ADT-CNC4860
Milling Controller
User Manual

ADTECH (SHENZHEN) CNC TECHNOLOGY CO., LTD

5th Floor, 27-29th Building, Tianxia IC Industrial Park, Yiyuan road, Nanshan District,

Shenzhen Post code: 518052

Tel:86-755-26099116 Fax: 86-755-26722718

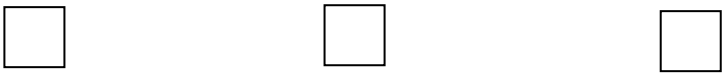
E-mail: export@adtechen.com http://www.adtechen.com

Copyright Statement

Upgrade Information

Item No.	Version No.	Revision Date	Remark
XT20080506B	1.0	2011-01-25	Edition 1

Note: Meanings of the three numbers in version number are as follows:



Library main version No. Library secondary version No. Reserved

Remark:

1. This user manual is strictly emended and checked by ADTECH (SHENZHEN) CNC TECHNOLOGY CO., LTD, however, it is not guaranteed that the user manual has no any mistake or error.
2. ADTECH (SHENZHEN) CNC TECHNOLOGY CO., LTD commits itself to improving the product functions and the service quality consistently. Therefore, the company reserves the right of changing any products as described, any software program, and the content of the user manual, without prior notice.

CONTENT

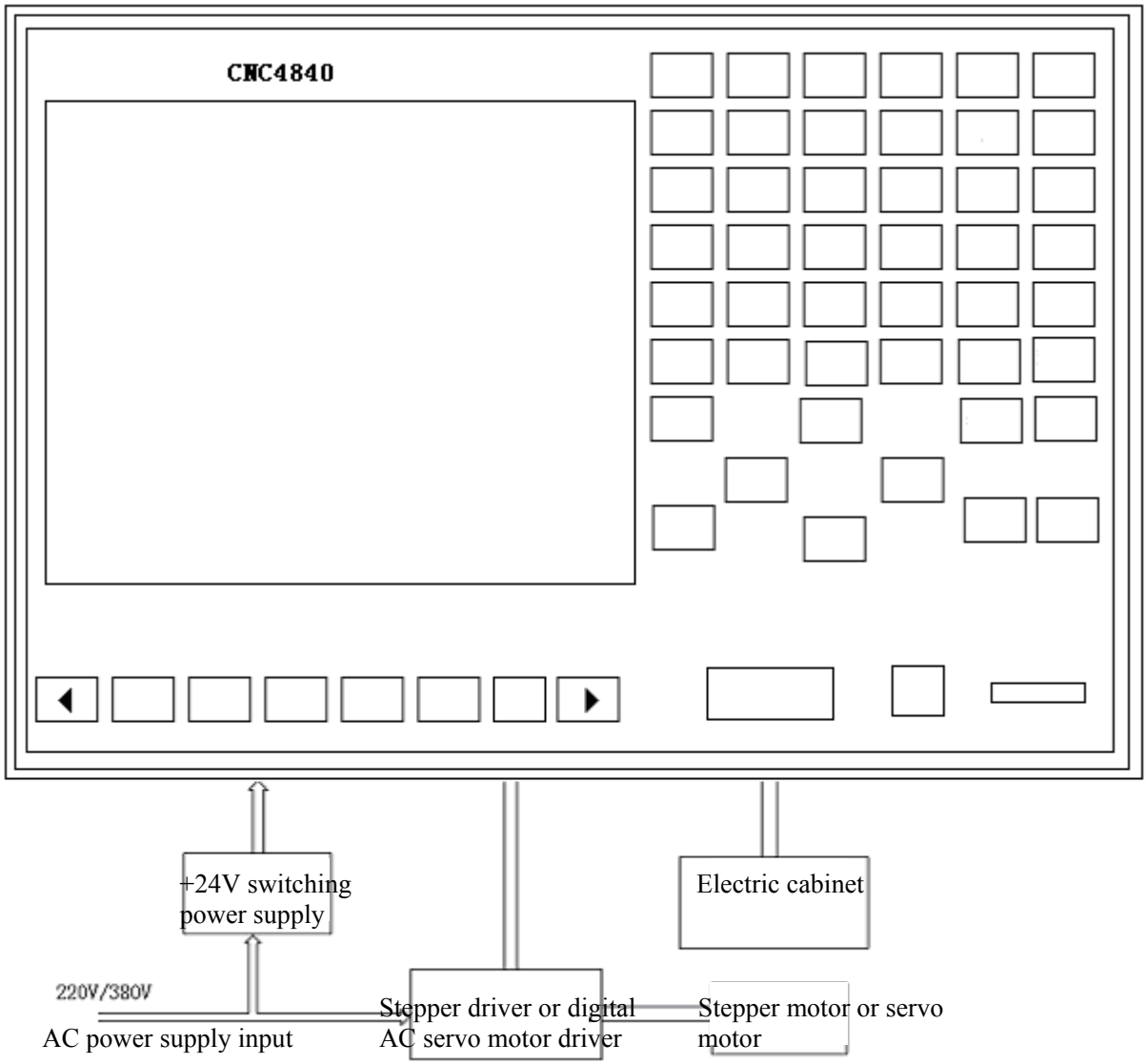
Chapter I Definition of System Interface and Wiring Description

I. System Structure

1. Parts of CNC4860 NCS

CNC4860 NCS is composed by the following main units:

1. CNC control unit (Control device CNC4860)
2. Stepper motor driver (Digital AC servo driver)
3. Stepper motor (Servo motor)
4. Electric cabinet



2. Notice for Installation

Conditions for mounting electric cabinet

- The electric cabinet should be able to prevent the entry of dust, cooling liquid and organic solution effectively.
- The electric cabinet should be designed in a way that the distance between rear cover and the casing should not be less than 20CM. Considering the temperature rise inside the electric cabinet, the difference in temperature between inside and outside should not exceed 10°C.
- Fan should be installed inside the electric cabinet so as to ensure the good air circulation inside.
- Display panel should be installed in a place away from the cooling liquid.
- Try to reduce the external electric interference to prevent it from transmitting to the system.

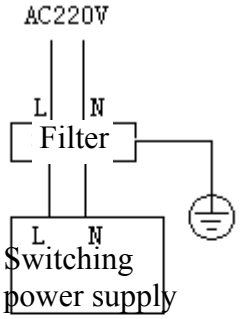
Methods to prevent the interference

When designing the system, several anti-interference measures such as shielding space electromagnetic radiation, absorbing impulse current, and filtering power supply noise are adopted, which to a certain extent prevents the external interference source from affecting the system. To ensure the stable operation of the system, the following measures should be done when installing:

1: CNC should be away from the devices that generate the interference (such as transducer, AC contactor, electrostriction generator, high pressure producer, and segment separator of dynamic line). At the same time, the switching power supply should be connected with individual filter to enhance the anti-interference capability of CNC. (As picture 1-4)

2: The system should be powered by isolating transformer, and the machine tool on which the system is installed should be grounded. CNC and driver should connect the individual earth line from the ground point.

3: Interference suppression: Connect a RC return circuit (0.01μF, 100~200Ω, as picture 1-5) in parallel at the two ends of AC coil. The RC returning circuit should be as close to the inductive load as possible when installing. Connect a freewheeling diode in parallel reversely at the two ends of DC coil (as picture 1-6). Connect surge absorbers in parallel at the winding ends of AC motor (as picture 1-7).



Picture 1-4

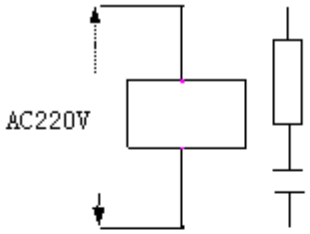
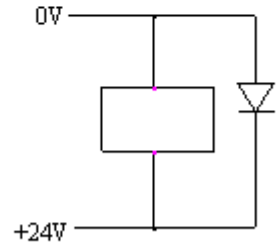
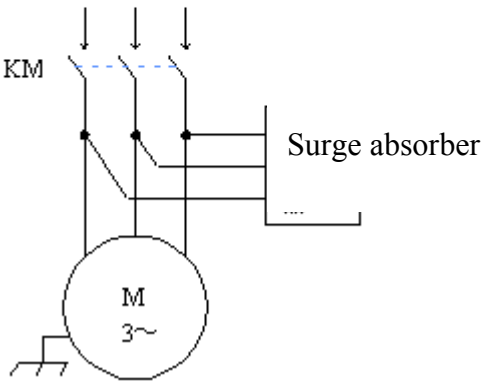


图 1-5



Picture 1-6



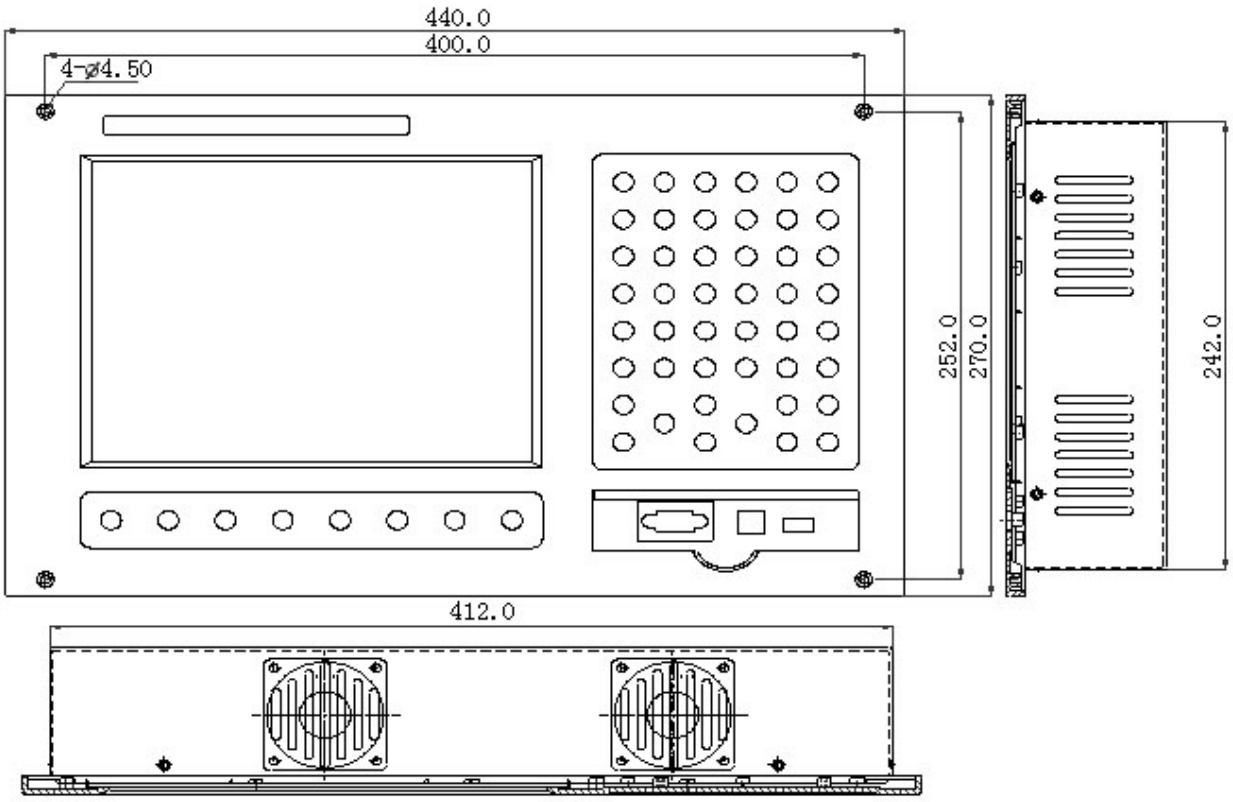
Picture 1-7

4: To reduce the interface between the CNC signal cables and the electric cables, the wiring should follow the rules below:

Group	Type of Cable	Wiring Requirements
A	AC supply line	Bind the cable of group A and group B and C separately, reserve the distance of at least 10cm, or electromagnetic shielding the group A cable
	AC coil	
	AC contactor	
B	DC coil (24VDC)	Bind the cable of group B and group A separately, or shield the cable of group B. Cables of group B and group C should be placed as far as better.
	DC relay (24VDC)	
	Cable connecting system and electric cabinet	
	Cable connecting system and controller	
C	Cable connecting system and servo driver	Bind the cable of group C and group A separately, or shield the cable of group C. The distance between group C and group B should be at least 10cm, and the cable uses the twisted pair.
	Position feedback cable	
	Position coder cable	
	Handwheel cable	
	Other cables for the purpose of shielding	

