine tool. That is to say, the two-digit T code—T ×× will be used for designating the cutter number, regardless of which sheath the cutter is in. The value of T in the address can be any integer from 1-99.

Warning:

The cutter table must be set correctly. If the contents in the table are inconsistent with the reality, the machine tool will be severely damaged and unpredicted consequences may occur.

3.4 G code template programming(DXF lead-in rule programming)

ADT series products imbibe many programming mode, considering the popularity of international current CAD software, we make the DXF format file compatible in order to strengthen the convenience of the programming.

The DXF format file is a kind of vector format storage file, it can save the precision up to 14 digits after the decimal point, so it can picture any graphics without distortion. In addition, due to the maturity of the CAD software, editing the DXF is very easy; because the storage of the DXF is a kind of order less storage, namely the storage order of any graphics only has something to do with its generation time, changing the DXF graphics file into processing route file is very troublesome, recently the current way is using the mould design software to do the second edit to produce the processing knife path. 4240 controller adopts another kind of idea to achieve one time picturing lead-in processing, recently it has been widely applied in the fixed-point processing industry such as impacting and drilling; We make use of the convenience of the G code, apply the template concept, combine with the DXF efficiently, generate the G code file and process it directly. Now we come to the template programming format of the G code.

```
[HEADER]                    //head of template
%
O0001
 [ADTLAYER 1 HEAD]          //layer 1 head
G54G00G90G17                //layer 1 head
T1M06
[ADTLAYER 1 PROCESS]        //layer 1 process
G01X[#X]Y[#Y]

[ADTLAYER 1 FINIS]          //layer 1 end
M09

[ADTLAYER 2 HEAD]           //layer 2 head
G55G00G90G17
T2M06
[ADTLAYER 2 PROCESS]        //layer 2 process
G01X[#X]Y[#Y]

[ADTLAYER 2 FINIS]          //layer 2 end
M08
…                           //each layer , as above

[END]                       //end of template
M30
%
```

Instruction of the rule of the template and DXF file

1. The rule of the template is wrote freely, use the square brackets to wrap the name of each rule and occupy a exclusive line;
2. The rule can be edited via PC and cover it in GTEMPLET.GT of the controller ADT directory, it will load this file as template when load it next time.
3. The rules of the template support 16 layer recognition conversion of the DXF format file, but the layer name should be capital letters ADTLAYER and layer number 1, for instance ADTLAYER1…ADTLAYER16. The [ADTLAYER layer number rule name] of the template correspond to each layer of DXF. We should note that [HEADER] and [END] two rules are two public rule, namely it has nothing to do with the layer.
4. The drawing of the DXF supports many kinds of graphics, fully supporting looks huge and increase the programming complexity, so we just choose the X,Y,Z coordinate which is specified inside it, that is to say, no matter what kind of pictures stored in the DXF, finally it only identify the information of the layer point, but about other information, the system will filtrate automatically without process. Due to the storage characteristic of DXF, all the stored point coordinate are orderless, to our processing procedure, it will result in inefficiency. So we optimize it, that is the shortest path to sort, owing to compositor optimization has many kinds of equations, considering the calculating efficiency of our controller, we only choose one kind, so maybe the route come out is not the best, but compared to the time before optimization, the operation efficiency has been improved considerably.
5. The conventional rules of the template are as follows:
   [HEADER]+『[ADTLAYER 1 HEADER]+{[ADTLAYER 1 PROCESS] …}+[ADTLAYER 1 FINIS]』（number 1 layer rule）+『…』(the rules of each layer number）+[END];

   According to the above order of the rule, arrange the letters inside the rules with the same order and generate G code; thereinto, except the PROCESS rule, if the other rules exist, all are used one time, but PROCESS rule is a loop body, namely the test content of this rule will be cycled use, about the time of the cycle, it is the same number of the point number of the corresponding layer, the way of call the point coordinate in PROCESS is: [#X],[#Y],[#Z]; As long as the keyword appear, then it will call the X,Y,Z coordinate of the current point (reserve 4 digits after decimal point) then generate corresponding coordinate value.

6. Call-in conversion：
   The conversion is completed automatically, in file managing interface, choose the DXF file is to be transformed, then choose the readin working area the same as the G code file, after the system will read, optimize, transform, and generate the NC files with the same name automatically after estimates the suffix name, and it loads this NG file to the current system working area automatically, the course is as easy as reading the G code.

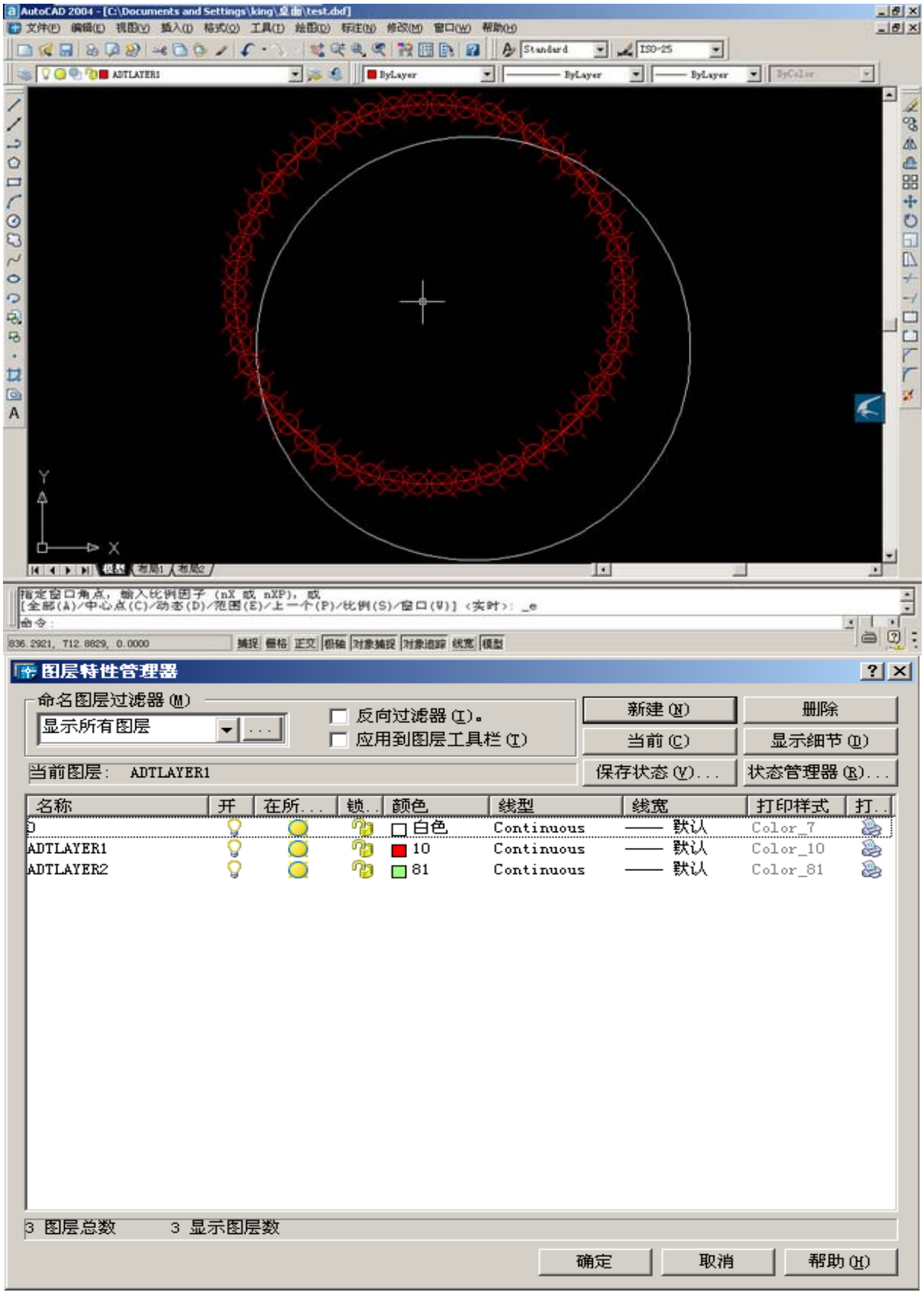
7. Routine：

Template as follows：

```
[HEADER]
%
O0001
[ADTLAYER 1 HEAD]
G54G00G90G17
```

```
T1M06
[ADTLAYER 1 PROCESS]
G01X[#X]Y[#Y]
[ADTLAYER 1 FINIS]
M09
[END]
M30
%
```

DXF graph:

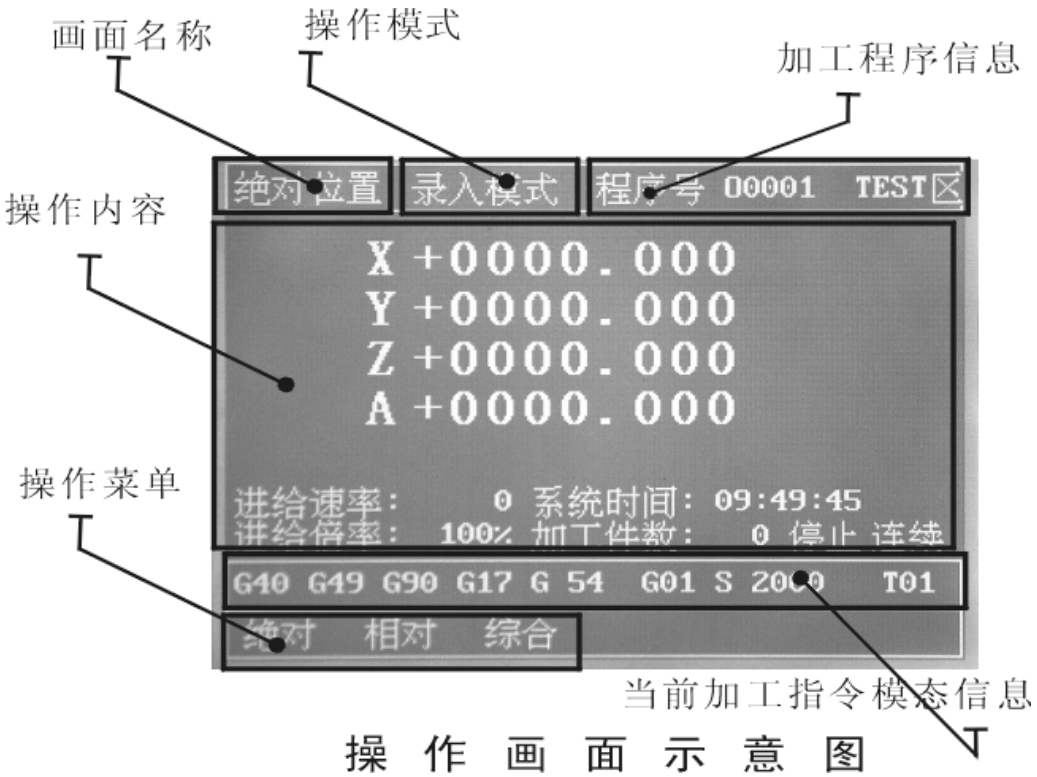

AutoCAD 2004 - [C:\Documents and Settings\king\桌面\test.dxf]
图层特性管理器
命名图层过滤器(M)
显示所有图层
反向过滤器(I)。
应用到图层工具栏(T)
新建(N)
删除
当前(C)
显示细节(D)
保存状态(V)...
状态管理器(R)...
当前图层: ADTLAYER1
名称 开 在所... 锁.. 颜色 线型 线宽 打印样式 打..
0 白色 Continuous 默认 Color_7
ADTLAYER1 10 Continuous 默认 Color_10
ADTLAYER2 81 Continuous 默认 Color_81
3 图层总数 3 显示图层数
确定
取消
帮助(H)

The generated G code are as follows:

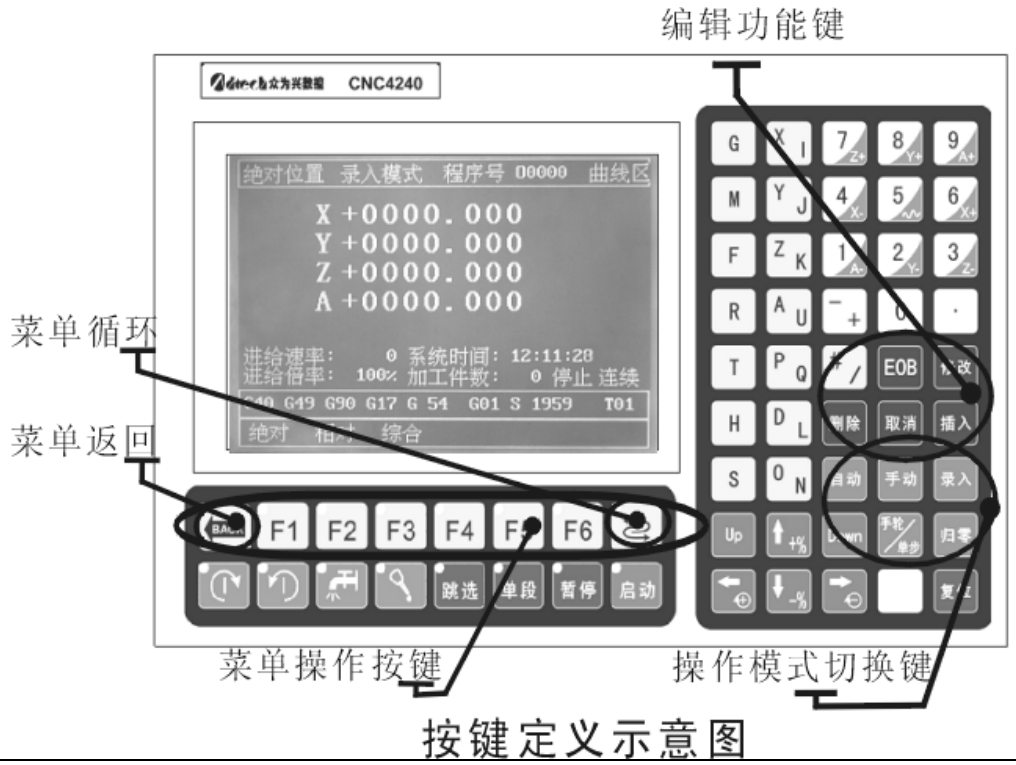
```
%
O0001
G54G00G90G17

T1M06
G01X1253.6957Y728.5054
G01X1250.5159Y779.0471
G01X1241.0266Y828.7917
…
G01X1144.4001Y452.4560
G01X1176.6802Y491.4760
G01X1203.8154Y534.2341
G01X1225.3775Y580.0561
G01X1241.0266Y628.2191
G01X1250.5159Y677.9637
M09
M30
%
```

4 System Operation Instruction

The operating method with multi-level menu is adopted in 4240 system, whose interfaces are classified as display mode, operation mode, instruction information of current mode status, operation menu and operation contents. See the following diagram:



操 作 画 面 示 意 图



按键定义示意图

4.1 Operation Mode

There are several modes of operation modes:

- Auto mode
- Handwheel run
- Manul mode
- MDI
- Handwheel mode
- single-step mode
- Zero mode

4.1.1 Auto Mode(Handwheel,Startup and program inspection)

The auto mode is used for executing the machining program. Under this mode, the parameters can't be modified, nor cutter aligned. Only machining is effective. Auto mode may be differently defined based on the situation. In non-pretreatment state, the System can only enter the auto mode. In the pretreatment state, the System can realize the functions of Handwheel, Startup and Program Inspection.

Handwheel Run :Handwheel Run is a commissioning mode under pretreatment state. I this mode, the speed NC executes the program depends on the speed you turn the handwheel.

Startup: When without the pendant, you can use this mode to realize the commissioning mode that provides the same effects as handwheel run. However, the machining speed in the program can't be modified. To activate this mode, you should use the startup key to replace the handwheel. So long as you press the startup key without releasing, the program will run till its end. Once released, the program will pause accordingly.

Program Inspection: In this mode, the System will scan the NC program on grammatical basis so that the wrong programming can be found out. As some of the comprehensive instructions in this mode can't be scanned yet, we will go on enhancing this inspection function in the system of new version so as to realize the completeness on function .

4.1.2 Manual Mode

In the manual mode, the numeric keys 1-9 are used for the continuous movement of each shaft. Key 5 is used for doubling the speed in manual mode. That is to say, each time you press this key, the speed will be doubled. To cancel this doubling function, press it repeatedly. The doubled speed will be highlighted white in the item of "manual rate" displayed below the position interface.

The actual speed is the sampled value of the moving speed of the shaft, which can tell the real rate of the shaft movement. It is unit is mm/min.

4.1.3MDI

The MDI mode is used for setting parameters for MDI operation, program edition and other operations. Only this mode can change the settings of the system.

4.1.4Handwheel or single-step mode

This is a multiplex key, which has two modes for switchover. To switch them, just press it repeatedly.

Single-step mode: Single-step mode is similar to manual mode, and their operations are the same. Use the numeric keys to move the coordinate axes. Only one set pulse increment can be moved each time.

To select the pulse increment in single-step mode, use the direction keys Up (+) and Down (-) .

Handwheel mode: The feed value is determined by detecting the signal in the pendant. In this mode, the feed shaft and feed unit are determined by the shaft-selection signal in the pendant. The hanwheel speed is set by an independent parameter. The handwheel mode is full frequency feed and is not influenced by the handwheel rate.

4.1.5Zero Mode

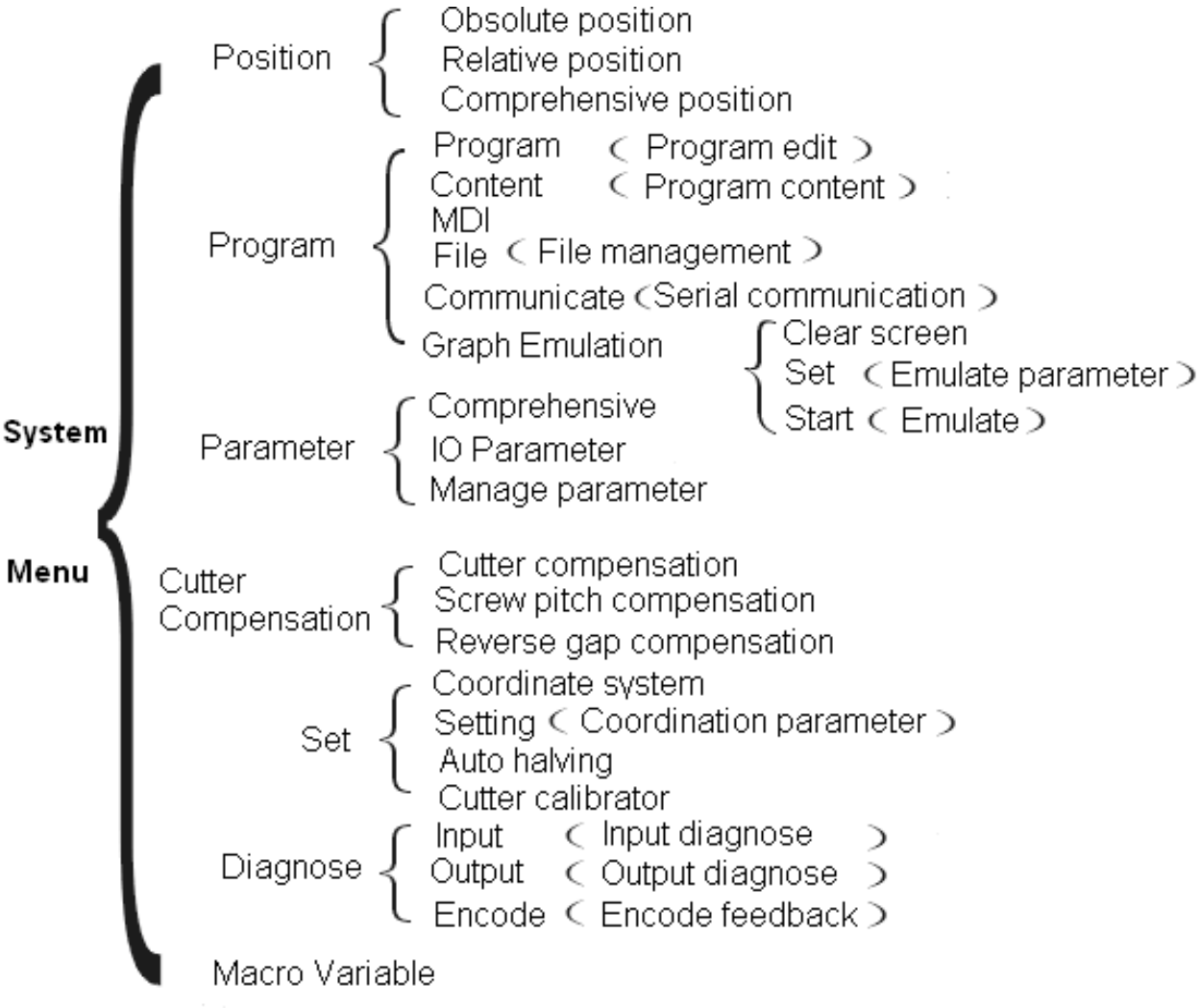
In zero mode, after you press X-, Y-, Z- and A- in the numeric keypad, the corresponding axis will be zeroed. Two types of zero mode are available: program zero and mechanical zero. To use this function, you should set "zero mode" via the parameters.

In mechanical zero mode, press "single segment", you can switch over between program zero and mechanical zero. That is a shortcut key. Use and parameter setting do not interfere with each other. Users can make choices according to their own needs.

In program zero mode, the zero action is completed when the cutter moves to zero point of the machine tool coordinate. There are two types of mechanical zero mode: one is that the origin of the machine tool coordinate is located only by the external switch, and the zero action is completed after the locating is induced; the other is that if the movement unit is a servo system, the corresponding Z phase in the IO configuration parameters can be enabled as 1. Thus, after the external switch is located, the System will automatically detect the signal of servo origin so as to realize the second mechanical locating. At present, this locating method can the highest precision.

In zero mode, in the absolute interface, you can manually enter the axis number. When the axis number is displayed on the reverse-rotation background, you can press Cancel to zero the position of the current axis in the machine tool's coordinate system. That is to say, the current point is used as the origin of the coordinate system of the machine tool. After this operation, the System will consider that a zero action has been completed. Therefore, no alarm for zero will be sent out in executing the program.

4.2 System Menu

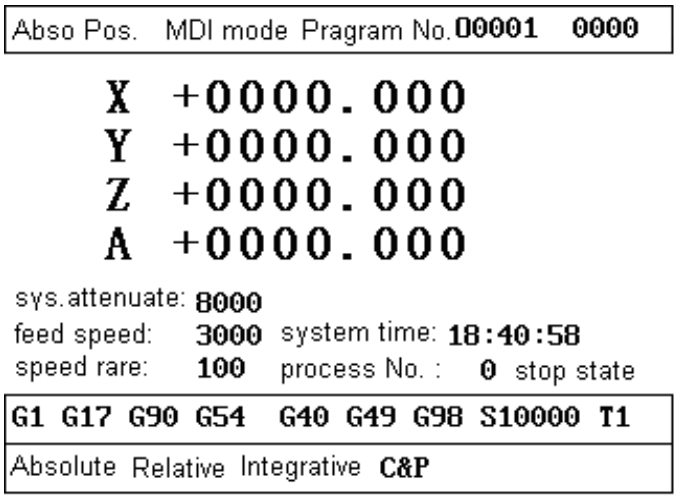


The multi-level design is adopted for the system menu. The menu operation keys, namely F1-F6, correspond to the menu options on the screen. To enter the corresponding menus, press these keys. If the menu contains submenu, the system will enter the submenu after it is pressed. If the menu is a function one, the corresponding function will be performed. The cyclic keys are used for scrolling the menu pages, whereas Menu Return is used for returning to the previous menu.

4.2.1 Position Interface（Position）

The position picture is used for displaying the current tool coordinate information, including absolute position, relative position, integrative position.

- **Absolute Position**



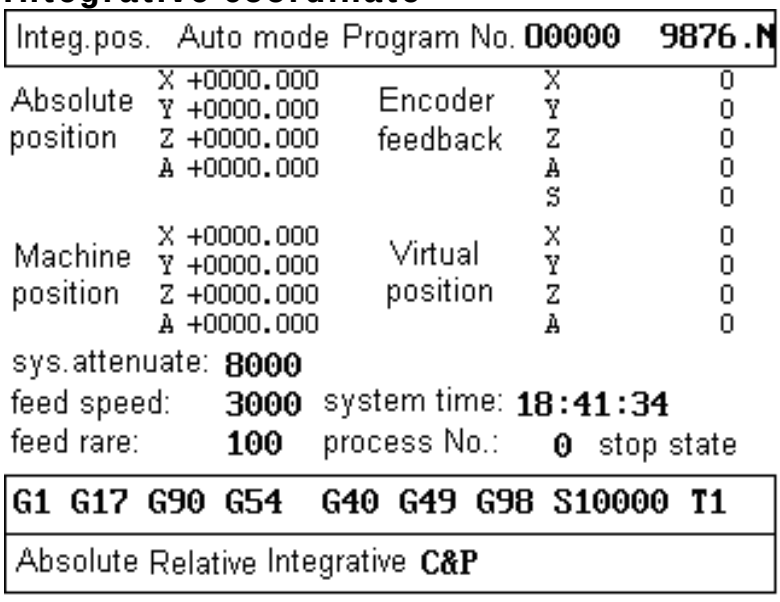
The position value of the current tool coordinate point relative to the coordinate origin of the workpiece coordinate system.

- **Relative Position**

1. In manual mode, the system can display the coordinate value by zeroing. After zeroing, the distance the cutter moves is the actual position before the offset is zeroed. Thus it is called relative position. Such interface can facilitate the calculation of the distance between the two points in some cutter-calibrating occasions. With the improvement of auto halving function, its use becomes increasingly rare.
2. The operating method is:

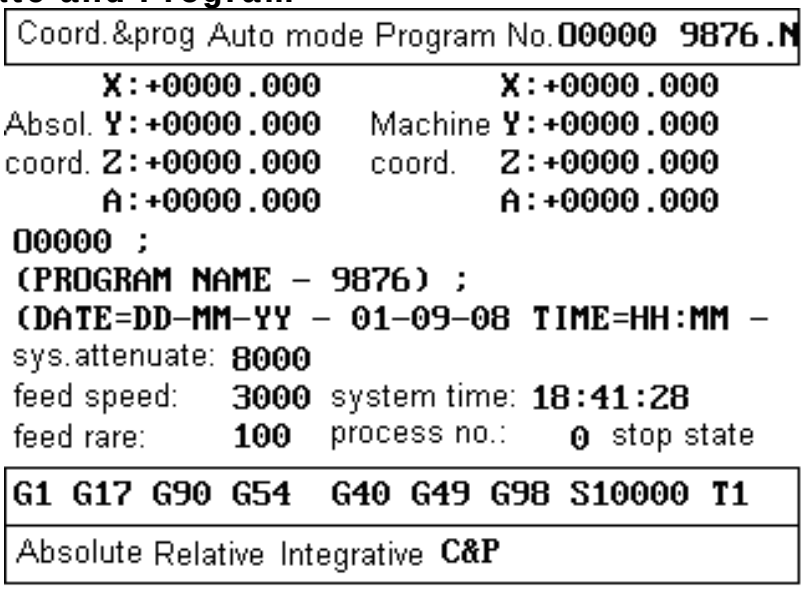
enter the relative position picture, then enter the manual-mode -> push the axis number which is to reset, for example, 'X'->X the coordinate flickering display -> push the "cancel" button -> X coordinate becomes 0;

- **Integrative coordinate**



Interface shared by both absolute coordinate and machine-tool coordinate.