3. Installing Dimension

Installing dimension of CNC4860 controller

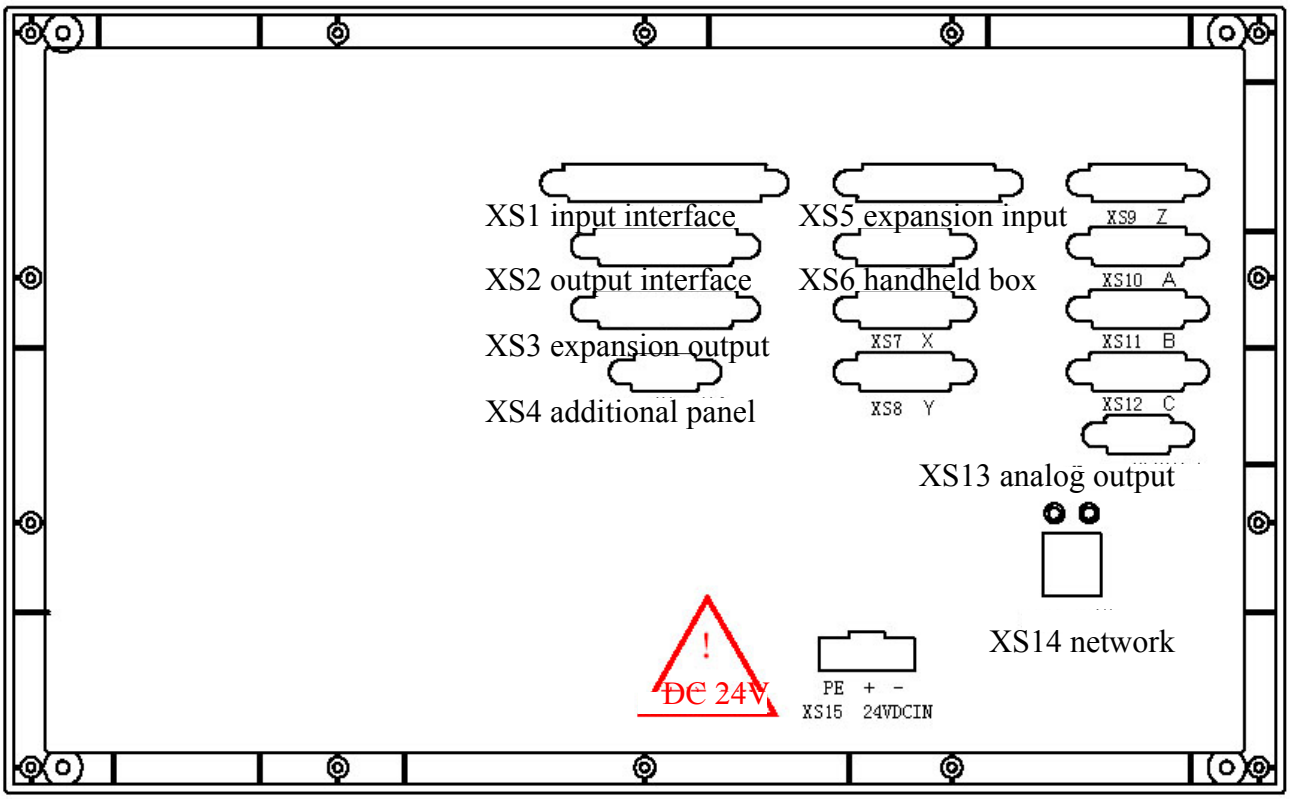


II. External Connection

1. External Interface

CNC4860 control unit is connected to the external devices via the rear and front interfaces.

1. The outer casing of CNC4860 is defined as follows:

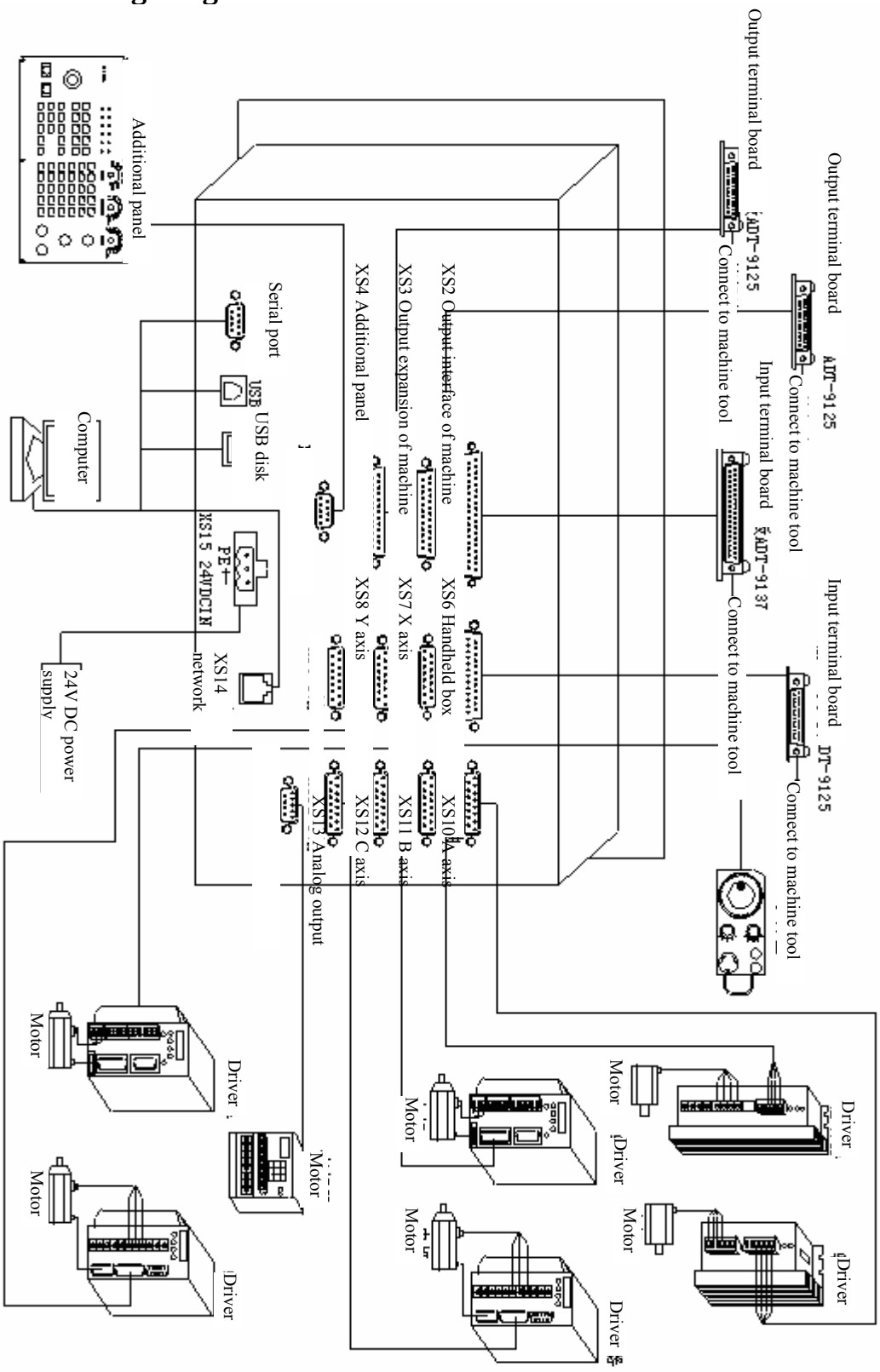


X, Y, Z, A, B, and C refer to the connecting signal of stepper motor driver or digital AC servo driver of each axis. CNC4860 controller uses X, Y, Z, A, B, and C axes at the moment.

Input interfaces and expansion input interfaces of the machine are limit and digital input signals of each axis. Output interfaces and expansion output interfaces are the digital output signal.

CNC4860 controller uses the 24V DC power supply, and the internal power consumption is about 5W.

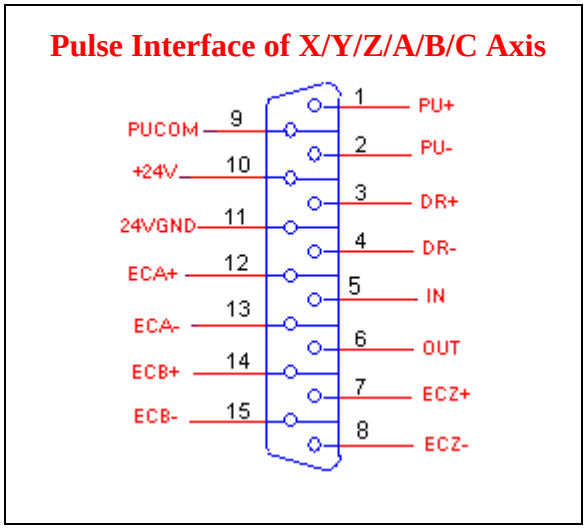
General Wiring Diagram



2. Control Interface of Motor Driver

There are 8 interfaces for the driver (X, Y, Z, A, B, C axes), and the interface definition is the same. Refer to the following picture:

● Pulse interface of axis 1-6

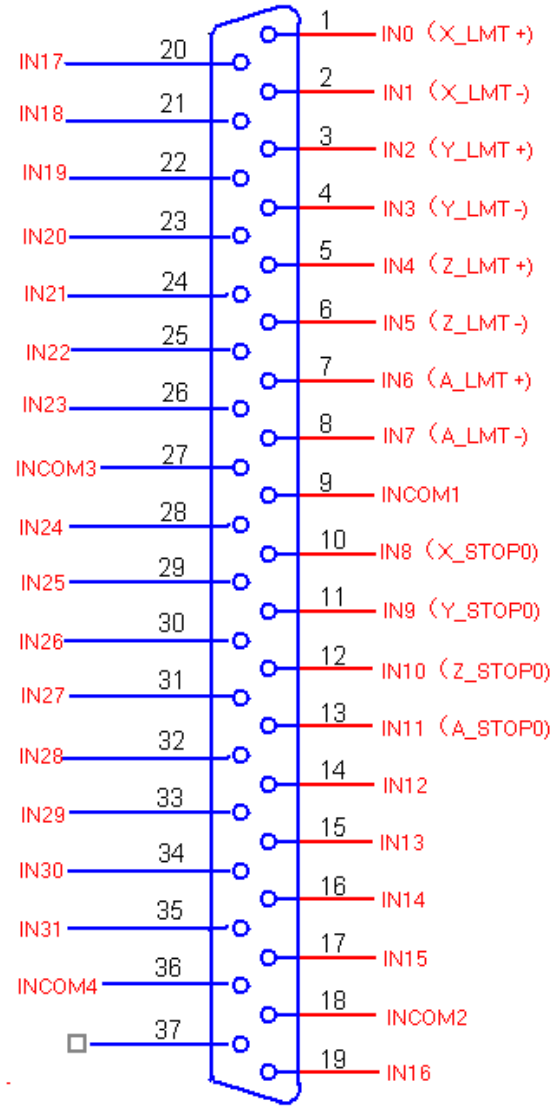


Line S/N	Name	Function
1	nPU+	Pulse signal +
2	nPU-	Pulse signal -
3	nDR+	Direction signal +
4	nDR-	Direction signal -
5	IN	General input, can be used as alarm input (X-66 Y-67 Z-68 A-69 B-70 C-71)
6	OUT	General output (X-48 Y-49 Z-50 A-51 B-52 C-53)
7	nECZ+	Coder Z-phase input + (X-72 Y-73 Z-74 A-75 B-76 C-77)
8	nECZ-	Coder Z-phase input -
9	PUCOM	Used for driver with single-end input
10	+24V	Provide internal 24V power supply, directly connected with 24V power supply of controller
11	24VGND	
12	nECA+	Coder A-phase input + (X-78 Y-80 Z-82 A-84 B-86 C-88)
13	nECA-	Coder A-phase input -
14	nECB+	Coder B-phase input + (X-79 Y-81 Z-83 A-85 B-87 C-89)

15	nECB-	Coder B-phase input -

3. Input Interface of Machine

1) The digital input interfaces include the zero points of XYZABC axes, hardware limit signal of XYZA axes, etc. The definition is as follows:



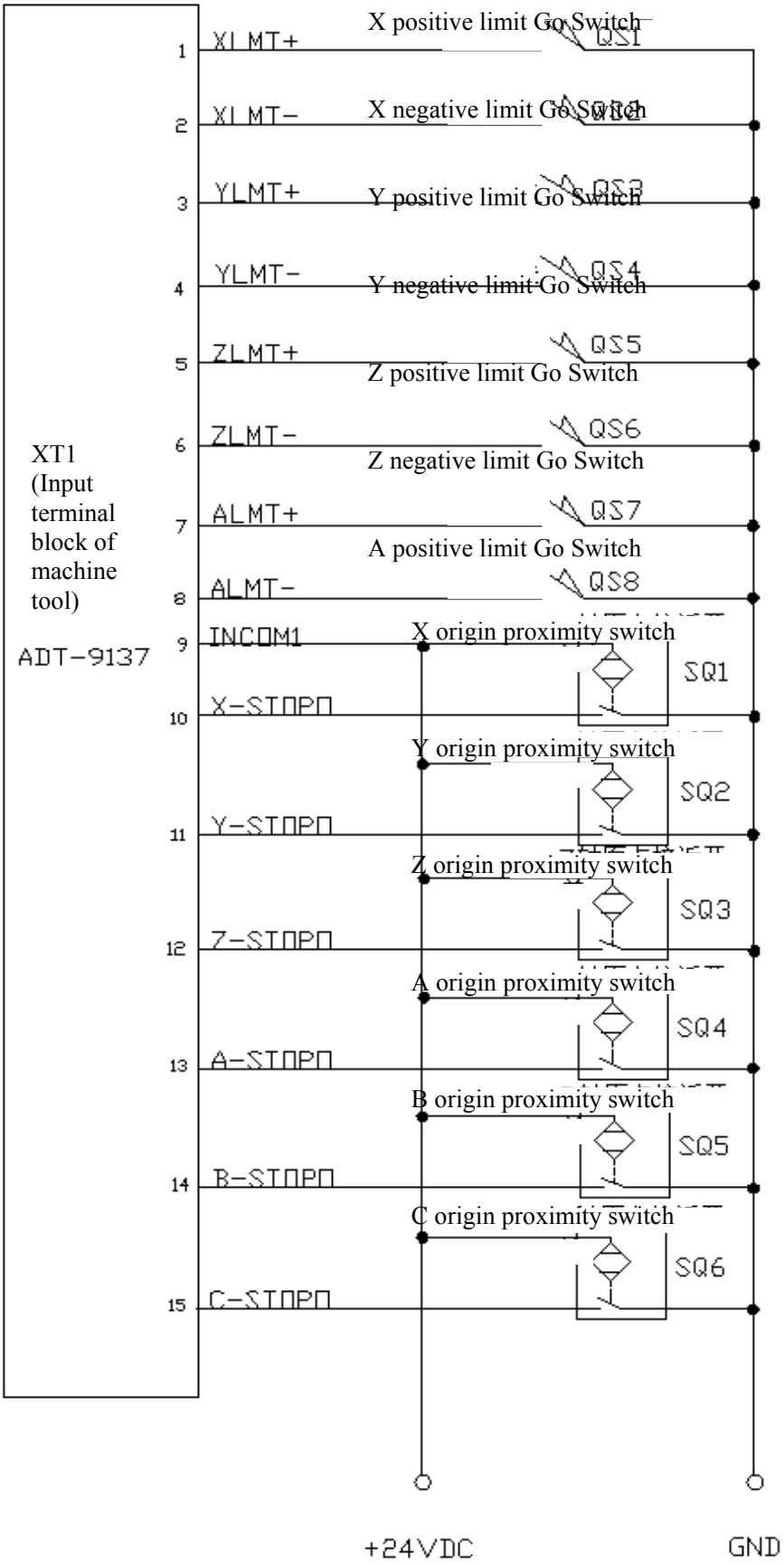
Input Interface of Machine

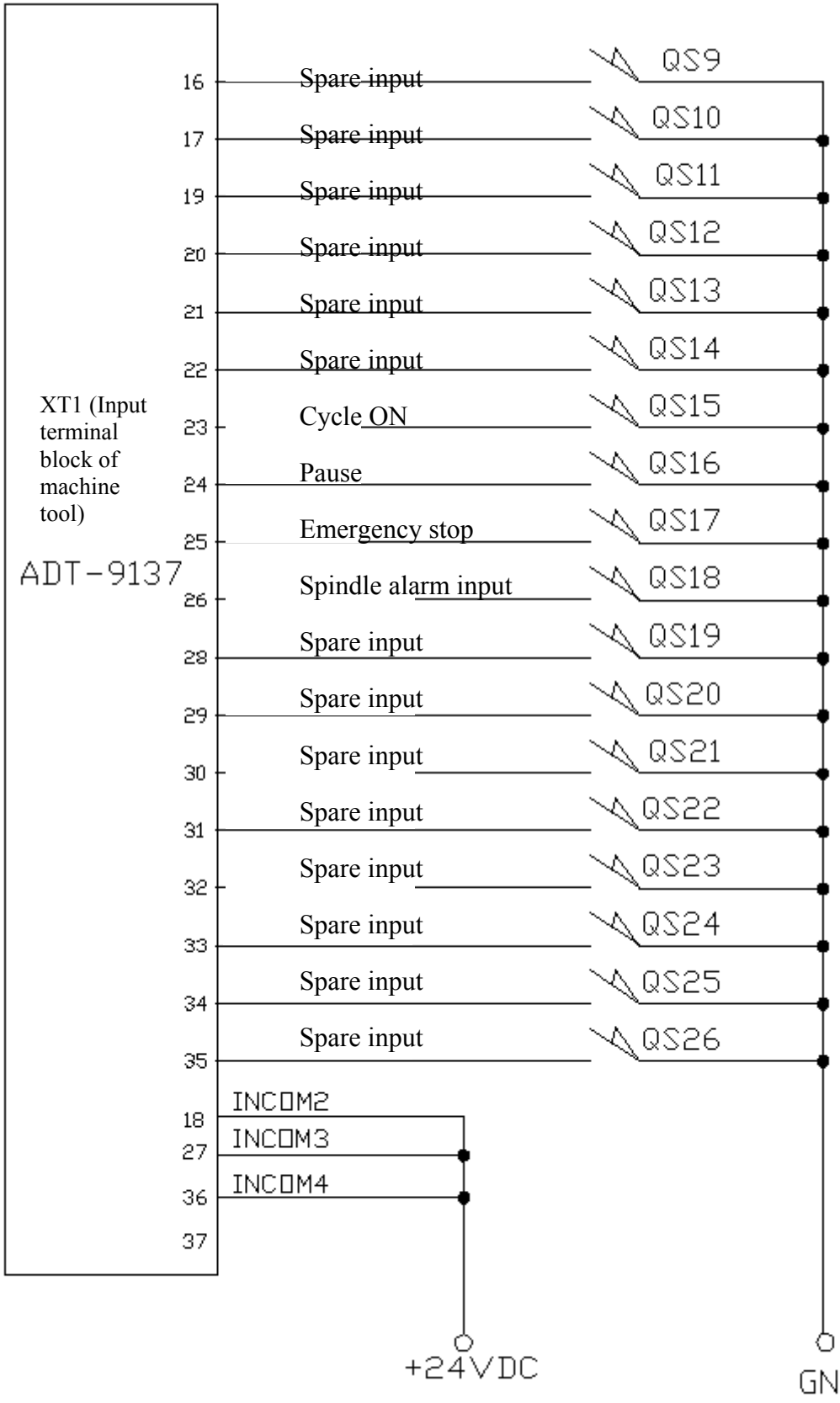
Line S/N	Name	Function
1	IN0 (X_LMT+)	X positive limit
2	IN1	X negative limit

	(X_LMT-)	
3	IN2 (Y_LMT+)	Y positive limit
4	IN3 (Y_LMT-)	Y negative limit
5	IN4 (Z_LMT+)	Z positive limit
6	IN5 (Z_LMT-)	Z negative limit
7	IN6 (A_LMT+)	A positive limit
8	IN7 (A_LMT-)	A negative limit
9	INCOM1	Common input terminal (24v+, 12v+)
10	IN8 (X_STOP0)	X axis zero
11	IN9 (Y_STOP0)	Y axis zero
12	IN10 (Z_STOP0)	Z axis zero
13	IN11 (A_STOP0)	A axis zero
14	IN12 (B_STOP0)	B axis zero
15	IN13 (C_STOP0)	C axis zero
16	IN14	Air pressure alarm input
17	IN15	Spare input (used to detect the material-champing alarm input signal during the operation)
18	INCOM2	Common input terminal (24v+, 12v+)
19	IN16	B positive limit
20	IN17	B negative limit
21	IN18	C positive limit
22	IN19	C negative limit
23	IN20	Cycle ON
24	IN21	Pause
25	IN22	Emergency stop
26	IN23	Alarm input of main axis
27	INCOM3	Common input terminal (24v+, 12v+)
28	IN24	Input for triggering feeler device
29	IN25	Input for protecting feeler device
30	IN26	Spare input
31	IN27	Spare input
32	IN28	Spare input
33	IN29	Spare input
34	IN30	Spare input

35	IN31	Spare input
36	INCOM4	Common input terminal (24v+, 12v+)
37		

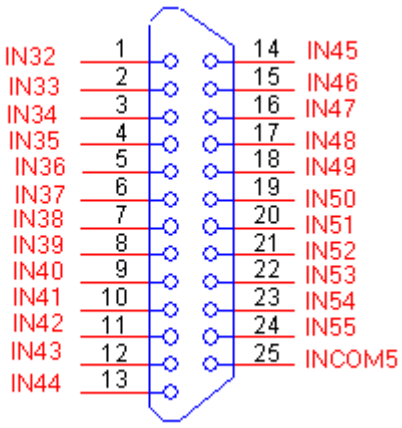
2) Diagram of wiring between input interfaces and photoelectric switch/proximity switch is as follows:





4. Input Expansion Interface

1) Digital input interfaces include BC hardware limit, other spare input signals, etc. the definitions are as follows:

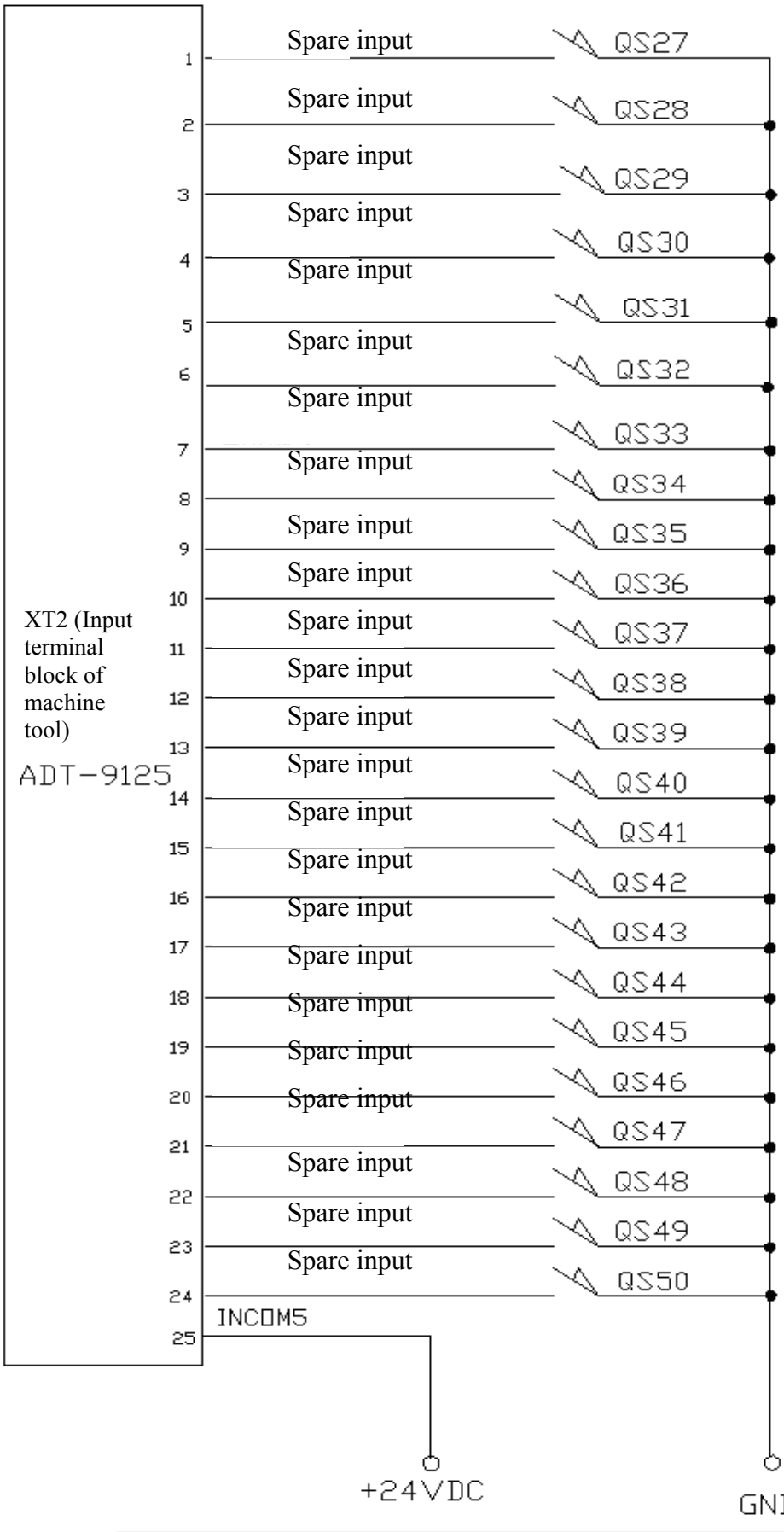


Input Expansion Interface

Line S/N	Name	Function
1	IN32	Spare input
2	IN33	Spare input
3	IN34	Spare input
4	IN35	Spare input
5	IN36	Spare input
6	IN37	Spare input
7	IN38	Spare input
8	IN39	Spare input
9	IN40	Spare input
10	IN41	Spare input
11	IN42	Spare input
12	IN43	Spare input
13	IN44	Spare input
14	IN45	Spare input
15	IN46	Spare input
16	IN47	Spare input
17	IN48	Spare input
18	IN49	Spare input
19	IN50	Spare input
20	IN51	Spare input
21	IN52	Spare input