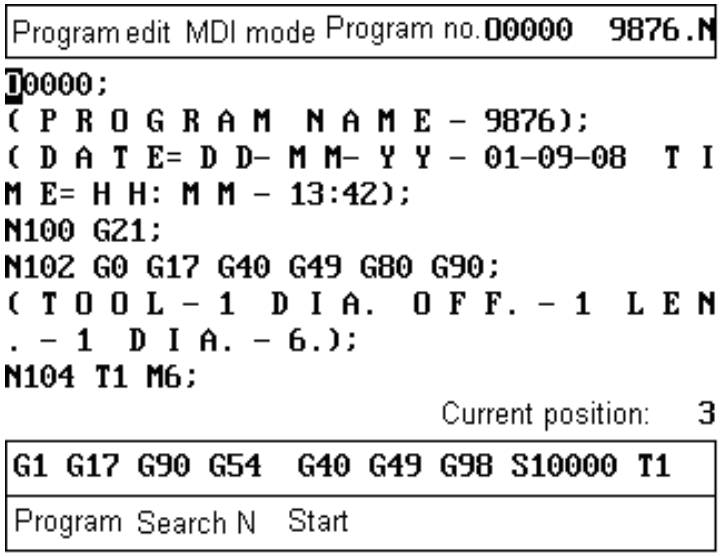
- **Coordinate and Program**



The picture of the coordinate and program can display the current coordinate of the workpiece and tool in real time and display the front three lines carried out by the program, it is convenient to adjust the program of the change of the relative coordinate value.

4.2.2 Program Interface（Program）

4.2.2.1 Program Edit



This interface displays the contents of NC's current machining program. In MDI mode, you can edit the NC program. In run mode, you can view the current running state. The operation method of the program edition interface are described as follows. The following edition functions are performed under on the basis of the MDI mode.

- Retrieval Function

The retrieval function is used for searching for the position of some key word.

Retrieval function can be realized via the program interface, with two methods available, namely, instruction word retrieval and program segment retrieval.

Instruction word retrieva: To retrieve, you can enter the instruction word. For example, if you want to search for the position of M30 via the retrieval program , you can enter M30 or M, then press Up and Down, and the cursor will move to the retrieved position. "M30" and "M" belong to different retrieval method. The former is performed with the condition of "full compliance", and the latter is performed with the condition that contents contain the letter M. Therefore, the latter can provide more actual retrieval results.

Program segment retrieval: Press Up and Down after entering the full program segment, or press Up and Down after entering O. The results can be similar with the instruction word retrieval.

- Delete Function

Similar to retrieval, the delete function can be classified as instruction word delete and program segment delete.

Instruction word delete: Move the cursor to the targeted instruction word, and press Delete.

Program segment delete: Enter the corresponding program number, and press Delete. To delete all the program segments in the current workspaces, enter "O-9999" and press Delete.

- Insert Function

The insert function can also be classified into two types: instruction word insert and program segment insert.

Instruction wor insert: Move the cursor to the targeted position, enter the instruction word and press Insert.

Program segment insert: To switch to the newly established program segment for programming, enter the program segment and press Insert.

- Modify Function

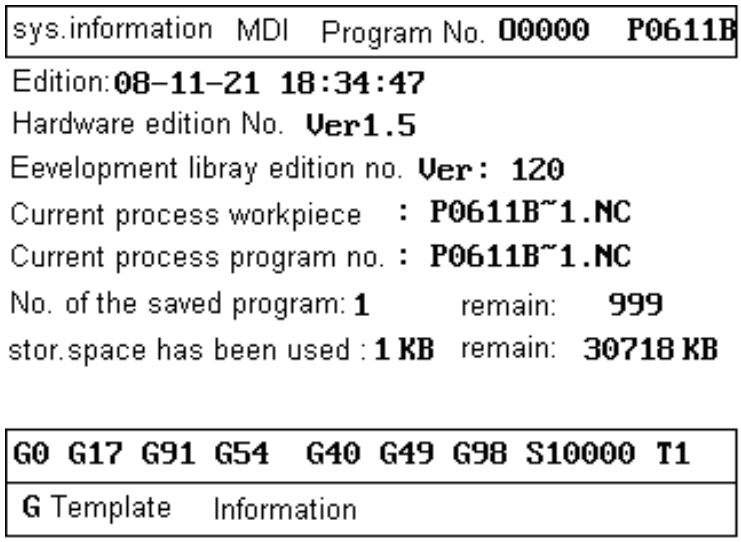
The modify function can only be used for editing the instruction words. To do that, move the cursor to the targeted position, enter the new instruction word, and the current instruction word at the cursor will be replaced.

Note:

1.To save the file, you should press Reset after all operations.

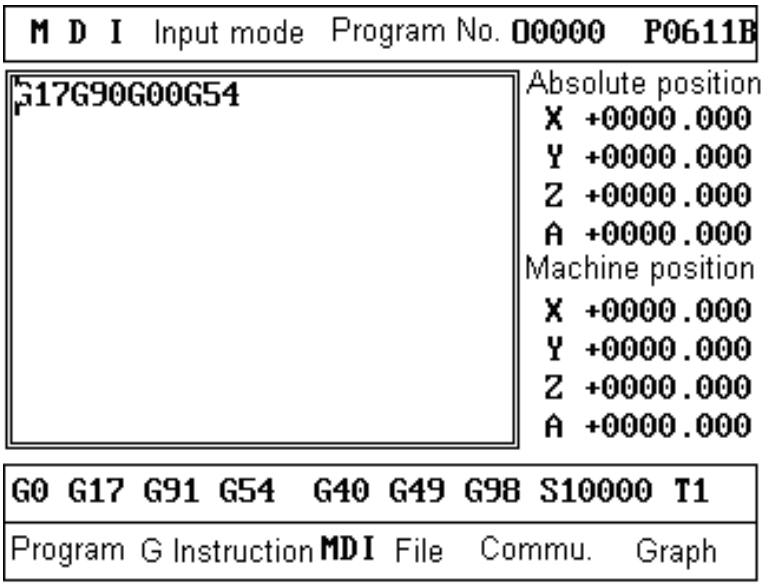
2.As 4240 employs the latest file mapping technology, it can introduce the machining files that exceed its own memory. Therefore, it is specified that machining files greater than 2M can't be edited, but can be retrieved and processed.

4.2.2.2 Information



The system information makes an aggregation display of the program segment of the current processing area and calculate the occupancy resource situation of the current processing area. the right-up of the corner program list picture displays the information of the current controller software edition. If our tenicians want to confirm the software edition of the controller, please keep down the edition information and give back to our company.

4.2.2.3 MDI

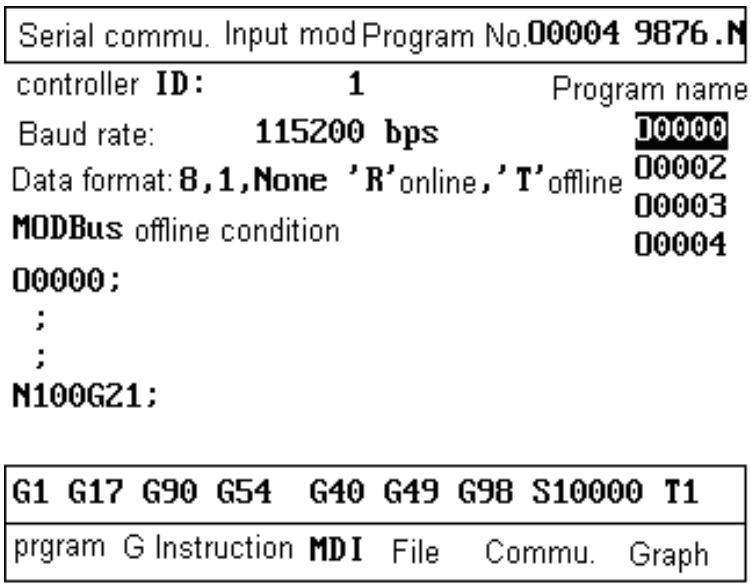


The MDI mode is mainly used for executing G codes at some occasions. The MDI interface is an user-interaction interface that executes single-segment G codes. To enter the NC instructions to the corresponding places, enter the full NC code characters and press Startup.

MDI interaction interface is shown below:

The MDI interactive interface is as follows:

4.2.2.4 Communication



To allow the System to be compatible with the former transmission modes, this traditional RS232 transmission means is still maintained. In order to make it compatible with the early transfers mode, we still reserve this traditional RS232 transfers mode(serial interface communication); In order to guarantee our correctness of the transfer, we adopt the modbus protocol to ***shake hands***, so need the serial interface software provided by our company;

In addition, 4240 controller fully exerts the communication advantage, can utilize the RS485 bus for the multi tools communication, namely one host computer do the file management of multiple 4240 controller, just only need a 485 conversion equipment; For the transfers distance, theoretical value can reach 1KM, so it is really suitable for some controller processing area need the centralized management. If it is just one-on-one, we suggest utilize the RS232 wire connects to the computer for communication provided by our company, it is easy and convenient.

Notice:

Because we adopt the protocol packing, our transfers speed will decrease along with the scene interfering situation. If the file is very big(more than 2M), we do not suggest this method to transfer, can use the USB wire or U-disk to transfer.

4.2.2.5 File Management

File Management:

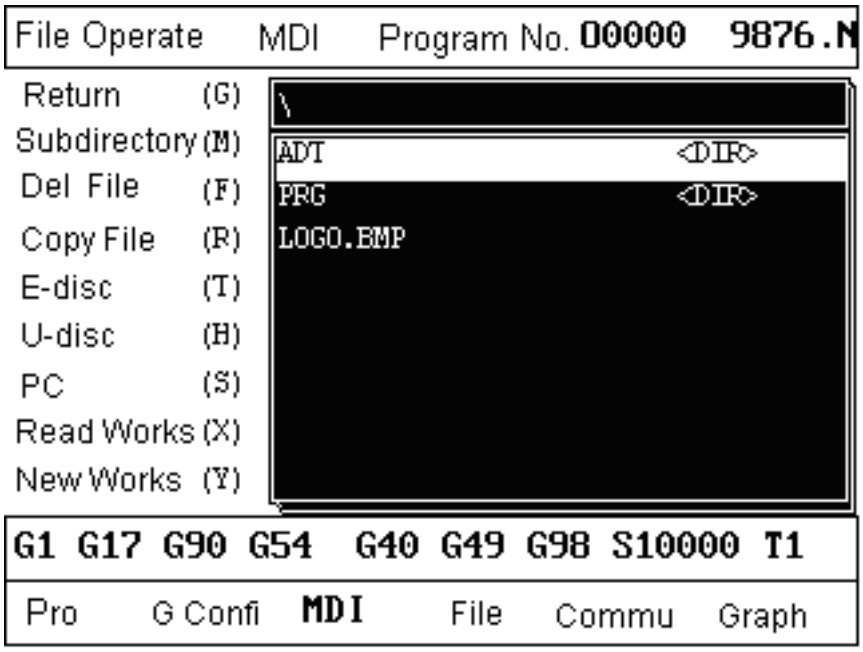
1 Connect to U disc and copy files between U disc and electronic discs.

2 Connect to PC and use PC to copy files from electronic discs.

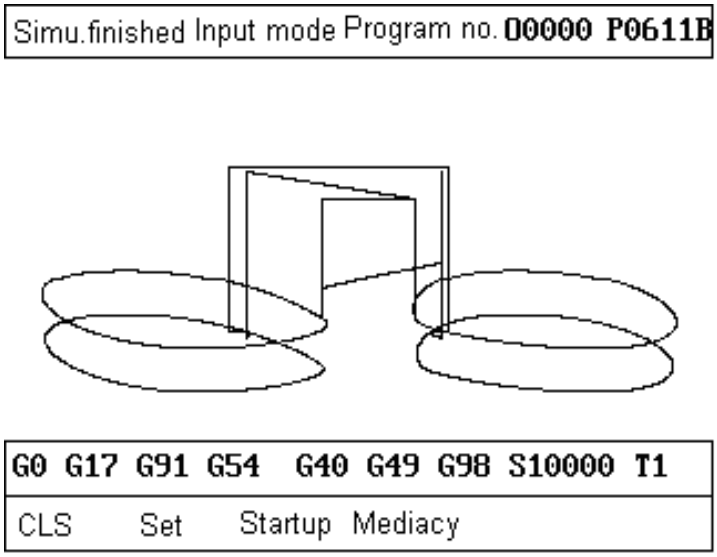
3 To upgrade the software system, you can use the aforesaid two connection modes to upgrade the software of controller.

4 Soft start the controller. Under the file management mode, press reset to re-start the controller. This differs from the restart of power interruption. In some cases, you can restart the controller and activate some function faster.

The interface of file management is shown below:



4.2.2.6 Graph Emulation

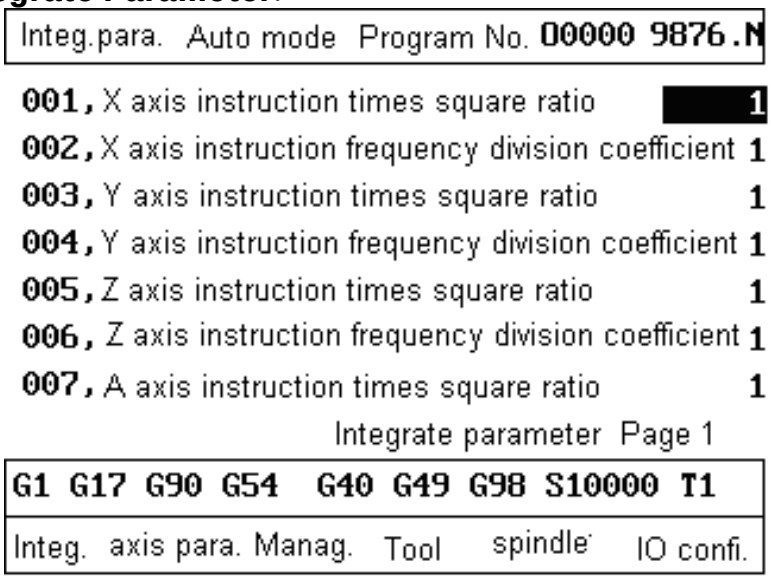


The Graph Emulation is used for displaying the emulated NC machining program. To display the emulation, enter the graph menu and pres Startup. The parameters of the graph emulation are used for adjusting the displayed central position of coordinate and scaling, and to show the margin position.

The Graph Emulation is only the approximate path of the machining plane can be displayed, and the grammatical errors in the program can't be found. Thus it can't replace the inspection mode for NC program. The interface of graph emulation is shown below:

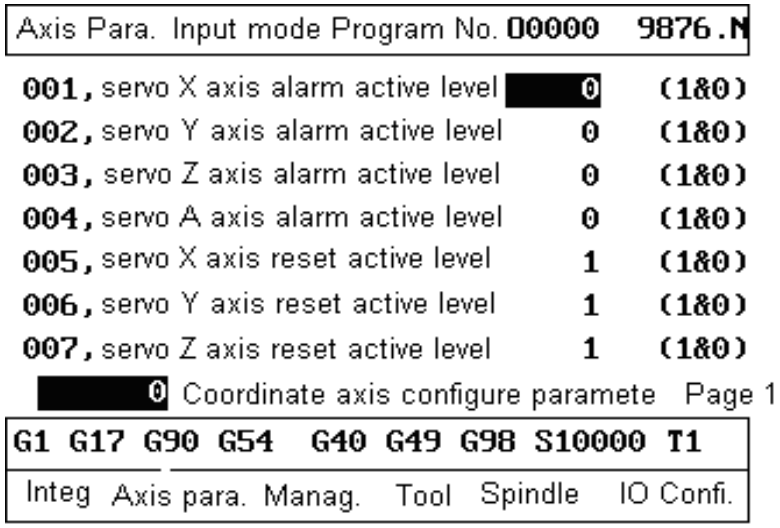
4.2.3 Parameter Interface

Integrate Parameter:



It is a set aggregation for system functions that are not classified in details. For example, the zeroing method and manual speed.

Axis parameter:



The axis parameter is the parameter aggregate of the interface characteristic of the controlling position axis.

Management Parameters:

It is also a functionality aggregation for confirming the identity and initializing the system.

Tool magazine parameter:

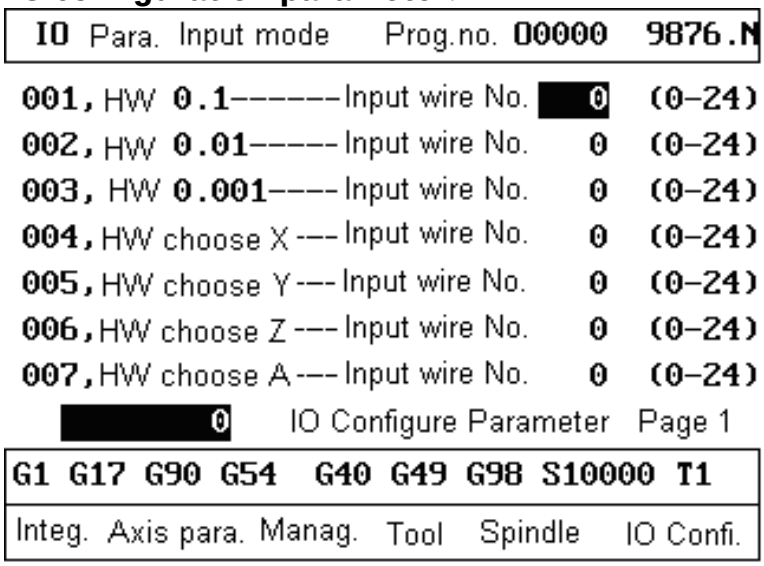
Tool magazine parameter, Tool magazine parameter aggregates integrant parameters of some familiar tool magazine, the specific meaning of the parameter needs

the tool magazine of the tool manufacturer to decide, so its real meaning needs to refer to the specification documents provided by the tool manufacturer.

Spindle parameter:

Spindle parameter includes some electrical characterisric setting function aggregate. The specific application depends on the spindle type of the tool manufacturer, the meaning of the servo parameter and the axis parameter is the same, so we can refer to the specification of the axis parameter.

IO configuration parameter:

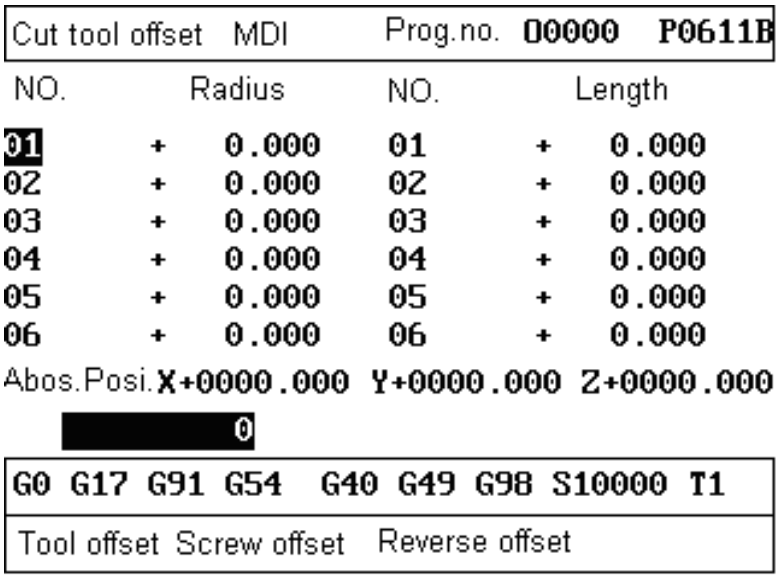


IO configuration parameter is the distribution setting of the hardware interface. This parameter aggregate appoints output and input crus sequence of the system IO function number, improve the system flexibility.

The specific parameter meaning can refer to the parameter in chapter six.

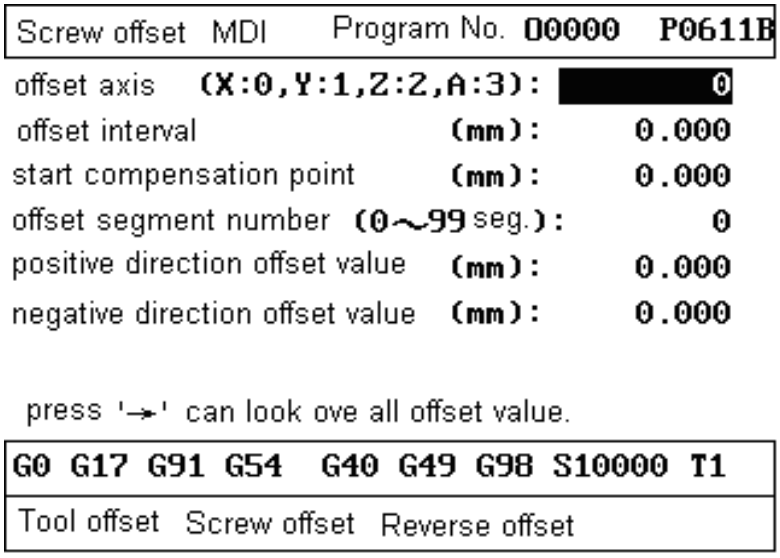
4.2.4 Cutting tools compensation parameters picture(cutting tools offset)

Cutter offset:



Cutting tools offset picture includes two kinds of offset variable, the cutting tool length offset and cutting radius offset; corresponding to G43，G44 and G41，G42；after input the offset value into the corresponding offset number, call the offset number in the NC program then achieve the offset. The cutting tools offset number includes 18 sets variable to be setted.

Screw pitch offset:



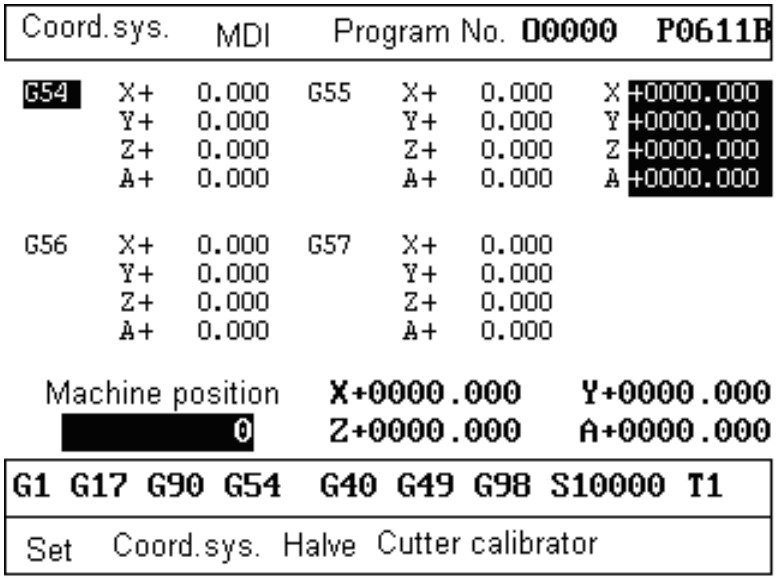
When the interval of the screw pitch is not the same length, then we can add some value in different position segment to offset. The screw pitch includes 99 segments to be setted.

Reverse clearance:

The fit clearance between axis and axis sleeve, can eliminate it according to the reverse clearance offset function.

4.2.5Setting picture of the workpiece coordinate system

workpiece coordinate system



Display the coordinate system of the workpiece, namely the offset between the zero point of the workpiece and the zero point of the tool.

There are G54 ~ G59,G591 ~ G599, six basical workpiece coordinate system and nine extended coordinate system.

Setting of the coordinate auxiliary parameters

Some auxiliary operation parameters when set the coordinate of the workpiece, mainly include the origin offset and the adjusting parameter of the automatic adjusting cutting tools machine.

Origin offset:

The origin offset is used when set the coordinate system, always take the coordinate value of the workpiece adds the this offset as the value of the coordinate value, it will come into effective next time after been setted;

This parameter is used when some workpieces need multiple working procedures, the first proccessing procedure maybe destroy the adjusting position of the workpiece so the next processing procedure can not position a right cutting position, we need a consulting cutting tool adjusting point, the offset from reference point to the real cutting position can be setted in this parameter, so no matter which working procedure is adjusting the cutting tool, just adjust this reference point, namely find the zero point position of the workpiece.

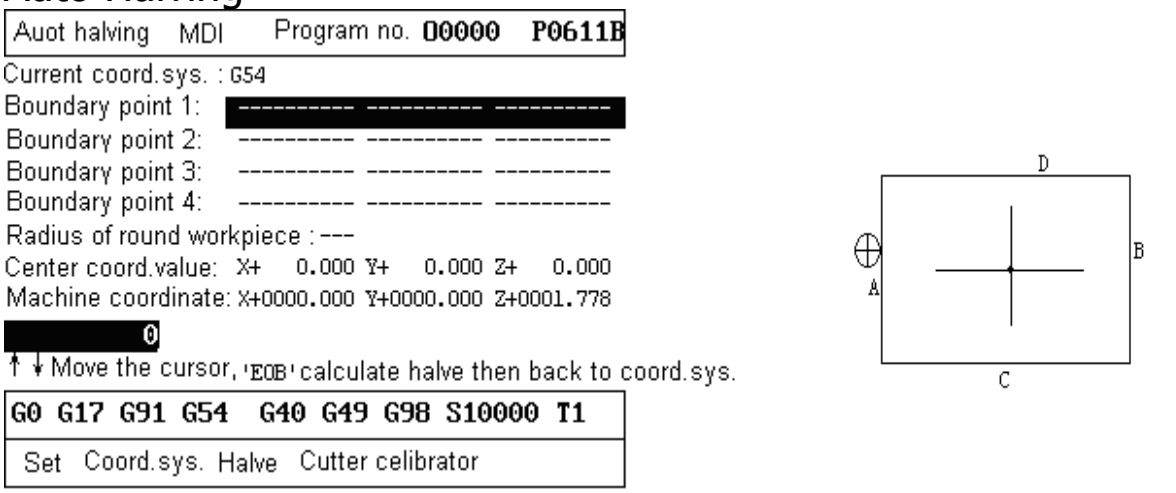
Tool auto-checking instrument, effective signals, tool auto-checking:

The X and Y coordinate of the tool auto-checking instrument is the mechanical coordinate of the tool auto-checking in the machine tool, only set this coordinate properly then the tool auto-checking instrument can finish the self positioning cutting checking properly.

The active level is the setted signal interface level of the tool auto-checking instrument, need to set it according to the real interface of the tool auto-checking instrument.

Auto-checking the cutting tool after change it, after the changing instruction is setted properly the function will do it automatically to improve the efficiency.

Auto Halving



The item Halving performs the function of cutter calibrating, by which the system can automatically compute the central position of the workpiece. Through this item, operations such as line halving, rectangle halving and circle center locating can be conducted.

Single-Axis Halving：

In handwheel mode or manual mode, move the cutter and allow its side blade to touch the surface of side A, then move the cursor to margin point 1 of workpiece and press Insert. Move the cutter and allow its side blade to touch the surface of side B, then move the cursor to margin point 2 of workpiece and press Insert. After the aforesaid operations, the two points of a line have been entered. Then, press EOB, the system will automatically compute the coordinate value of the midpoint of these two points. The computed result will appear in the related column for reference. If it is confirmed, you can press EOB again to return the computed value to the designated coordinate system.