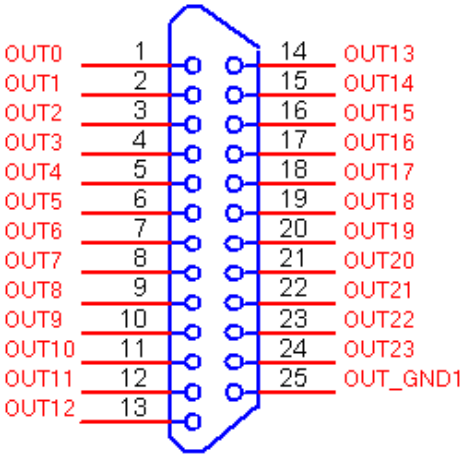
22	IN53	Spare input
23	IN54	Spare input
24	IN55	Spare input
25	INCOM5	Common input terminal (24v+, 12v+)

2) The wiring of expansion input interface and proximity switch is shown as follows:



5. Output Interface

1) The output interface and the wiring are defined as follows:

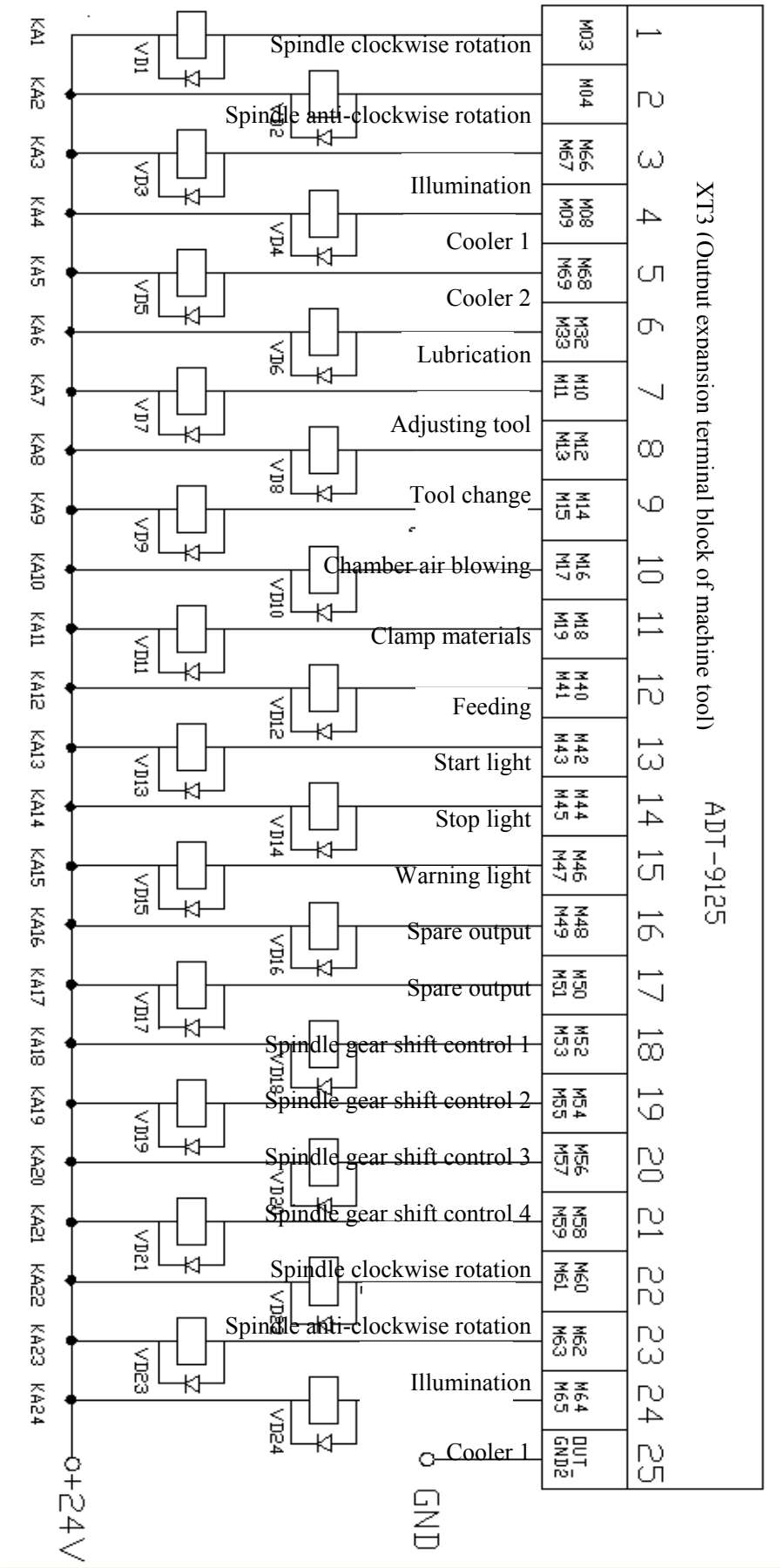


Output Interfaces

Line S/N	Name	Function
1	OUT0	Spindle clockwise rotation (M03)
2	OUT1	Spindle anti-clockwise rotation (M04)
3	OUT2	Illumination (M66, M67)
4	OUT3	Cooler 1 (M08, M09)
5	OUT4	Cooler 2 (M68, M69)
6	OUT5	Lubrication (M32, M33)
7	OUT6	Adjusting tool (M10, M11)
8	OUT7	Tool change (M12, M13)
9	OUT8	Chamber air blowing (M14, M15)
10	OUT9	Clamp materials (M16, M17)
11	OUT10	Feeding (M18, M19)
12	OUT11	Start light (M40, M41)
13	OUT12	Stop light (M42, M43)
14	OUT13	Warning light (M44, M45)
15	OUT14	Dumping (M46, M47)
16	OUT15	Discharging scraps (M48, M49)
17	OUT16	Knife warehouse + (M50, M51)

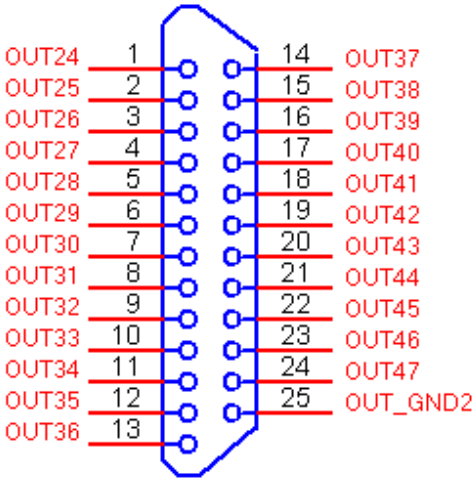
18	OUT17	Knife warehouse - (M52, M53)
19	OUT18	Spare output (M54, M55)
20	OUT19	Spare output (M56, M57)
21	OUT20	Spindle gear shift control 1 (M58, M59)
22	OUT21	Spindle gear shift control 2 (M60, M61)
23	OUT22	Spindle gear shift control 3 (M62, M63)
24	OUT23	Spindle gear shift control 4 (M64, M65)
25	OUT_GND1	Common power supply of 12v-, 24v- output

2) Wiring diagram of output interface is as follows:



6. Output Expansion Interface

1) The expansion output interface and the wiring are defined as follows:

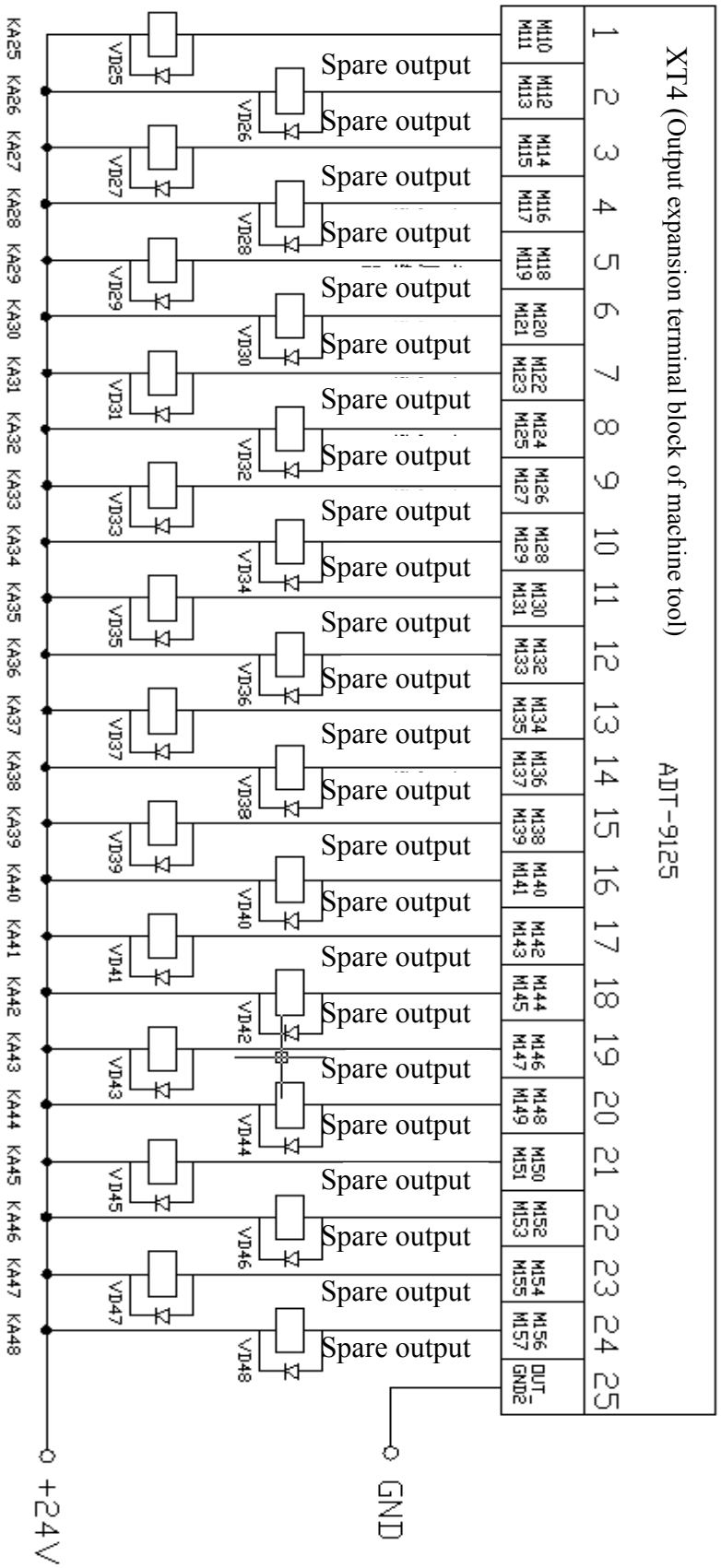


Output Expansion Interface

Line S/N	Name	Function
1	OUT24	Spare output (M110, M111)F10
2	OUT25	Spare output (M112, M113)F11
3	OUT26	Spare output (M114, M115)F12
4	OUT27	Spare output (M116, M117)F13
5	OUT28	Spare output (M118, M119)F14
6	OUT29	Spare output (M120, M121)F15
7	OUT30	Spare output (M122, M123)F16
8	OUT31	Spare output (M124, M125)F17
9	OUT32	Spare output (M126, M127)F18
10	OUT33	Spare output (M128, M129)F19
11	OUT34	Spare output (M130, M131)F20
12	OUT35	Spare output (M132, M133)
13	OUT36	Spare output (M134, M135)
14	OUT37	Spare output (M136, M137)
15	OUT38	Spare output (M138, M139)
16	OUT39	Spare output (M140, M141)
17	OUT40	Spare output (M142, M143)
18	OUT41	Spare output (M144, M145)

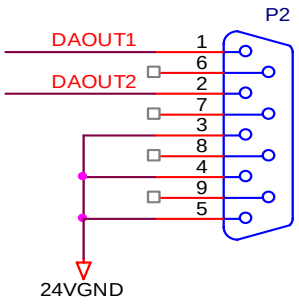
19	OUT42	Spare output (M146, M147)
20	OUT43	Spare output (M148, M149)
21	OUT44	Spare output (M150, M151)
22	OUT45	Spare output (M152, M153)
23	OUT46	Spare output (M154, M155)
24	OUT47	Spare output (M156, M157)
25	OUT_GND2	Common power supply of 12v-, 24v-

2) Wiring diagram of expansion output interface is as follows:



7. Analog Output

1) The analog output interface is defined as follows:

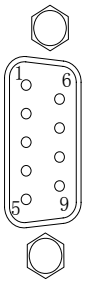


Analog Output

Line S/N	Name	Function
1	DAOUT1	Analog voltage output (0V—12V+)
2	DAOUT2	Analog voltage output (0V—12V+)
3	24V-	Provide internal24V grounding
4		
5		