Square Halving：

Similar to single halving, in Square Halving, the X value is obtained from "workpiece margin point 1"and "workpiece margin point 2"to determine the coordinate value of the midpoint along axis X, and Y value is obtained from "workpiece margin point 3"and "workpiece margin point 4"to determine the coordinate value of the midpoint along axis Y. Thus, we can get the coordinate value of the center of a square. Axis Z is not involved in the computation and the original value remains unchanged. After aforesaid operations, press EOB, the system will automatically compute the coordinate value of the midpoint of this rectangle. The computed result will appear in the related column for reference. If it is confirmed, you can press EOB again to return the computed value to the designated coordinate system.

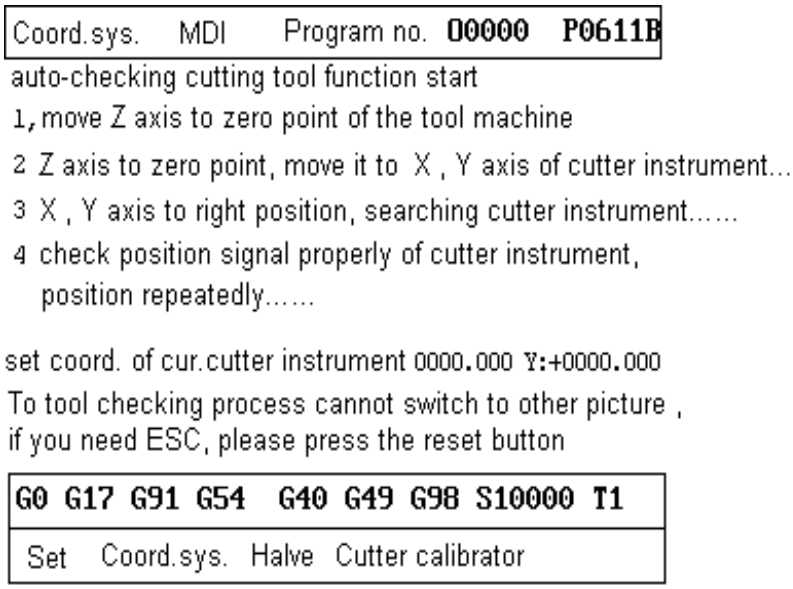
Plane Circle (XY Plane) Halving：

There are two ways for circle halving. One is that three points are used for determining the center of the circle. The other is that two points and the designated radius are used. The determination process will be executed in an intelligent way. If the

user only enters the coordinate values of two points into the items of workpiece margin point, and R is a designated value, the system will automatically compute with these two points and the radius to determine the center of the circle. If the user enters the coordinate values of three points into the items of workpiece margin point, the system will automatically use these three points to determine the center of the circle, and R will be shielded. After aforesaid operations, press EOB, the system will automatically compute the coordinate value of the center of the circle. The computed result will appear in the related column for reference. If it is confirmed, you can press EOB again to return the computed value to the designated coordinate system.

Continuously press EOB twice, and the System will automatically compute the values of the two axes' centers and insert it into the workpiece coordinate system. At the time, the workpiece coordinate system of axis Z should be reset, and mechanical zeroing should be used for cutter calibration.

Auto Cutter Calibrator



Auto Cutter Calibrator is a function menu. Press this menu, the system can automatically execute the cutter calibrating program. In the middle of the process, only after the cutter calibration is finished, can other operations be carried out. Or, you can press Reset to stop calibrating the cutter so as to carry out other operations.

Principle of cutter calibrating:

To allow the cutter to be calibrated, a horizontal reference point of Z phase is set and used. In the process of machining, the System will take advantage of the reference point to self modify the Z value in the current coordinate system after the cutter is changed. Thus the function of cutter calibration is realized.

Operating method:

To calibrate the cutter, you should first set the related parameters. In the setup interface, press F2 to display the parameters for cutter calibration. After setting, press F4 in the same interface and the System will execute the cutter calibration program.

Action sequence:

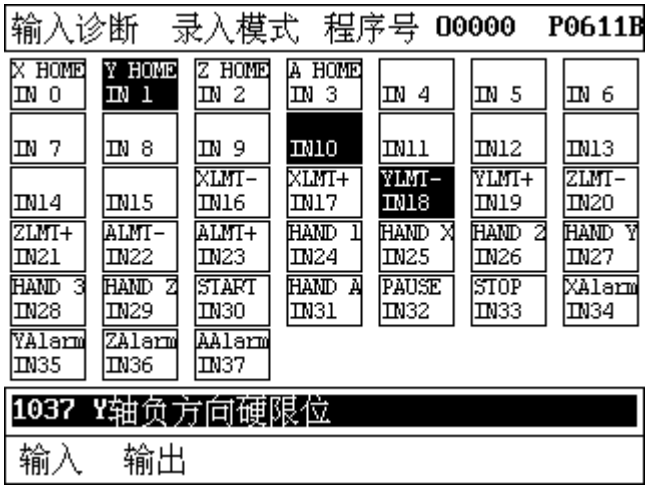
1.First allow axis Z to be zeroed. Then position the spindle to the coordinate X, Y for the cutter calibrator.

2.Axis Z moves downward. It will retract after it meets the inductive switch of the cutter calibrator. Once the inductive switch leaves away, Z axis will move downward at a lower speed. When it meets the switch, the current coordinate value of axis Z in machine tool's coordinate system will be automatically recorded, and the coordinate value of Z will be provided in the currently selected workpiece coordinate system.

3.Axis Z returns to zero position.

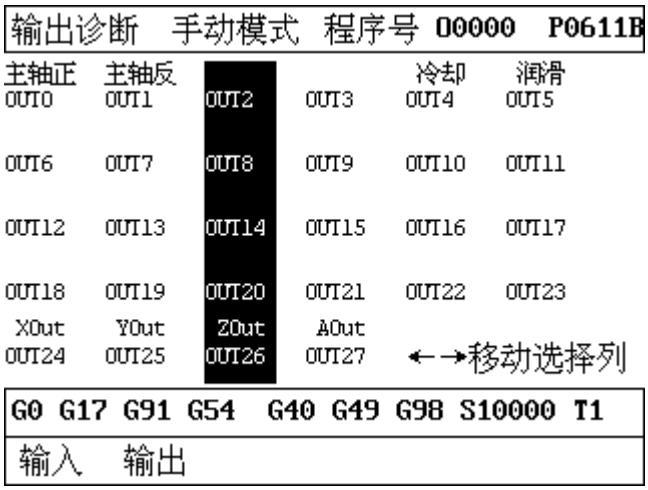
4.2.6 Controller Diagnosis Interface (Diagnosing)

Input:



You can see the current input signal condition in this menu under any mode. If the input IO is in low level, then the corresponding IO will reverse video. The no-load IO is in high level condition.

Output:



Must shift to the hand mode then enter. This function menu can output the IO condition by hand. Press the left and right button to move the cursor to the IO function number you want to operate, then press G，M，F，R，T which is corresponding to the line 1，2，3，4，5 to appoint the IO output, the IO been appointed will reverse video. Under normal condition, the IO with load is in high level condition, it will be in low level after it is appointed.

4.2.7 Macro Variable View Interface (Macro Variable)

This interface shows the view menu for the register of the macro variables. After entering this menu, you can scroll the pages to view the macro variables, or, in MDI mode, you can directly enter the value to the variable register.

4.3 Information on Instructions in Current Mode Status

Display the information on G code mode status of the current system;
For the reason of the screen, mode statuses concerning the following aspects are displayed:

- Mobile command: G00,G01
- Select plane: G17,G18,G19
- Coordinate Logic: G90,G91
- Workpiece coordinate sytem: G54,…G59,G591…G599
- Radius compensation : G40,G41,G42
- Length compensation: G43,G44,G49
- compound instruction backoff plane: G98,G99
- Speedrate spindle: S
- Cutter No.: T

5 Parameter

The parameters are described as follows on the basis of application and functionalities:

Comprehensive parameters
IO Configure Parameters
Manage Parameters
Coordinate set parameters
Graphic emulation parameters

Comprehensive parameters give the relatively full view of the system, including those for basic operations of controller and system setup, such as for spindle, hanwheel, zeroing and cutter warehouse. IO configuration parameters are used when the machine tool is installed and adjusted so as to allow the interfaces of the machine tools and motor driver to be adapted. The parameters of coordinate setting in the Setup menu are used for configuration in cutter calibrating. The graph emulation parameters are prpovided in the item Graph of the menu Program, which are displayed and used for setting the function of graph emulation.

- To modify the parameters, the user's identity must be authenticated. Two levels of user authority are set through the controller, namely, superuser and operator. The superuser can modify all parameters and user passwords. The operator can only handle the parameters which needs to be adjusted in use and modify his own password.
- Based on the nature of application, parameters will become effective either immediately or after the System is restarted.
- Some of the parameters are set in binary system. The binary system can be converted to decimal system through the following method:
  Bit0: Set as 1 to correspond to 1 of decimal system.
  Bit1: Set as 1 to correspond to 2 of decimal system.
  Bit2: Set as 1 to correspond to 4 of decimal system.
  Bit3: Set as 1 to correspond to 8 of decimal system.
  Bit4: Set as 1 to correspond to 16 of decimal system.
  Bit5: Set as 1 to correspond to 32 of decimal system.
  Bit6: Set as 1 to correspond to 64 of decimal system.
  Bit7: Set as 1 to correspond to 128 of decimal system.

If more digits are available, according to the regulation, the decimal value corresponding to the binary value of the last position should be multiplied by 2. So long as the corresponding bit is 1, the set value can be obtained by adding up the corresponding numbers in decimal system according to the comparison table.

For example, If we set Bit 0, Bit 1 and Bit 5 are one, and parameter setting value is one add two add thirty-two equal to 35.

5.1 Parameter index list

Parameter type	No.	Description	Effective way	Default	Page
Integrated parameter（P1.）	001	Multiplication ratio of axis X's instruction	Immediately	1	
Integrated parameter（P1.）	002	Frequency-division coefficient of axis X's instruction	Immediately	1	
Integrated parameter（P1.）	003	Multiplication ratio of axis Y's instruction	Immediately	1	
Integrated parameter（P1.）	004	Frequency-division coefficient of axis Y's instruction	Immediately	1	
Integrated parameter（P1.）	005	Multiplication ratio of axis Z's instruction	Immediately	1	
Integrated parameter（P1.）	006	Frequency-division coefficient of axis Z's instruction	Immediately	1	
Integrated parameter（P1.）	007	Multiplication ratio of axis A's instruction	Immediately	1	
Integrated parameter（P1.）	008	Frequency-division coefficient of axis A's instruction	Immediately	1	
Integrated parameter（P1.）	009	Axis X's fast rate	Immediately	3000	
Integrated parameter（P1.）	010	Axis Y's fast rate	Immediately	3000	
Integrated parameter（P1.）	011	Axis Z's fast rate	Immediately	3000	
Integrated parameter（P1.）	012	Axis A's fast rate	Immediately	3000	
Integrated parameter（P1.）	013	Axis X's start rate	Immediately	100	
Integrated parameter（P1.）	014	Axis T's start rate	Immediately	100	
Integrated parameter（P1.）	015	Axis Z's start rate	Immediately	100	
Integrated parameter（P1.）	016	Axis A's start rate	Immediately	100	
Integrated parameter（P1.）	017	Axis X's acceleration	Immediately	1000	
Integrated parameter（P1.）	018	Axis Y's acceleration	Immediately	1000	
Integrated parameter（P1.）	019	Axis Z's acceleration	Immediately	1000	
Integrated parameter（P1.）	020	Axis A's acceleration	Immediately	1000	
Integrated parameter（P1.）	021	Axis X's positive soft limit	Immediately	＋9999.999	
Integrated parameter（P1.）	022	Axis X's negative soft limit	Immediately	－9999.999	
Integrated	023	Axis Y's positive soft limit	Immediately	＋	

parameter（P1.）				9999.999	
Integrated parameter（P1.）	024	Axis Y's negative soft limit	Immediately	－9999.999	
Integrated parameter（P1.）	025	Axis Z's positive soft limit	Immediately	＋9999.999	
Integrated parameter（P1.）	026	Axis Z's negative soft limit	Immediately	－9999.999	
Integrated parameter（P1.）	027	Axis A's positive soft limit	Immediately	＋9999.999	
Integrated parameter（P1.）	028	Axis A's positive soft limit	Immediately	－9999.999	
Integrated parameter（P1.）	029	Feed speed	Immediately	3000	
Integrated parameter（P1.）	030	Start feeding speed	Immediately	200	
Integrated parameter（P1.）	031	acceleration of feed	Immediately	500	
Integrated parameter（P1.）	032	Axis X's reverse gap compensation	Immediately	0	
Integrated parameter（P1.）	033	Axis Y's reverse gap compensation	Immediately	0	
Integrated parameter（P1.）	034	Axis Z's reverse gap compensation	Immediately	0	
Integrated parameter（P1.）	035	Axis A's reverse gap compensation	Immediately	0	
Integrated parameter（P1.）	036	Zero	Immediately	0	
Integrated parameter（P1.）	037	IO level filtering (restart)	Immediately	0	
Integrated parameter（P1.）	038	Manual speed	Immediately	1000	
Integrated parameter（P1.）	039	Maximum feed speed.	Immediately	6000	
Integrated parameter（P1.）	040	reserve	Immediately	0	
Integrated parameter（P1.）	041	Handwheel speed	Immediately	9000	
Integrated parameter（P1.）	042	Retain time of M code	Immediately	100	
Integrated parameter（P1.）	043	x axis zero pulse offset	Immediately	0	
Integrated parameter（P1.）	044	Y axis zero pulse offset	Immediately	0	
Integrated parameter（P1.）	045	Z axis zero pulse offset	Immediately	0	
Integrated	046	A axis zero pulse offset	Immediately	0	