2) Wiring diagram of analog output is as follows:

)efinition of analog
utput interface



XS13 interface

Connection of CNC4840 transducer

CNC4840 (analog output)
XS13 interface

1	DAOUT1
2	DAOUT2
3	24V-
4	24V-
5	24V-

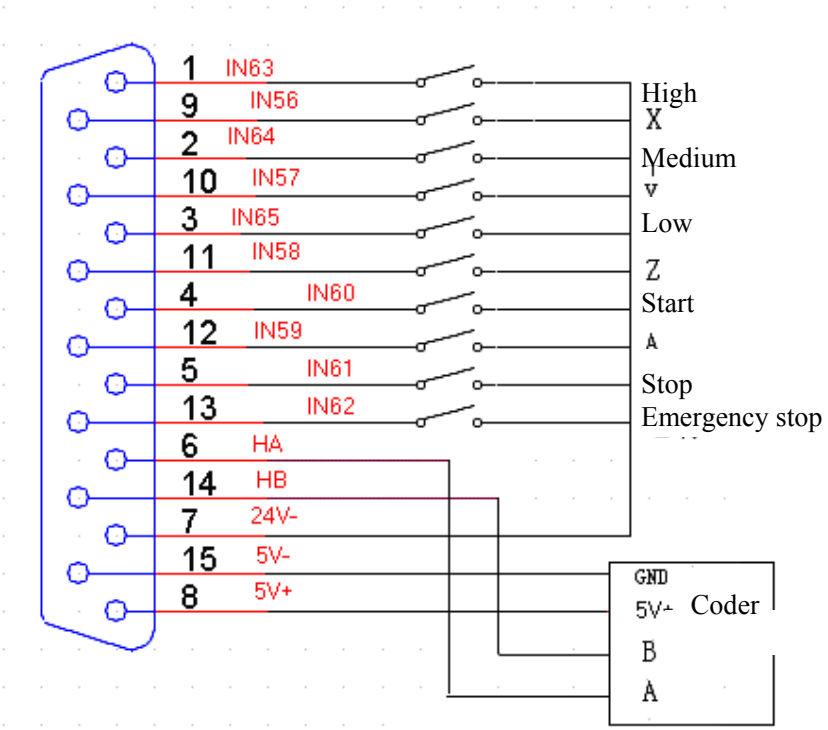
CWF-S1 transducer signal interface

	VI1
	CM

Note: Choose either DAOUT1 or DAOUT2 to connect the transducer

8. Handheld box

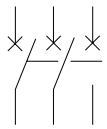
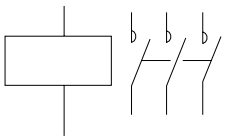
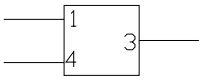
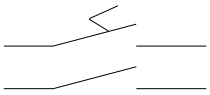
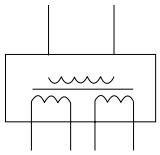
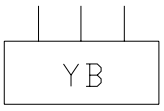
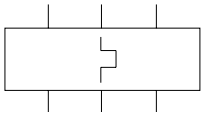
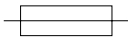
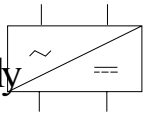
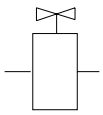
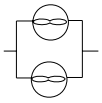
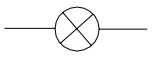
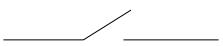
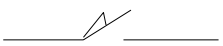
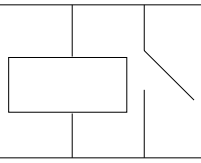
Handheld box: Connected with our standard handheld box ADT-CNC4A, multiplexing with manual pulse generator count and Y-axis coder



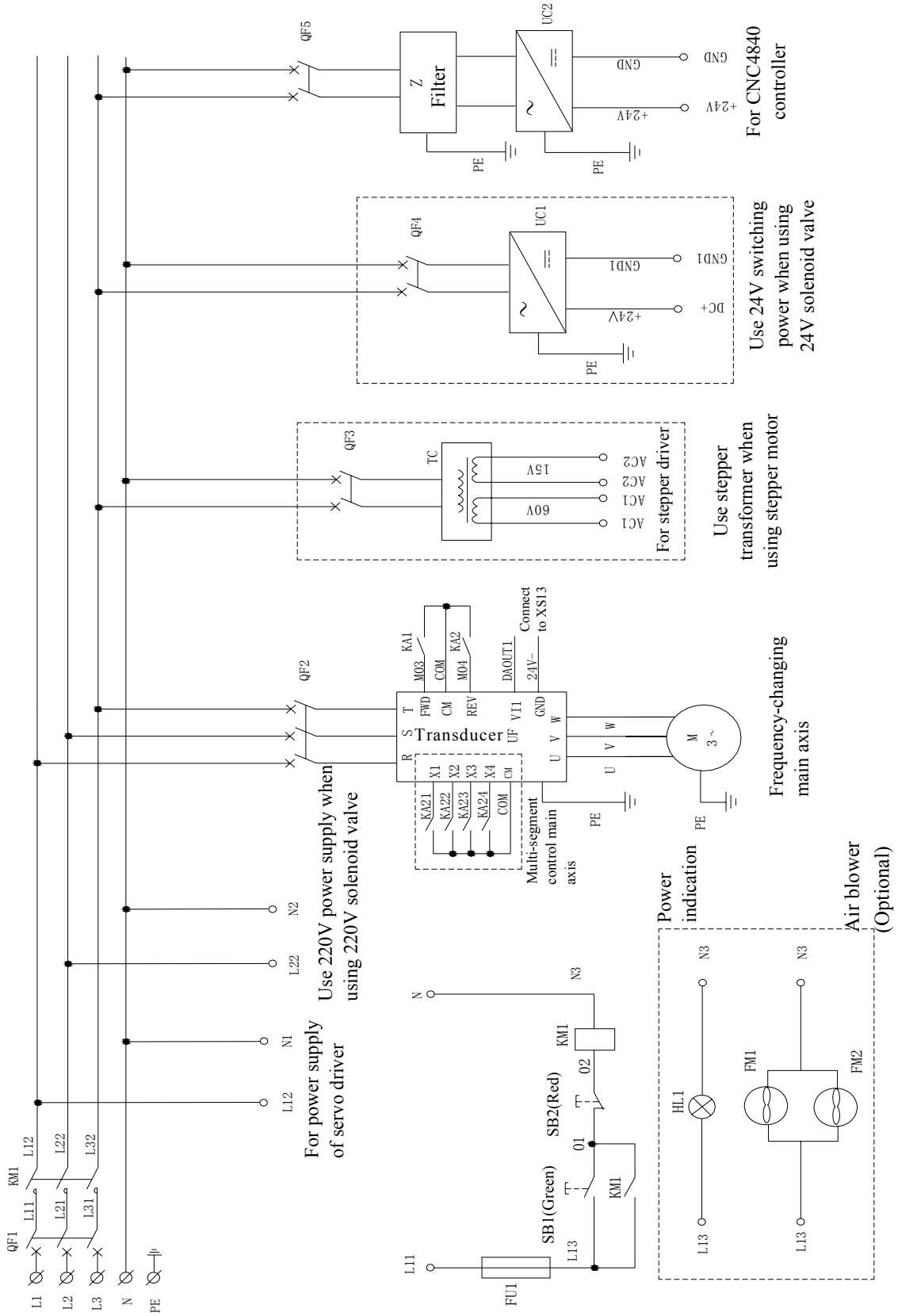
Definition of corresponding casing: Handheld box

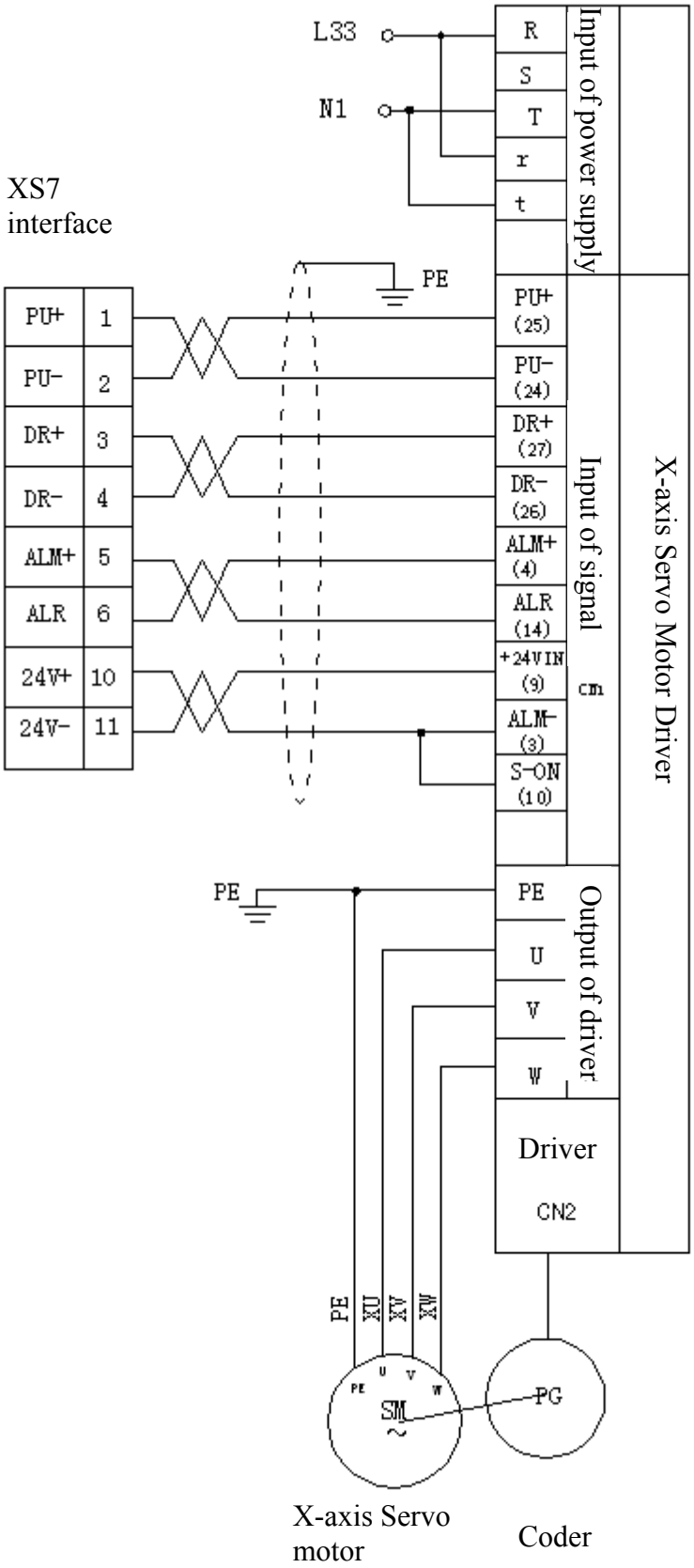
Line S/N	Name	Function
1	IN63	0.1 shift switch-High
2	IN64	0.01 shift switch-Medium
3	IN65	0.001 shift switch-Low
4	IN60	Start-up
5	IN61	Stop
6	HA	Handle coder phase-A input signal
7	24V-	Internal -24V power supply
8	5V+	Internal +5V power supply
9	IN56	Select X axis
10	IN57	Select Y axis
11	IN58	Select Z axis
12	IN59	Select A axis
13	IN62	emergency stop
14	HB	Handle coder phase-B input signal
15	5V-	Internal -5V power supply