parameter（P1.）					
Integrated parameter（P1.）	047	Incremental Line number	Immediately	0	
Integrated parameter（P1.）	048	system baud rate(restart)	Immediately	115200	
Integrated parameter（P1.）	049	ID number of controller(restart)	Immediately	1	
Integrated parameter（P1.）	050	X-axis direction of zero	Immediately	1	
Integrated parameter（P1.）	051	Y-axis direction of zero	Immediately	1	
Integrated parameter（P1.）	052	Z-axis direction of zero	Immediately	0	
Integrated parameter（P1.）	053	A-axis direction of zero	Immediately	0	
Integrated parameter（P1.）	054	arc-circle interpolation of feed value	Immediately	0.2	
Integrated parameter（P1.）	055	G73 cutter retract value in cycle	Immediately	2	
Integrated parameter（P1.）	056	G83 cutter retract value in cycle	Immediately	2	
Integrated parameter（P1.）	057	arc-circle interpolation of acceleration value	Immediately	100	
Integrated parameter（P1.）	058	speed interpolation mode	Immediately	0	
Integrated parameter（P1.）	059	Pretreatment mode of code	Immediately	0	
Integrated parameter（P1.）	060	CNC item File Scan	Immediately	1	
Integrated parameter（P1.）	061	Frequency simulation amount control mode	Immediately	1	
Integrated parameter（P1.）	062	X-axis speed of zero	Immediately	1000	
Integrated parameter（P1.）	063	Y-axis speed of zero	Immediately	1000	
Integrated parameter（P1.）	064	Z-axis speed of zero	Immediately	1000	
Integrated parameter（P1.）	065	A-axis speed of zero	Immediately	1000	
Integrated parameter（P1.）	066	safe signal checking efficient level	Immediately	0	
Integrated parameter（P1.）	067	air pressure signal checking efficient level	Immediately	0	
Integrated parameter（P1.）	068	feed signal checking efficient level	Immediately	0	
Integrated parameter（P1.）	069	starting set of the lubricant pressure timing	Immediately	0	
Integrated parameter（P1.）	070	keeping time set of the lubricant pressure	Immediately	0	
Axis parameter（P2.）	001	servo X axis alarming efficient level	Immediately	0	

Axis parameter （P2.）	002	servo Y axis alarming efficient level	Immediately	0	
Axis parameter （P2.）	003	servo Z axis alarming efficient level	Immediately	0	
Axis parameter （P2.）	004	servo A axis alarming efficient level	Immediately	0	
Axis parameter （P2.）	005	servo X axis replacement efficient level	Immediately	1	
Axis parameter （P2.）	006	servo Y axis replacement efficient level	Immediately	1	
Axis parameter （P2.）	007	servo Z axis replacement efficient level	Immediately	1	
Axis parameter （P2.）	008	servo A axis replacement efficient level	Immediately	1	

5.2 Integrative parameters（P1.）

001	Multiplication ratio of axis X's instruction（**X_CMR**）
002	Frequency-division coefficient of axis X's instruction（**X_CMD**）
003	Multiplication ratio of axis Y's instruction（**Y_CMR**）
004	Frequency-division coefficient of axis Y's instruction（**Y_CMD**）
005	Multiplication ratio of axis Z's instruction（**Z_CMR**）
006	Frequency-division coefficient of axis Z's instruction（**Z_CMD**）
007	Multiplication ratio of axis A's instruction（**A_CMR**）
008	Frequency-division coefficient of axis A's instruction（**A_CMD**）

Effective range : 1～65535

Unit : Non

User : Upon operating administrators

Initialization : 1

Effective time : Immediately

Explain : When lead screws with different screw pitches are configured with motors of various step angles, or with servo motors of different pulse number per round, or connections are realized through different gears, the programmed values can remain consistent with the actual moved distance by setting the parameter of the electronic gear ratio of the system.

CMR/CMD =P/（L×1000）

CMR: Numerator of gear ratio

CMD: Denominator of gear ratio

P: Pulse number per motor round

L: Moved distance per motor round (mm)

The value of CMD/CMR is the pulse equivalent, which tells the moved distance per pulse, with its unit as 0.001mm.

Example 1: The motor rotates one circle very 5000 pulses, after which the machine tool moves 5mm, then:

CMR/CMD=5000/（5*1000）=1/1

That is to say, we can set the values as: CMR=1, CMD=1. Here, the pulse equivalent is 0.001mm.

Example 2: The motor rotates one circle very 5000 pulses, after which the machine tool moves 10mm.

CMR/CMD=5000/（10*1000）=1/2

That is to say, we can set the values as: CMR=1, CMD=2. Here, the pulse equivalent is 0.002mm.

009	Axis X's fast rate
010	Axis Y's fast rate
011	Axis Z's fast rate
012	Axis A's fast rate
013	Axis X's start rate
014	Axis Y's start rate
015	Axis Z's start rate
016	Axis A's start rate
017	Axis X's acceleration
018	Axis Y's acceleration
019	Axis Z's acceleration
020	Axis A's acceleration

Effective range : 1～9999，1～9999，1～8000

Unit : mm/min,mm/min,mm/sec

User : Upon operating administrators

Initialization : 3000,200,1500

Effective time : Immediately

Explain : This parameter is used for setting the data of trapezoidal acceleration and deceleration, which acts on GOO instruction.

When the stepping motor is used, it is recommended the start rate be set as 1-2 round/s. As discussed above, the machine tool will move 5mm when the motor rotates for one circle. The rate of 1-2 round/s equals 5-10mm/s. After this rate is converted into mm/min, the start rate can set as 300-600mm/min. In the case of servo motor, the start rate can be set to the degree that no vibration occurs when the motor is started and stopped. If the speed is too high, the equipment will vibrate during running and the stepping motor may lose steps.

The handwheel speed and returning to zero speed is also influenced by the acceleration and starting speed in some non-interpolation locomotion occasion

021	Axis X's positive soft limit
022	Axis X's negative soft limit
023	Axis Y's positive soft limit
024	Axis Y's negative soft limit
025	Axis Z's positive soft limit
026	Axis Z's negative soft limit
027	Axis A's positive soft limit
028	Axis A's negative soft limit

Effective range : -9999～9999

Unit : mm

User : Upon operating administrators

Initialization : Max. positive or negative value

Effective time : Immediately

Explain :

Hard limit signals are set in regular lethes. In this case, the soft limit should not be used . The positive limit can be set as +9999.999 and negative limit -9999.999.

If no hard limit switch is installed in the system, the soft limit can be used. The soft limit takes the coordinate system of the machine tool as its base point. Both the positive and negative limits are benchmarked by the actual distance, with the unit as mm.

As the soft limit employs the mode that the system decelerates for stop once the limit point is reached, it is likely that the movement may exceed the set value a little bit. The exceeded distance is associated with the accelerate time and speed.

029	Feeding speed
030	Start feeding speed

031	Acceleration of feed
039	Max. feed rate

Effective range	:	1～9999，1～9999，1～8000，1～9999
Unit	:	mm/min,mm/min,mm/sec,mm/min
User	:	Upon operating administrators
Initialization	:	3000,200,1000,3000
Effective time	:	Immediately
Explain	:	In the executing instruction G01, G02 and G03, the system moves at the rate designated by F. If instruction F isn't designated in the program, the system will move at the rate given this parameter for executing the above instructions. If instruction F is designated, this parameter will become ineffective. The item of max. feedrate can play a role in restricting the instruction F in machining. In other words, no matter how great the set value of instruction F is, the actual rate can't exceed the set value of this parameter. This parameter can avoid damages on the system caused by the unexpected rate programming errors when the machining files are invoked.

032	Axis X's reverse gap compensation
033	Axis Y's reverse gap compensation
034	Axis Z's reverse gap compensation
035	Axis A's reverse gap compensation

Effective range	:	1～20000
Unit	:	Pulse
User	:	Upon operating administrators
Initialization	:	0
Effective time	:	Immediately
Explain	:	Compensating the coordination gap of the machining shaft.

036	Zero

Effective range	:	0～1
Unit	:	Non
User	:	Upon operating administrators
Initialization	:	0(Program zero)
Effective time	:	Immediately
Explain	:	0 Program zero 1 Mechanical zero Program zero means the system is zeroed when the coordinate value becomes zero. To realize mechanical zero, the external inspection switch is needed to position the zero point. To that effect, the system moves to the set zeroing direction at fast speed, and moves back at low speed when the signal is detected. Then, the system will advance slowly after disconnected upon the detection of signal, and the zeroing process ends till the signal becomes effective again. When the switch is enabled to start by the inspection of servo phase Z in the IO configuration parameters, the mechanical zero mode will automatically start phase Z to position the zero point after the signal from the inspection device reaches.

037	IO filter grade(restart)

Effective range	:	0～8
Unit	:	Non
User	:	Super administrator
Initialization	:	0
Effective time	:	go into effect after restart
Explain	:	set the wave filter constant : If the interfering is grave in the current surrounding, such as raining and thundering influence the induction switch, you can set a wave filter value, the bigger of the value, the longer of the checking time, the more the reliability; 0 means that do not filter the wave;

038	Manual speed
041	Handwheel speed

Effective range : 1～9999

Unit : mm/min

User : Upon operating administrators

Initialization : 1000，9000

Effective time : Immediately

Explain : Set the manual speed, handwheel speed.

In this mode, the start rate is set via the following parameters: 013，014，015，016，017，018，019，020；

042	Retain time of M Code

Effective range : 1～9999

Unit : ms

User : Upon operating administrators

Initialization : 100

Effective time : Immediately

Explain : Set the retaining time after M code is executed, with the unit as millisecond.

043	X-axis return to reference point of coordinate
044	Y-axis return to reference point of coordinate
045	Z-axis return to reference point of coordinate
046	A-axis return to reference point of coordinate

Effective range : -9999～9999

Unit : Pulse

User : Upon operating administrators

Initialization : 0

Effective time : Immediately

Explain	:	Set the compensation zero offset amount after the axis return to zero, unit pulse. The specific process is do the mechanical zero returning first then deviate corresponding pulse, set this point as the mechanical zero point. Notice: when the program is returning to the zero point, this parameter is void.

047	Incremental Line number

Effective range	:	0～64
Unit	:	Non
User	:	Upon operating administrators
Initialization	:	0
Effective time	:	Immediately
Explain	:	When G code is edited manually, an Nxxxxx line number will be automatically added at the time of line feed. When set as 0, it means this function is disabled.

048	system baud rate(restart)

Effective range	:	9600～115200
Unit	:	Non
User	:	Upon operating administrators
Initialization	:	115200
Effective time	:	restart
Explain	:	DNC or other upper computer software communicate with this controller in RS232 way, the communication speed setting adopted.

049	The ID number of controller (restart)

Effective range	:	1～255
Unit	:	Non
User	:	Upon operating administrators
Initialization	:	1
Effective time	:	restart

Explain : DNC or other upper computer software communicate with this controller via MODBUS protocol, the ID number setting of this controller.

050	X-axis direction of zero
051	Y-axis direction of zero
052	Z-axis direction of zero
053	A-axis direction of zero

Effective range : 0～1

Unit : Non

User : Upon operating administrators

Initialization : 1,1,0,0

Effective time : Immediately

Explain : Zeroing direction for each machining shaft is set.

0 positive direction of zero

1 negative direction of zero

054	arc-circle interpolation of feed value

Effective range : 0～1

Unit : mm

User : Upon operating administrators

Initialization : 0.2

Effective time : Immediately

Explain : Split equivalent for arc is set.

If the this value is set too small, the approaching precision of the arc will be quite high. However, this will bring about great amount of computation, which can cause pauses obviously. In result, the machining effect will be influenced.

055	G73 cutter retract value in fixed cycle
056	G83 cutter retract value in fixed cycle

Effective range : 0.1～100

Unit : mm

User	:	Upon operating administrators
Initialization	:	2.000
Effective time	:	Immediately
Explain	:	The cutter retract value for instruction G73 and G83 after value Q is fed is set. This parameter is set on the basis of the actual effect of chip discharge. The defaulted value is 2mm.

057	arc-circle interpolation of acceleration value

Effective range	:	10～500
Unit	:	mm/sec
User	:	Upon operating administrators
Initialization	:	100
Effective time	:	Immediately
Explain	:	The acceleration equivalent for arc is set. If this value is set too small, the acceleration will be slow. This parameter should be set as high as possible in accordance with the size of arc.

058	speed interpolation mode

Effective range	:	0～1
Unit	:	Non
User	:	Upon operating administrators
Initialization	:	0（acceleration）
Effective time	:	Immediately
Explain	:	In the ineffective pretreatment mode (059 is set as 0), this parameter determines the acceleration and deceleration modes of instruction G01. 0: trapezoidal acceleration and deceleration; 1: uniform speed.

059	Pretreatment mode of code

Effective range	:	0～1
Unit	:	Non

User : Upon operating administrators

Initialization : 0（Real-time machining）

Effective time : Immediately

Explain : 0 Real-time machining. Suitable for commissioning.
1 Pretreatment mode. Two seconds will be buffered for pre-reading after the System enters the machining state. In this pretreatment mode, only the direction and size of the feeding line segment can be judged. Therefore, the speed can be automatically adjusted to optimize the speed of automatic running.

In pretreatment, Single Segment executes to disable the running. Press Single Segment, you can switch over among the following modes: Handwheel, Startup and Program Inspection.

060	CNC item File Scan

Effective range : 0～1

Unit : Non

User : Upon operating administrators

Initialization : 1

Effective time : Immediately

Explain : The item File Scan can improve the invoking speed when the file size increases.

When NC files are invoked, the System needs to scan files from the beginning to the end to position each program segment. Therefore, if the file has only one program segment and big size, more time may be spent on waiting. Turn this item off, and the System will only scan the first program segment address and then exit.