9. Electrical Connection Diagram

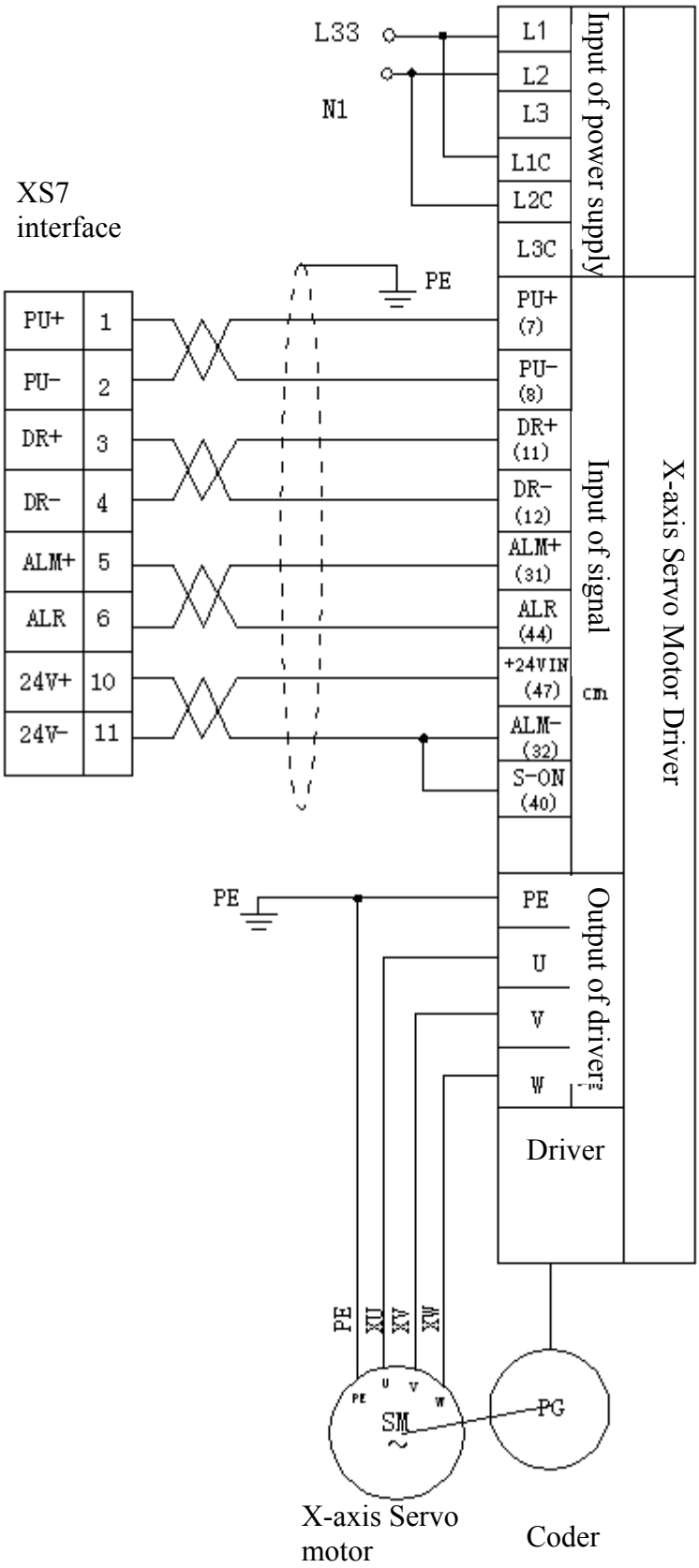
Sign	Name	Chart	Sign	Name	Chart
QF	Breaker		SM	Servo motor	
KM	Contactor		M	Stepper motor	
UF	Transducer		SQ	Proximity switch	
M	Motor		SA	Foot switch	
TC	Transformer		YB	Thermal relay	
Z	filter		FR	thermal relay	
FU	Fuse		UC	Switching power supply	
SB	Button		YV	Solenoid valve	
FM	Air blower		C	compacitor	
HL	indicator		R	resistor	
QS	Touch switch		QS	Go switch	
PG	Coder		KA	Relay	

10. Legend of connection between CNC4860 and servo/stepper driver

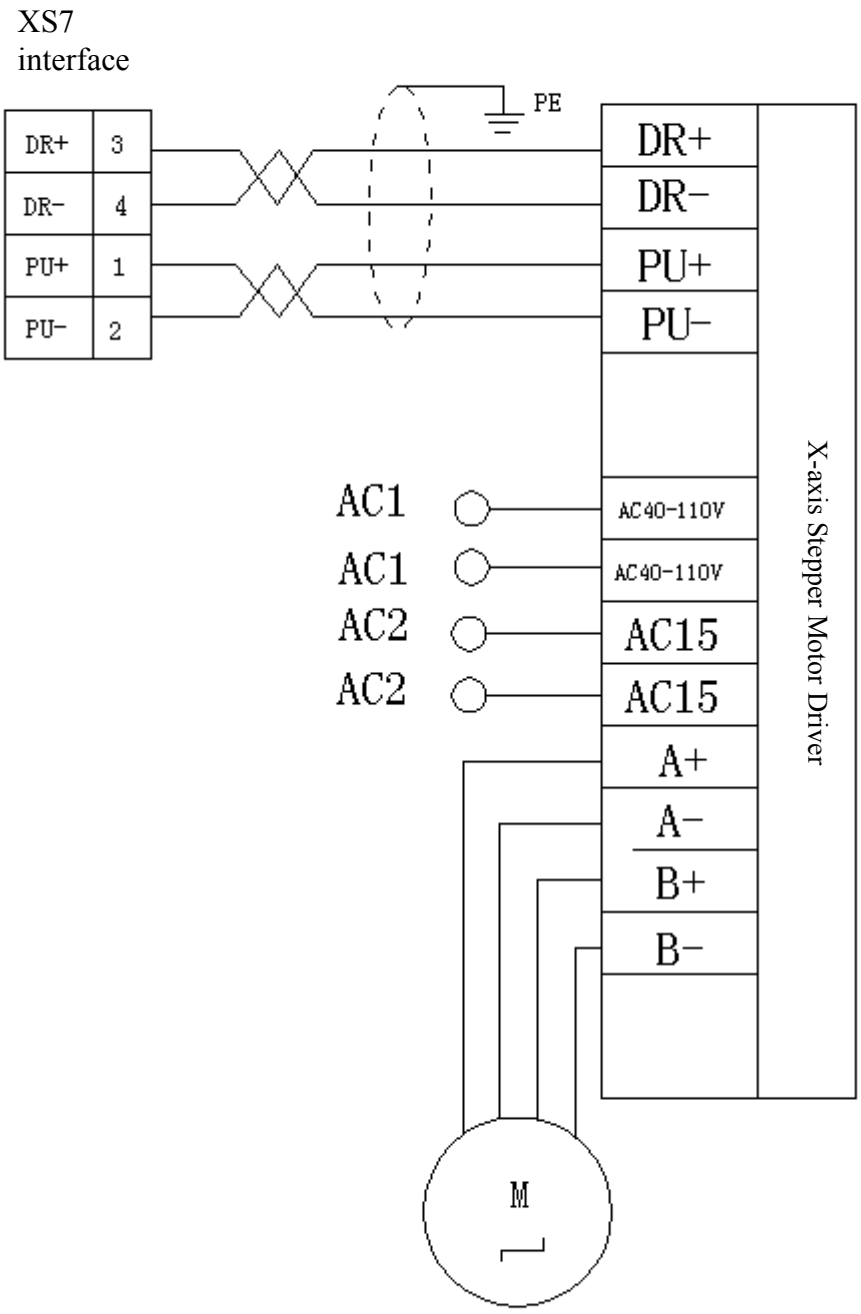




Example 1: Connection with JaBao QS5 driver

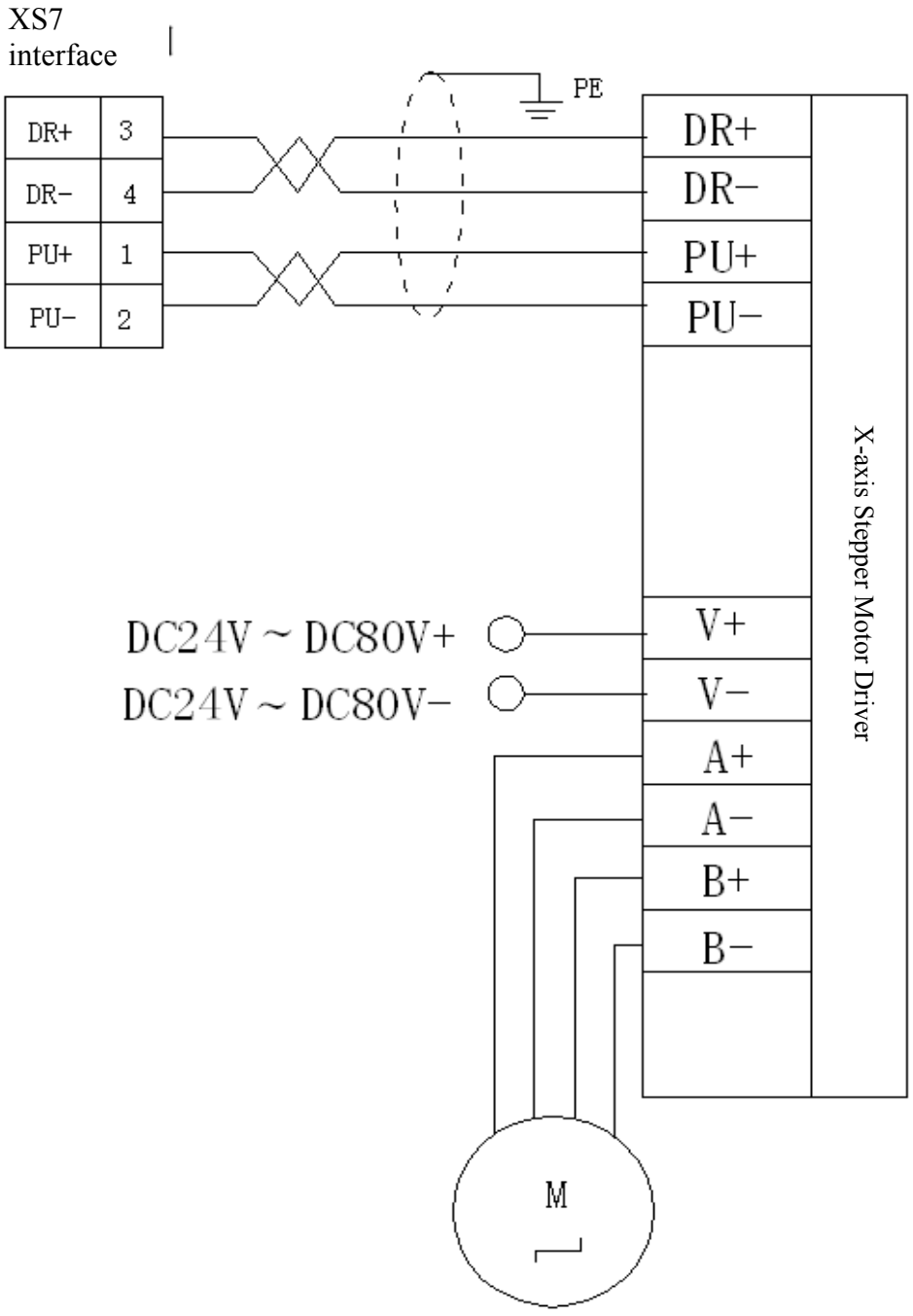


Example 2: Connection with SGDM driver



X-axis Stepper Motor

Example 3: Connection with Q2BYG1106M stepper driver



X-axis Stepper Motor

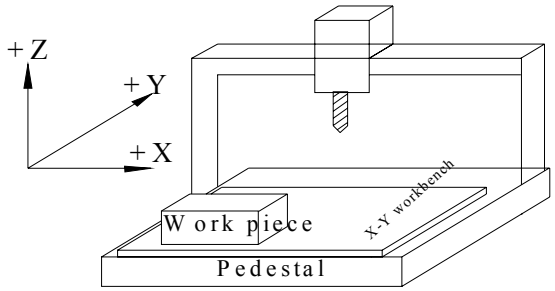
Example 4: Connection with Q2BYG808M stepper driver

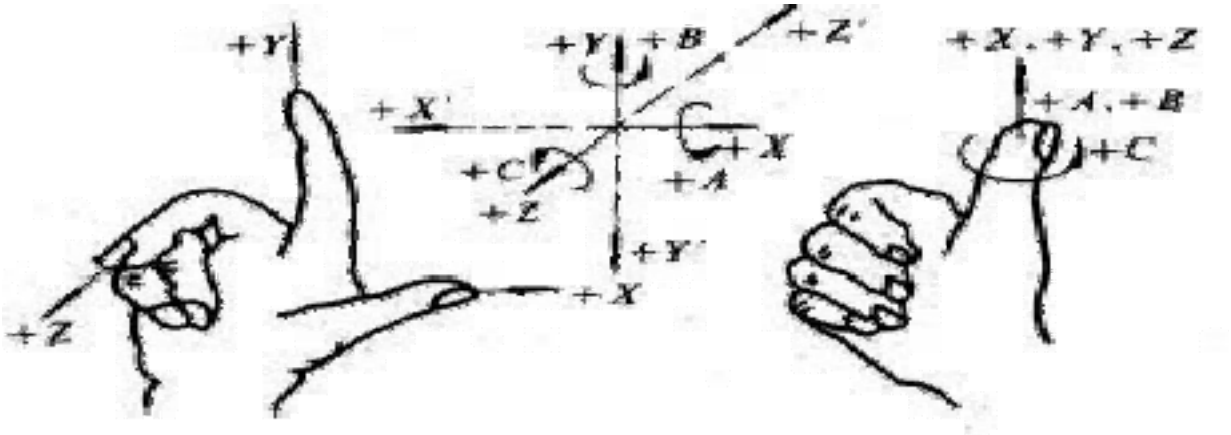
Chapter II Programming

G code programming

1.1 Basic knowledge of programming

1.1.1 Moving direction and definition of control axis





This system can control the quick move of 4 axes, and the feeding can control interpolation of 3 axes.