061	Frequency control mode

Effective range : 0～1

Unit : Non

User : Upon operating administrators

Initialization : 0

Effective time : Immediately

Explain	:	Output of the corresponding S code. 0 Output of analog quantity 1 Segment rate regulating (four segments), see below: OUT23------S0 OUT22------S1 OUT21------S2 OUT20------S3 When the analog quantity is output, the maximum value is V＝S/MaxRPM S is the revolution value setted for the user, but MaxRPM is the maximum revolution value of the spindle setted for the parameter （P4.017）; S code corresponds to the set value of the parameter 048. When the switching quantity is output, it will be output according segment 0-15. And the value of S code is also limited to 0-15.

062	returning to zero speed of axis X
063	returning to zero speed of axis Y
064	returning to zero speed of axis Z
065	returning to zero speed of axis A

Effective range	:	0～9999
Unit	:	mm/min
User	:	Upon operating admnistrator
Initialization	:	1000
Effective time	:	Immediately
Explain	:	set the speed of returning to zero of each axis separately

066	Safe signal checking efficient level

Effective range	:	0～1
Unit	:	LOGIC VOLTAGE LEVEL
User	:	Upon operating administrator
Initialization	:	0
Effective time	:	Immediately

Explain	:	Setting the effective level of the system safe signal, the user can define the origin of the safe signal, normally it is the door of the electrical control box such kind of sensitive occasion. If there are some places should consider the safety, you can connect the signals parallelly and then connect it to the safe signal detection foot of the system. Safe signal considers the convenience of the maintenance, only when the system start and process then it will do the safety examine, it will not alarm when it is in the non-running condition.

067	Air pressure signal checking efficient level

Effective range	:	0～1
Unit	:	LOGIC VOLTAGE LEVEL
User	:	Upon operating administrator
Initialization	:	0
Effective time	:	immediately
Explain	:	Set the effective level of the system air pressure alarming.Air pressure and the urgent-stop alarming are the same, they are global effective.

068	Feed signal checking efficient level

Effective range	:	0～1
Unit	:	LOGIC VOLTAGE LEVEL
User	:	upon operating administrator
Initialization	:	0
Effective time	:	Immediately
Explain	:	set the effect level of the system nip material alarming; the nip material alarming is checking during the system running process.

069	starting set of the lubricant pressure timing (min）
070	keeping time set of the lubricant pressure timing (sec)

Effective range	:	
Unit	:	
User	:	upon operating administrator

Initialization	:	0
Effective time	:	Immediately
Explain	:	➢ set the start by set date parameter and the holding time parameter of the system automatic pump. ➢ The time opening setting parameter will time after the system starts, the unit is minute, when it comes to the time has been setted before do the pump output(OUT10) ➢ The output signal will stop after it keeps the keeping seconds setted in the P1.070 parameter.(reversed phase) ➢ Output the low electric level is efficient.

5.3 Axis parameter configuration（P2.）

001	Signal that indicates alarm of servo axis X is effective
002	Signal that indicates alarm of servo axis Y is effective
003	Signal that indicates alarm of servo axis Z is effective
004	Signal that indicates alarm of servo axis A is effective
005	Signal that indicates resetting of servo axis X is effective
006	Signal that indicates resetting of servo axis Y is effective
007	Signal that indicates resetting of servo axis Z is effective
008	Signal that indicates resetting of servo axis A is effective

Effective range : 0～1

Unit : LOGIC VOLTAGE LEVEL

User : Super Administrators

Initialization : 0，1

Effective time : Immediately

Explain : To match the interface parameters of the servo driver. Details for setting the parameters are subject to the electrical level of the interface for the servo.

009	To enable phase Z inspection at servo axis X
010	Signal that indicates phase Z of servo axis X is effective
011	To enable phase Z inspection at servo axis Y
012	Signal that indicates phase Z of servo axis Y is effective
013	To enable phase Z inspection at servo axis Z
014	Signal that indicates phase Z of servo axis Z is effective
015	To enable phase Z inspection at servo axis A
016	Signal that indicates phase Z of servo axis A is effective

Effective range	:	0～1
Unit	:	LOGIC VOLTAGE LEVEL
User	:	Super Administrators
Initialization	:	0
Effective time	:	Immediately
Explain	:	After this parameter is set and used, the System will automatically use phase Z for locating in mechanical zero mode. It is called "servo zero" localization. Under this mode, the precision of repeated returning to zero positioning depends on the precision of the servo positioning, so normally we suggest start this function when use the servo. Because the stepper motor do not has the coder, it can not start this function, or there will be a fault when the tool return to zero it can not find signal.

017	X-axis positive limit of hardware(restart)
018	X-axis negative limit of hardware(restart)
019	X-axis limit the effective signal(restart)
020	Y-axis positive limit of hardware(restart)
021	Y-axis negative limit of hardware(restart)
022	Y-axis limit the effective signal(restart)
023	Z-axis positive limit of hardware(restart)
024	Z-axis negative limit of hardware(restart)
025	Z-axis limit the effective signal(restart)
026	A-axis positive limit of hardware(restart)
027	A-axis negative limit of hardware(restart)
028	A-axis limit the effective signal(restart)

Effective range	:	0～1
Unit	:	Non

User	:	Super Administrators
Initialization	:	0
Effective time	:	Take effect after restart
Explain	:	There are two kinds of modes of hard positive stop, one is hardware response mode, the other is software scanning mode;

- The hardware response mode is self-bring of the sports chip, it is triggered by the effective level of the circuit examination limiting displacement foot, so the real time is very high, but it also brings a disadvantage, when it is disturbed heavily by the outside condition, it will disturb pulse, the system will not alarm due to it has no time to read the fault, it will cause the false appearance of pulse lose; So when using this function normally requires constant-closed wire connection of the connection switch, namely effectiveness is high level; Considering the complexity of the field environment, the default is shuted dowm.
- The system is equipped with the scan mode and it can no be screened. The scan mode adopts the appointing function visiting input signal, it adopts software anti-jamming examine technology and is able to estimate if whether it is the limiting displacement or disturbing no-action. This need some time to estimate, so real time effective is not better than the stop-type limiting displacement. But most of the time (when the processing speed is 10mm/min), it can satisfy the safety examination.
- The hardware response function of the hardware limiting displacement take precedence of scan response function, namely if the hardware response starts, it will quicken the response speed, noteworthiness, the hardware response function only can stop the pulse with immediate-stop mode, so when the speed is very fast, the effectiveness of immediate-stop may cause the vibration of the tool. But the software scanning mode adopt the mode of maximum acceleration decelerating stop, decelerate according to the maximum speed of each axis set by user(parameter P2.074~077), so will be some overshoot phenomenon.

029	set the axis X pulse command format(restart)
030	set the axis Y pulse command format(restart)
031	set the axis Z pulse command format(restart)
032	set the axis A pulse command format(restart)

Effective range : 0～1

Unit : Non

User : Super Administrators

Initialization : 1

Effective time : Restart

Explain : the setting of the pulse command format is the mode of configuration input pulse, need to know the command format received by the motor driver in advance.

0 pulse+pulse

1 pulse+direction

033	Logic direction of X pulse(restart)
034	Logic direction of Y pulse(restart)
035	Logic direction of Z pulse(restart)
036	Logic direction of A pulse(restart)

Effective range : 0～1

Unit : Non

User : Super administrators

Initialization : 1

Effective time : Restart

Explain : After the direction of pulse is set, if the direction of controller is the opposite of the actual driver, this parameter can be modified to adjust the rotate direction of motor.

037	external zero of X effective signal
038	external zero of Y effective signal
039	external zero of Z effective signal
040	external zero of A effective signal

Effective range : 0～1

Unit : LOGIC VOLTAGE LEVEL

User : Super administrators

Initialization : 0

Effective time : Immediately

Explain : Set the signal for effective electrical level of the external zero switch when the system is zeroed.

0 Low electric level

1 High electric level

041	set value of axis X ROUND (restart)
042	set value of axis Y ROUND (restart)
043	set value of axis Z ROUND (restart)
044	set value of axis A ROUND (restart)

Effective range : 0～9999999

Unit : Pulse

User : Super administrator

Initialization : 0

Effective time : restart

Explain :

- The loop function only exists in the hardware edition 1.5 or above.
- This functoin can prevent the overflow error that the axis logic counting exceeds the maximum counting range（2147483648）.
- Normally the overflow will happen if set the axis as the rotary axis, so after the system get the

P2.062~P2.069 parameter, if it finds user setted the current axis as the rotary axis and adopt 360° displaying mode, then the system calculate the the corresponding pulse threshold value according to the gear ratio of the current axis and endue with the round parameter of the corresponding axis, the user can see the the change of the parameter after start the rotary axis displaying function. User can amend the parameter after been changed, the final displaying numbers are effective.

- The parameter can be carried out after restart; The executing qualification is the corresponding axis must be setted as the rotary axis and set it as 360°displaying（P2.062~069）;

045	The appointing interface axis number of X axis (restart)
046	The appointing interface axis number of Y axis (restart)
047	The appointing interface axis number of Z axis (restart)
048	The appointing interface axis number of A axis (restart)

Effective range : 0～4

Unit : pulse interface number

User : Super administrator

Initialization :

Effective time : restart

Explain : In the default mode, the real axis number of each coordinate defining axis is corresponded to the silk-screen number of the product shell, when there is some abnormity with some function axis, you can change the axis according to the appointing function. For example, set the P2.045 as 4, set the 4, P2.048 as 1, then the operation of any x axis in the system is the operation of the A axis coding interface in the product shell.

0：have no axis

1～4：corresponding axis one to axes four

049	spindle appoint the interface axis number（restart）

Effective range : 0～4

Unit	:	pulse interface number
User	:	Super administrator
Initialization	:	
Effective time	:	restart
Explain	:	set it as 0 in the default mode, it stands for the main shaft is the frequency conversion controlling mode, namely it is the analog or shift controlling mode, if you want to use the servo main shaft, then it must occupy a coding interface,(the servo main shaft must be the position controlling mode); Then you can amend this parameter and appoint the function. 0 :the frequency conversion main shaft of the analog adjusting 1 ~ 4: correspond to number 1 to number 4 axis **Notice: If you appoint some pulse port as the function port of the main shaft, need to delete the function axis number corresponding to this pulse before or when restart, the system will allocate the main shaft preferentially and the original function will be invalid.**

050	line number of the X axis coder
051	line number of the Y axis coder
052	line number of the Z axis coder
053	line number of the A axis coder

Effective range	:	0～9999
Unit	:	Line number
User	:	Super administrator
Initialization	:	2500
Effective time	:	immediately
Explain	:	set the coder line number connected to each pulse port(AB phase pulse), because it is fourfold dividing frequency when do the intrinsic call, the inputing parameter value is the pulse number of the coder collect one diameter and divided four.

058	axis X pulse logic voltage level (restart)
059	axis Y pulse logic voltage level (restart)

060	axis Z pulse logic voltage level (restart)
061	axis A pulse logic voltage level (restart)

Effective range	:	0～1
Unit	:	LOGIC VOLTAGE LEVEL
User	:	Super administrator
Initialization	:	0
Effective time	:	restart
Explain	:	set the normal level when the pulse is working, if set a level is different from the normal level which the motor driver requires, then at each time of positive and negative movements(it is nothing to do with the pulse amount), there is an accumulative error in one direction. So you find that the machine has the accumulative error in one direction, please notice whether this parameter is not compatible.

062	axis X feature (rotaty 0, linear 1)
063	axis Y feature (rotaty 0, linear 1)
064	axis Z feature (rotaty 0, linear 1)
065	axis A feature (rotaty 0, linear 1)

Effective range	:	0～1
Unit	:	non
User	:	Super administrator
Initialization	:	1
Effective time	:	immediately
Explain	:	set the features of the axis. 0：rotary axis 1：linear axis The setting of this parameter and P2.066 ~ 069 corresponding axis will influence the setting of P2.041 ~ 044, for details please refer to the parameter instructions of P2.041 ~ 044.

066	the rotary display mode of X axis

067	the rotary display mode of Y axis
068	the rotary display mode of Z axis
069	the rotary display mode of A axis

Effective range	:	0～1
Unit	:	non
User	:	Super administrator
Initialization	:	0
Effective time	:	immediately
Explain	:	set the coordinate display mode of axis. This parameter is valid when P2.062～P2.065 is setted as 0. 0：0～360 degree displaying 1：－9999.999～9999.999 degree displaying The setting of this parameter and P2.062～065 corresponding axis will influence the setting of P2.041～044, for details please refer to the parameter instructions of P2.041～044.

070	the rotary route of X axis optimizing
071	the rotary route of Y axis optimizing
072	the rotary route of Z axis optimizing
073	the rotary route of A axis optimizing

Effective range	:	0～1
Unit	:	non
User	:	Super administrator
Initialization	:	1
Effective time	:	immediately
Explain	:	when this parameter is valid when P2.062～P2.065 and P2.066～P2.069 is setted as 0; set whether choose automatic search the shortest route moving, if this axis is rotary axis and it does not process during the positiong course, start this function, it can reduce the moving time.

0: do not optimize the route
1: start the shortest route
Notice: If during the moving course it needs the cutting process, then the optimizing of the shortest route maybe is not the processing trajectory you want.

074	the maximum acceleration of X axis
075	the maximum acceleration of Y axis
076	the maximum acceleration of Z axis
077	the maximum acceleration of A axis

Effective range	:	100～8000
Unit	:	Kpps（Kilo Pulse Per Second）
User	:	Super administrator
Initialization	:	2000
Effective time	:	immediately
Explain	:	set the maximum acceleration of each axis can endure, this setting can influence the pretreatment of the trajectory speed optimizing. When it is setted high, it can quicken the axis response time. You can set it higher according to the features of tool and motor. The return-to-zero function and limiting displacement stop function are also influenced by this parameter. Hard limiting displacement function: the hard limiting displacement adopts software scanning mode, because the hard limiting displacement of the software scanning mode decelerate and stop according to the maximum acceleration of each axis. So setting too high will result in tool vibration and too low will result in too much impulse. The return-to-zero function: the return-to-zero acceleration of each axis all adopt this value.