For the definition of axis direction, Cartesian coordinates is adopted, as follows (facing the machine tool):

Z: If the tool moves up and down corresponding to the work piece, it is the Z-axis motion. If the tool moves upward, it is Z-axis positive motion; and if the tool moves downwards, it is the Z-axis negative motion.

X: If the tool moves left and right corresponding to work piece, it is the X-axis motion. If the tool moves left, it is the X-axis negative motion; and if the tool moves right, it is the X-axis positive motion.

Y: If the tool moves forward and backward corresponding to work piece, it is the Y-axis motion. If the tool moves forward, it is the Y-axis positive motion; and if the tool moves backward, it is the Y-axis negative motion.

Spindle: When downward looking the work piece, clockwise rotation is the positive rotation while the anti-clockwise is the reverse rotation.

A, B, C: The positive direction of rotating coordinate axis is at the positive direction of X, Y, or Z coordinate axis respectively. Use the forwarding direction of right-hand screw to determine the positive direction.

Note: Descriptions of X, Y, Z, A, B, or C-axis motions in this user manual always refer to those motions of tool corresponding to the work piece, meaning that it is supposed the coordinate system of work piece is set.

1.1.2 Coordinate system of machine tool and work piece (G53, G54~G599)

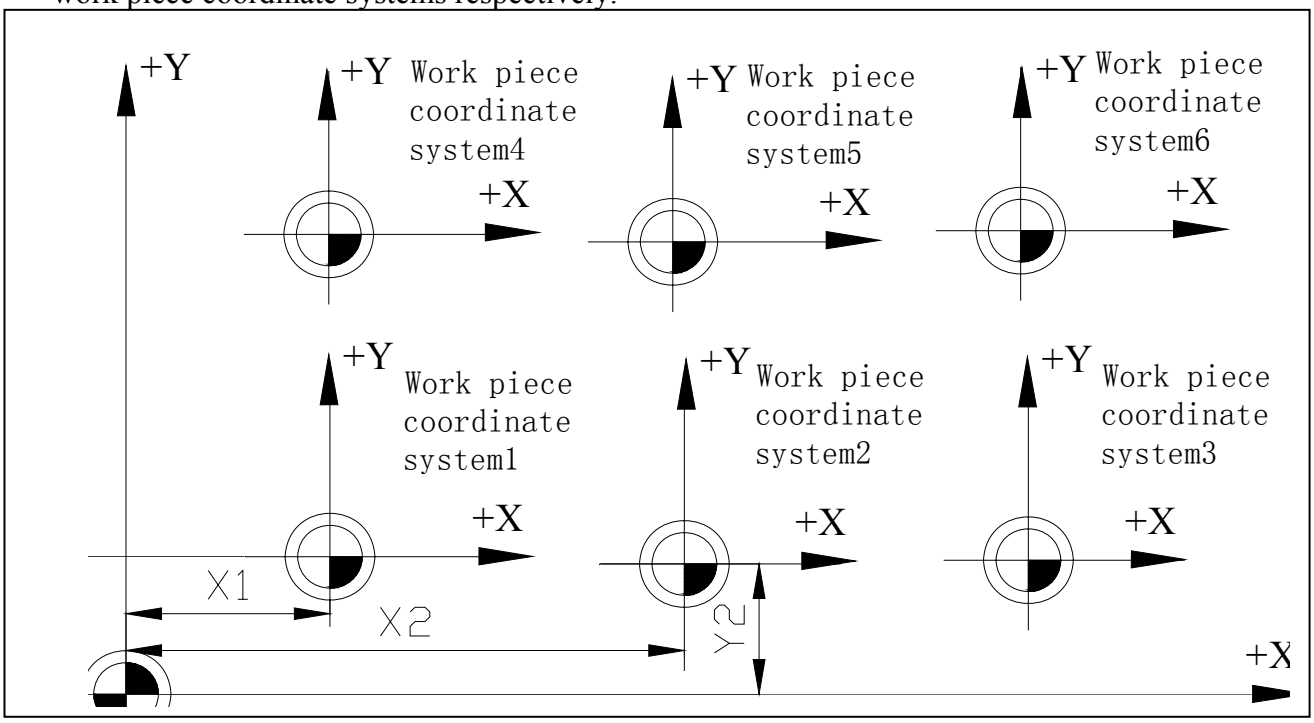
1) Coordinate system of machine tool

The coordinate system of machine tool is fixed, which is set every time when returning back to the reference point after electrifying. To choose the coordinate system of machine tool, use the G53 instruction.

2) Coordinate system of work piece

Coordinate system of work piece refers to that used during the programming processing, and that in which a certain reference center of work piece is set as the origin of coordinates. Usually, when programmers start to edit the programme, they do not know exactly where the work piece is located on the machine tool, so the program for work piece is edited taking a certain point on the work piece as the reference point. Therefore, the coordinate system formed basing on this reference point is called as coordinate system of work piece. Once the work piece is fixed on the worktable, first you should move the tool to the appointed reference point of work piece and set the machine coordinate value of this point as the origin of work piece coordinate system. In this way, when the system is performing the processing program, the tool will then process according to the program instructions while taking the work piece coordinate system as the reference. Therefore, the origin off-set function is very important for CNC machine tool.

In this system, all together 6 work piece coordinate systems can be preset (Nine expansion coordinate systems G591-G599 are added in new edition). Set the offset of origin of each work piece coordinate system corresponding to the origin of machine tool coordinate system, and then use G5X (5X refers to the No. of actual work piece coordinate system, the following is just the same) instruction to choose. G5X are mode instructions, and are corresponding to 1#~6# preset work piece coordinate systems respectively.



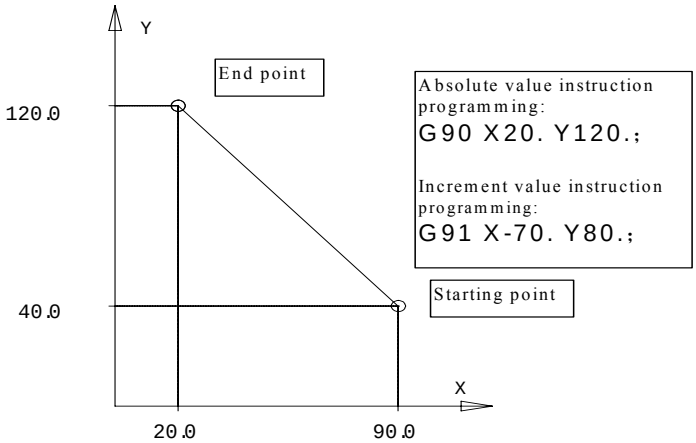
3) Programming of absolute coordinate and relative coordinate (G90, G91)

Tool motion instructions include: Absolute value instruction and increment value instruction. In absolute value instruction, the specified value is the coordinate value of end point in the current coordinate system. In increment value instruction, the specified value is the distance of all coordinate axes moving corresponding to the starting point.

G90……Absolute value instruction

G91……Increment value instruction

Instance:



From the above instance, we can better understand the programming in mode of absolute value and increment value.

1.1.3 Mode function and modeless function

Mode function refers to that once a code is specified in the current program segment, it will be valid until another code of the same group appears in the segment, and you need not specify the code if it this instruction is used again in the next program segment.

Modeless function refers to that a certain code is valid only in the program segment which it belongs to. If the instruction is used in the next program segment, you should specify the code again.

For example:

N0 G54 G0 X0 Y0; (choose the work piece coordinate system, locate to X0 Y0 quickly)

N1 G01 X150. Y25. F100; (linear interpolation to X150, Y25)

N2 X50. Y75. F120; (linear interpolation to X50, Y75; G01 is the mode instruction, and can be omitted.)

N3 X0; (linear interpolation to X0, Y75; F120 is the mode instruction, and can be omitted.)

1.1.4 Feed function

The feed of CNC machine tool can generally be divided into two classes, quick locating feed and cutting feed.

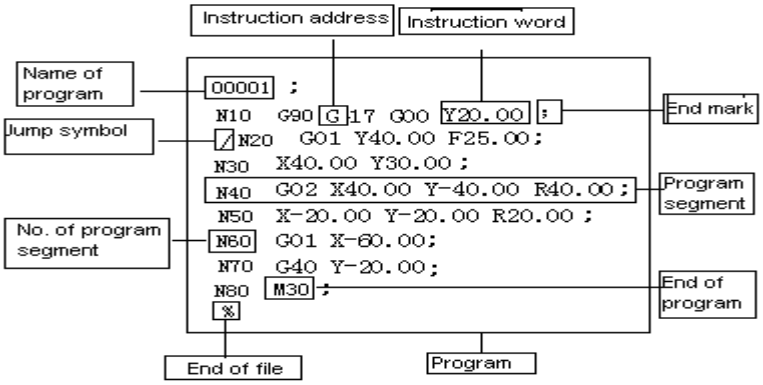
Quick locating feed occurs in the motion between quick feed and location in mode of instruction G00, manual quick move and fixed cycle, and the speed is determined by machine tool parameters. In quick locating feed, the motion of each involved axis is not related, which is moved at the set quick speed. Generally, the track of tool is a polyline or line.

Cutting feed occurs in the processing feed in mode of G01, G02/03, and fixed cycle, and the speed is specified by address F, unit in mm/minute. In processing program, F is the value of a mode, which means that the former F value is still valid before specifying a new F value. As the CNC system is electrified, the value of F is then specified by the system parameter. The involved axes are of the interpolation relationship, and the combination of their motion is the cutting feed.

The maximum value of F is controlled by system parameters. If the F of programming is larger than this value, the actual speed of cutting feed will be retained as this value.

The speed of cutting feed can also be controlled by the feed rate switch on control panel. The actual speed is the result of given value of F multiplying the feed rate, and the range of rate is 10%-150%.

1.1.5 Program structure



In processing program, an English letter is defined as an instruction address. In this manual, we call it "Address" for short. An instruction word is formed if the address is followed with a number. A program segment is composed by one or more instruction words, and ended with an end mark ";", and several program segments will make a process program. Instruction word is the basic unit of program segment. Each address has different meanings, which as a result that the value that follows would have different formats and ranges. Refer to the following table:

Function	Address	Range	Meaning
Name of program	O	1~9999	Program No.
No. of program segment	N	1~9999	Order number
Preparation function	G	00~99	Specify CNC function
Dimension	X, Y, Z	±99999.999mm	Coordinate value
	R	±99999.999mm	Radius of circular arc or fillet
	I, J, K	±9999.9999mm	Coordinate value of center
Feeding speed	F	1~100,000mm/minute	Feeding speed
Spindle rotation speed	S	1~4000 cycles per minute	Value of spindle rotation speed
Tool selection	T	0~99	Tool number
Auxiliary functions	M	0~99	Auxiliary function M-code No.
Tool off-set No.	H, D	1~200	Specify off-set No. of tool
Pause time	P, X	0~65s	Pause time (mm)
Specify subprogram No.	P	1~9999	For calling subprogram
Cycle times	P, L	1~999	For calling subprogram
Parameter	P, Q, R	P: 0~99999.999 Q: ±99999.999mm R: ±99999.999	Fixed loop parameter

In addition, a program segment can have an optional program segment number (N××××) at the beginning to mark it. It has to be noted that the sequence of program segment executed has something to do with their positions in program memory, but has nothing to do with their segment numbers, which means that if N20 program segment appears ahead of N10 program segment, the N20 will be executed first.

If a program segment is started with "/", it means the program segment is "if" program segment, meaning when the jump switch is at up position, this program segment is not executed, and when the jump switch is at down position, this program segment can still be executed.

1) Main program and subprogram

Processing program is divided into main program and subprogram. Generally, the NC executes the instructions of main program, but it turns to subprogram when there is a subprogram calling instruction. It executes the subprogram until it meets the return instruction and get back to main program.

If we need to run the same track for several times, we can edit this segment of track as subprogram and save it in program memory of machine tool so that every time when you execute this segment of track in program, you can call this subprogram.

When a main program is calling a subprogram, this subprogram can also call another subprogram; we call this as double nesting of subprogram. Generally, a machine tool is allowed to have at most quadruple subprogram nesting. In instruction of calling subprogram, you can execute the called subprogram repeatedly for as many as 999 times.

One subprogram should be in a format as follows:

O××××; No. of subprogram
……;
……; content of subprogram
……;
M99; Return to main program

At the beginning of program, there should be a subprogram number specified by the address O. Instruction M99 for returning main program is essential at the end. M99 does not have to be in an individual program segment, as the end of subprogram, the following program segment also works:

G90 G00 X0 Y100. M99;

In main program, the program segment for calling subprogram should contain the following content:

M98 P×××××××;

Here, the later four digits of numbers behind P are used to specify the program number of called subprogram, and the front three digits are used to specify the repetition time of calling.

M98 P51002; Call No. 1002 subprogram for 5 times
M98 P1002; Call No. 1002 subprogram for 1 time
M98 P50004; Call No. 4 subprogram for 5 times

Subprogram call instruction and motion instruction can be in the same program segment:

G90 G00 X–75. Y50. Z53. M98 P40035;

This program segment instructs X, Y, and Z axes to move to the specified position at quick locating feed speed, and then call and execute No. 35 subprogram for 4 times.