078	return-to-zero direction of the X servo zero point
079	return-to-zero direction of the Y servo zero point
080	return-to-zero direction of the Z servo zero point

081	return-to-zero direction of the A servo zero point

Effective range : 0～1
Unit : non
User : Super administrator
Initialization : 0
Effective time : immediately
Explain : after start the P2.009～P2.016 servo Z phase enabling parameter, this parameter decide the direction of Z phase search.
0: positive direction
1: negative direction

082	outside zero point enabling of X axis
083	outside zero point enabling of Y axis
084	outside zero point enabling of Z axis
085	outside zero point enabling of A axis

Effective range : 0～1
Unit : non
User : Super administrator
Initialization : 1
Effective time : immediately
Explain : Under the mode of mechanical return-to-zero, this parameter decides whether need search the outside deceleration switch. If this parameter is setted as 0 and P2.009～P2.016 is also setted as 0 (servo Z phase enabling), then under the mechanical return-to-zero mode, the action of return-to-zero is setting the current point as the zero point directly.
0: non
1: have

086	pulse logic direction of X axis coder
087	pulse logic direction of Y axis coder

088	pulse logic direction of Z axis coder
089	pulse logic direction of A axis coder

Effective range	:	0～1
Unit	:	non
User	:	Super administrator
Initialization	:	0
Effective time	:	immediately
Explain	:	When the logic direction the coder gets is opposite to the real moving direction of the axis, you can set this parameter and set in the same direction. 0: positive direction 1: negative direction

5.4 Manager Parameter（P3.）

001	Enter Administrator Mode
002	Modify suppersuser password
003	Modify operation user password

Effective range : Non

Unit : Non

User : Non

Initialization : Non

Effective time : Immediately

Explain :

1. In this menu, enter the password and press Insert. If the password is authenticated and confirmed, the System will enter this user mode.
2. Once the mode successfully entered, this menu will be changed to "exit XXX administrator mode", suggesting entering is successful.
3. In the changed menu, press Insert, the System will exit the administration mode. At the time, the parameters should be modified and the administration mode re-entered.
4. The authority of superuser can modify all passwords, whereas the authority of operator can only modify the password of the operation himself.
5. When the password is 0, it means in this mode the password will not be authenticated. If the parameters need to be modified, you don't have to enter the administrator mode.

004	The comprehensive initialing parameters are the factory defaults.
005	The initializing IO configuration parameters are the factory defaults.

Effective range : Non

Unit : Non

User : Non

Initialization : Non

Effective time	:	Immediately
Explain	:	Only in superuser mode, can the parameter table be initialized.

006	efficacy NORFLASH（restart）

Effective range	:	Non
Unit	:	Non
User	:	Non
Initialization	:	Non
Effective time	:	restart
Explain	:	1. NORFLASH is the storage hardware the parameter saved, if you find it can not store you can use this mode to examine, please notice, this examination will ruin all parameter, so please do the back-up in advance. 2. You can only do the NORFLASH efficacy under the super user mode.

007	Parameter back-up to sysconf.bak
008	Parameter retrieval from sysconf.bak

Effective range	:	Non
Unit	:	Non
User	:	Non
Initialization	:	Non
Effective time	:	immediately
Explain	:	1.You can do the parameter back-up and retrieval under the superuser mode. 2.The parameter back-up is the sysconf.bak file appointed under the root directory of the controller electronic disc, if it has existed a cognominal file in it, then the last back-up will bestrow this file. 3.The bestrow is also carried out under the root directory of the controller electronic disc according to the sysconf.bak file. When it is bestrowed, it will judge whether it belongs to the same parameter edition according to the parameter edition number of the system back-up, if it is not the same edition, then it will

not carry out the retrieval to the parameter.

4.The system will restart automatically after retrieval finished.

5.5 Tool magazine parameter（P4.）

001	cutting tool changing benchmark position X
002	cutting tool changing benchmark position Y
003	cutting tool changing benchmark position Z
004	safe height of cutting tool change
005	the interval of cutting tool change
006	the speed of cutting tool change (1-9000)mm/min
007	system amount of cutting tool
008	the starting number of the cutting tool

Effective range :

Unit :

User : Upon operating administrator

Initialization :

Effective time : Immeidately

Explain : This parameter is setted according to the tool magazine of the tool factory, please refer to the tool machine tool instruction of tool factory.

5.6 Parameter of spindle（P5.）

001	effective level of spindle alarming
002	effective level of spindle reposition
003	Z phase examining enabling of spindle coder
004	Z phase effective level of spindle coder
005	spindle positive limiting displacement(halt enabling)
006	spindle negative limiting displacement(halt enabling)
007	the effective level of spindle limiting displacement
008	pulse format of the spindle
009	logic direction of the spindle pulse
010	the effective level of spindle out zero-point
011	examining enabling of spindle out zero-point
012	set ROUND of spindle (restart)
014	zero-returning deviation pulse number of spindle
015	pulse logic level of spindle
016	rotary display mode of spindle
017	maximum acceleration of spindle
018	returning to zero direction of spindle out zero point
019	return-to-zero direction of spindle servo zero point
021	return-to-zero rev of spindle

Effective range :

Unit :

User : Upon operating administrator

Initialization :

Effective time :

Explain	:	the parameter of servo main shaft is the same as the parameter of the ordinary positioning axis. As long as the main shaft adoptes servo port to control, you can set it according to the axis parameter function.

013	spindle coder line number

Effective range	:	64～9999
Unit	:	non
User	:	Upon operating administrator
Initialization	:	2500
Effective time	:	immediately
Explain	:	The line number the coder received when the spindle rotates a period; It is the same as the ordinary axis coder, only receive AB-phase pulse, the line number of coder must be the pulse number it got then divided 4. The setting of this parameter will influence the tapping command of G74, G84. So should set it properly.

20	the maximum rotating speed of spindle

Effective range	:	1～30000
Unit	:	non
User	:	Upon operating administrator
Initialization	:	24000
Effective time	:	immediately
Explain	:	This setting is used for the output of the calculating controller analog; And suppose the analog of frequency conversion of control is linear control mode; The setting method is that set the rotating speed into this parameter according to the frequency conversion rotating speed corresponding to the analog 10v, since then you just call the rotating speed value, controller will output corresponding analog voltage automatically according to linear proportion.

022	Numerator item of spindle gear ratio (CMR)
022	Denominator item of spindle gear ratio (CMD)

Effective range	:	1～65535

Unit	:	non
User	:	Upon operating administrator
Initialization	:	1
Effective time	:	immediately
Explain	:	If the spindle exists a shift, then set the hardware gear ratio of the shift in this parameter, this parameter has not been used in the standard edition, but maybe be used in some special occasion.

5.7 IO Configuration（P6.）

001	Handwheel 0.1-------- input line NO.
…	…
014	Servo A alarming--------input line NO.

Effective range : 0～24

Unit : Non

User : Super Administrators

Initialization : Manual of the port table

Effective time : Immediately

Explain : 1. the function foot defining of the hand-held box and servo alarming;
2. input 8888, pressing insert means do not carry out mapping, use the line number corresponding to the instruction default. When operates successfully, it will display "----";
3.The inputting value is from 1 to 24: mapping to the IO panel corresponding to the pin.

015	IN0-------- output line NO.
…	…
038	IN23------- output line NO.

Effective range : 1～24

Unit : Non

User : Super administrator

Initialization : Manual of the port table

Effective time : Immediately

Explain : 1.Input the interruption of configure parameter.
2.The interruption number is what the System controls over IO. For example, when signal of X's external zero point is examined, the System will invoke the interruption number IN0 for inspection. In defaulted state, IN0 corresponds to pin 1 of the circuit. Therefore, the System will indirectly examine the input pin No. 1.

The interruption numbers are distributed to the wire numbers in defaulted state according to the matching relation of input port in the User's Manual. As this relation is not fixed, users can designate in these parameters. One interruption number is mapped to one input pin.

3.For example, if 10 is set for parameter 042, when the system is zeroed, it will examine pin 10 for inspecting the X's zero point signal, not examine the input terminal pin 1.

039	OUT0-------- output line NO.
…	…
062	OUT23------- output line NO.

Effective range : 1～24

Unit : Non

User : Super Administrators

Initialization : Manual of the port table

Effective time : Immediately

Explain : Output the interruption of configure parameter.

Similar to the input interruption parameters for configuration, the output interruption numbers and wire numbers should be mapped for configuration.

063		safety signal input interrupt number
064		air pressure signal input interrupt number
065		nip feed signal input interrupt number
066		system oil pump input interrupt number
067		tool checking instrument examining input interrupt number
068		alarming light output interrupt number
069		running light output interrupt number
070		frequency conversion segment 0 shift output interrupt number
071		frequency conversion segment 1 shift output interrupt number
072		frequency conversion segment 2 shift output interrupt number
073		frequency conversion segment 3 shift output interrupt number

074		lubricating output interrupt number
075		cooling output interrupt number
076		spindle positive rotating output interrupt number
077		spindle negative rotating output interrupt number

Effective range : 0～23

Unit : interrupt number

User : Super administrator

Initialization : The port comparison table of instruction

Effective time : Immediately

Explain :

1. the assigned interrupt number corresponding to the system function;

2. the interrupt number is set by the parameter 15 ~ 62, mapping to the appointed line number;

3. If you want to screen this function, input 8888 then press insert, this operation is hidden-type, after operates it successfully, It will display "---------".

6 System alarming

The system alarming is divided into multi-level alarming, so the alarming number has a level classification. As follows:

0 ~ 1023: G code program runs alarming information

1024 ~ 2048: system condition alarming information

6.1 NC Program executing alarming

0000	:	please replace
0001	:	program over
0004	:	cutting tool changing failure
0005	:	cutting tool invalid
0006	:	G program segment repeat error
0007	:	G program segment program number error
0008	:	G7x8x compound cycle command code can not run normally
0009	:	program abend error
0010	:	appoint the M01 code program halt
0011	:	M98 format error
0012	:	call motion executing failure
0013	:	this segment does not need compensation
0014	:	G program segment invalid format
0015	:	M99 command call abnormity, forbidden in current occasion.
0016	:	movement abnormity alarming
0017	:	illegal character
0018	:	annotation mark format error or no symmetrical annotation mark
0019	:	illegal G code
0020	:	the radius offset number of G code error or value error
0021	:	undefined G code radius offset error
0022	:	arc programming error
0023	:	appointing illegal plane exceeds G17,G18,G19
0024	:	calling error, probably exceeds the maximum value
0025	:	main shaft appointing hardware axis number error
0026	:	M code executing error
0027	:	main shaft appointing failure
0028	:	moving repeat requirement
0029	:	appointing arc does not exist
0030	:	lack X command error
0031	:	lack Y command error
0032	:	lack Z command error
0033	:	lack A command error
0034	:	lack B command error
0035	:	lack C command error
0036	:	lack D command error

0037	:	lack R command error
0038	:	lack F command error
0039	:	lack T command error
0040	:	lack S command error
0041	:	lack P command error
0042	:	lack M command error
0043	:	lack G command error
0044	:	lack I command error
0045	:	lack J command error
0046	:	lack K command error
0047	:	lack Q command error
0048	:	screw pitch value repeating appointing error
0049	:	system alarming and quit abnormally
0050	:	quit by man-interrupting
0051	:	no appointing G code parameter source
0052	:	non appointing G code program number sheet storage address

6.2 system environment alarming

1024	:	controller does not return-to-zero

1. Do not carry out return-to-zero after the system start.

1025	:	A axis negative direction soft limiting displacement
1026	:	A axis positive direction soft limiting displacement
1027	:	Z axis negative direction soft limiting displacement
1028	:	Z axis positive direction soft limiting displacement
1029	:	Y axis negative direction soft limiting displacement
1030	:	Y axis positive direction soft limiting displacement
1031	:	X axis negative direction soft limiting displacement
1032	:	X axis positive direction soft limiting displacement
1033	:	A axis negative direction hard limiting displacement
1034	:	A axis positive direction hard limiting displacement
1035	:	Z axis negative direction hard limiting displacement
1036	:	Z axis positive direction hard limiting displacement
1037	:	Y axis negative direction hard limiting displacement
1038	:	Y axis positive direction hard limiting displacement
1039	:	X axis negative direction hard limiting displacement
1040	:	X axis positive direction hard limiting displacement

1. the system gives the limiting displacement alarming corresponding to the hint, examine the corresponding limiting displacement induction point or parameter.
2. if it is hard limiting displacement, there is no problem with the induction point visual examination, then enter the diagnoses mode under the hand-mode, check the input port condition under the diagnoses mode, if the condition is effective, then eliminate it in turn, now pull out IO line, check whether the induction disappears, if it disappears then check the lines, if it still exists, the inside optical coupler maybe is destroyed, please contact the supplier.

1041	:	Emergency stop

1. hand-held box interface emergency stop button is effective.
2. the corresponding function interface is IN33, you can see it in the input diagnoses.

1042	:	servo X driver alarming
1043	:	servo Y driver alarming
1044	:	servo Z driver alarming
1045	:	servo A driver alarming

1. servo alarming, if the servo does not give the alarming, it is probably that the setting of parameter P2.001~004 is opposite to the servo real alarming

level, change the parameter.

2. the corresponding function interface is IN34～37, you can see it in the input diagnoses.

1046	:	axis number defining interface repeating error

1. the interface axis number of parameter P2.45～P2.49 exists repeated appointment

1047	:	Main shaft does not return to zero
1048	:	mould is not locked uo tightly
1049	:	system safety signal is not in the right position error
1051	:	insufficient system air pressure
1052	:	alarming of system clip feed signal void alarming