Different from other M codes, M98 and M99 do not send signal to machine tool side when they are executed.

NC will give out alarm if program No. specified by address P is not detected.

The subprogram cannot call M98 in MDI mode. If it's required to call a subprogram individually, you can edit the following program in editing mode, and then execute it in auto run mode.

O×××;
M98 P××××;
M30;

2) End of program

At the end of program when there are following codes, it means it's the end of program.

EIA	ISO	Meaning
M30 CR	M30 LF	The program ends and returns to the beginning of program.
M99 CR	M99 LF	End of subprogram

If such end code as above is detected when executing the program, the program will be stopped and changed to reset status. If it is M30 CR or M30 LF, it will return to the beginning of the program (in auto way). If it is at the end of subprogram, it returns to the program that calls the subprogram.

3) End of file

EIA	ISO	Meaning
ER	%	End of program

Note: If there is no M30 at the end of program but ER(EIA) or %(ISO) is executed, CNC will be changed to reset status.

1.2 Preparatory function (G code)

1.2.1 List of G codes

G code	Group	Function
G00	01	Positioning (Quick move)
G01		Linear interpolation (Cutting feed)
G02		Circular interpolation CW(Clockwise)
G03		Circular interpolation CCW(Anti-clockwise)
G04	00	Pause, warrant stop
G17	02	XY plane selection
G18		ZX plane selection
G19		YZ plane selection
G20	06	Imperial data input
G21		Metric data input
G28	00	Return to reference point
G29		Return from reference point
*G40	07	Tool radius compensation cancellation
G41		Left tool radius compensation
G42		Right tool radius compensation
G43	08	Positive tool length offset
G44		Negative tool length offset
*G49		Tool length offset cancellation
*G54	05	Work piece coordinate 1
G55		Work piece coordinate 2
G56		Work piece coordinate 3
G57		Work piece coordinate 4
G58		Work piece coordinate 5
G59		Work piece coordinate 6
G591		Expansion work piece coordinate 7
G592		Expansion work piece coordinate 8
G593		Expansion work piece coordinate 9
G594		Expansion work piece coordinate 10
G595		Expansion work piece coordinate 11
G596		Expansion work piece coordinate 12
G597		Expansion work piece coordinate 13
G598		Expansion work piece coordinate 14
G599		Expansion work piece coordinate 15
G65	00	Macro program instruction (4340 is not developed yet, testing edition)
G73	09	Fixed cycle of deep hole drilling
G74		Fixed cycle of reverse-screw tapping
G76		Fixed cycle of precision boring
*G80		Fixed cycle of cancellation
G81		Fixed cycle of drilling
G82		Fixed cycle of drilling
G83		Fixed cycle of deep hole drilling
G84		Fixed cycle of tapping
G85		Fixed cycle of precision boring
G86		Fixed cycle of precision boring
G87		Fixed cycle of reverse precision boring
G88		Fixed cycle of precision boring
G89		Fixed cycle of precision boring
*G90	03	Absolute value programming
G91		Increment value programming
G98	10	Return to original plane from fixed cycle

G99		Return to R point plane from fixed cycle

Note: Item with * is the default mode value of all groups of G codes for the system.

1.2.2 Interpolation (G00, G01, G02, G03)

1) Quick positioning (G00)

Format :

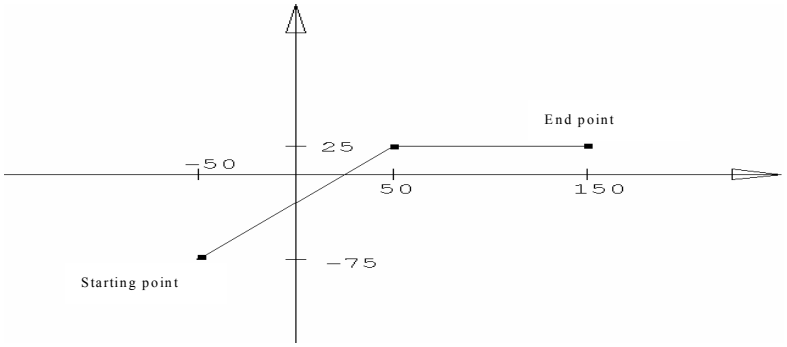
G00 X_Y_Z_ ;

X_Y_Z_: coordinate value, determine whether it is absolute position value or increment position value according to the mode value of G90 or G91

G00 instruction is used to allow each axis to move to the appointed position at the set quick move speed. The motion of each axis is not related, which means that the track of tool is a line or a polyline. Under the G00 instruction, the speed of all axes: X, Y, and Z axes are moved at speed set by parameters, which is not controlled by the current F value. When all motion axes arrive at end point, the CNC will consider it as the end of program segment and turn to execute the next one.

Example of G00 program:

Starting point is X-50, Y-75. ; instruction G00 X150. Y25.; and the tool will move in track as follows:



2) Linear interpolation (G01)

Format:

G01 X_Y_Z_F_;

X_Y_Z_ : refer to as the coordinate value, it is absolute value or increment value according to the status of G90 or G91 at that time

F : Speed

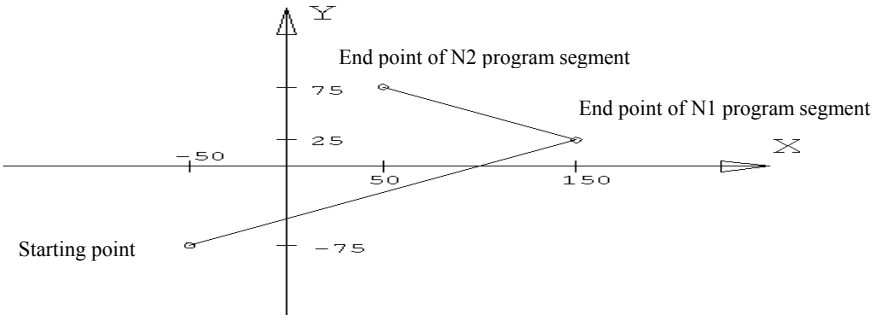
G01 instruction enables the current interpolation mode to be linear interpolation. The tool moves from the current position to position appointed by IP, and the track is a line. F specifies the speed of tool moving along the line, unit in mm/min.

Example of G01 program:

Suppose the current tool is at X-50. Y-75., the following program segment will allow the tool to move in track as the following picture:

N1 G01 X150. Y25. F100 ;

N2 X50. Y75.;

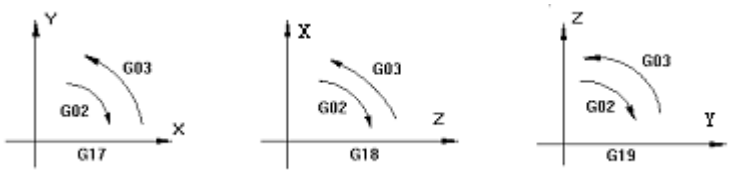


3) Circular interpolation (G02/G03)

The following instructions can allow the tool to move along the circular track:
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--Z plane
G19 { G02 / G03 } Y__ Z__ { (J__ K__) / R__ } F__ ;

S/N	Content		Instruction	Meaning
1	Plane selection		G17	Specify the circular interpolation on X--Y plane
			G18	Specify the circular interpolation on Z--X plane
			G19	Specify the circular interpolation on Y--Z plane
2	Direction of circular arc		G02	CW circular interpolation
			G03	CCW circular interpolation
3	End point position	G90 mode	Instruction of 2 axes among X, Y, Z	coordinate value of end point in current work piece coordinate system
		G91 mode	Instruction of 2 axes among X, Y, Z	Distance from starting point to end point (with direction)
4	Distance between starting point and center		Instruction of 2 axes among I, J, K	Distance from starting point to center (with direction)
	Radius of circular arc		R	Radius of circular arc
5	Feed rate		F	the speed moving along the circular arc

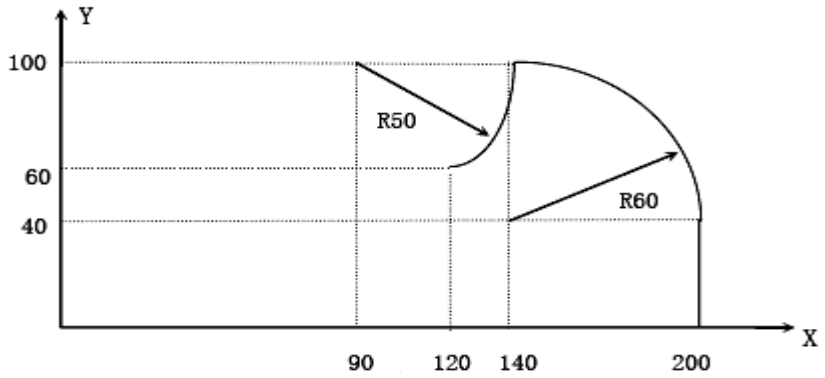
Here, the direction of circular arc, for X--Y plane, is that when viewing the plane from positive to negative of Z axis. Likewise, for X-Z or Y-Z plane, the viewing direction should be from the positive of Y or X to the negative of Y or X (applicable to right handed coordinate system, as follows).



The end point of circular arc is determined by address X, Y, and Z. In G90 mode (absolute value mode), address X, Y, and Z give the coordinate value of end point of circular arc at the current coordinate system. In G91 mode (increment value mode), address X, Y, and Z give the distance between the current point of tool to end point at the direction of each coordinate axis.

In X direction, address I gives the distance from current point of tool to center. In Y and Z directions, this distance is specified by address J and K. The sign of I, J, and K values is determined by their motion directions.

To program a segment of circular arc, other than using specified end point and center positions, we can also use specified radius and end point position, use address R to specify the radius value, replacing the address to specify center position. Positive R value is used to program a circular arc of less than 180˚, and a negative R value is to program a circular arc of more than 180˚. To program a circle, you can only use the method of specified center.



The track in above picture is programmed in absolute value mode and increment value mode respectively:

(1) Absolute value mode

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 mode

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;

The feed speed of circular interpolation is specified by F, and it is the speed the tool moves along the circular arc at tangent direction.

1.2.3 Pause instruction (G04)

Purpose: produce a pause between two program segments

Format: G04 P-

G04 X-

Address P specifies the pause time, if there is no decimal, the minimum unit for instruction is 0.001s.

Address X specifies the pause time, if there is no decimal, the minimum unit for instruction is 1s.

For example: G04 P 1000: pause 1000 milliseconds, equal to 1s

G04 X 1: pause 1s

1.2.4 Plane selection (G17, G18, G19)

This group of instructions is to choose the plane on which the circular interpolation and tool radius compensation are done. The methods are as follows:

G17……select XY plane

G18……select ZX plane

G19……select YZ plane

If G17, G18, and G19 are in program segment without instruction, the plane does not change.

For example:

G18 X_ Z_ ; ZX plane

X_ Y_ ; plane does not change (ZX plane)

In addition, moving instruction is not related to the plane selection. For example, under the following instruction, Z axis is not on XY plane and Z axis movement has nothing to do with XY plane.

G17 Z_ ;

For related instructions for plane selection, please refer to the relevant contents of circular interpolation and tool compensation instructions.

1.2.5 Instructions of Coordinate System (G53~G59, G591~G599, G92)

1) Machine tool coordinates (G53)

Format: G53 X_Y_Z_;

X_Y_Z_: the absolute coordinate value or relative position

If the instruction is executed in G90 mode, the tool moves to the coordinate of machine tool coordinate system specified by IP_ at quick feed speed. If the instruction is executed in G91 mode, the tool is moved at the increment value of selected coordinate system. G53 instruction is a modeless instruction, which means that it works only in the current program segment.

The distance between origin of machine tool coordinate system and the reference point is set by the parameters. Without any special explanation, the reference points of all axes and the origin of machine tool coordinate system are coincided.

2) Preset work piece coordinate system (G54~G59, G591~G599)

According to the clamp position of work piece on machine tool, the system can preset as many as 6 work piece coordinate systems (9 coordinate systems for new edition). Set the offset of each origin of work piece coordinate system from that of machine tool coordinate system via the LCD panel, and then use the G54~G59, G591~G599 instructions to choose them. G54~G59, G591~G599 instructions are mode instructions, and they are corresponding to 1#~5# preset work piece coordinate systems respectively, as the following shows:

Preset offset of 1# work piece coordinate system: X-150.000 Y-210.000 Z-90.000
Preset offset of 4# work piece coordinate system: X-430.000 Y-330.000 Z-120.000

Content of program segment	Coordinate value of end point at the machine tool coordinate system	Explanation
N1 G90 G54 G00 X50. Y50.;	X-100, Y-160	Select 1# coordinate system, and quick positioning
N2 Z-70.;	Z-160	
N3 G01 Z-72.5 F100;	Z-160.5	Linear interpolation, F is 100
N4 X37.4;	X-112.6	(Linear interpolation)
N5 G00 Z0;	Z-90	Quick positioning
N6 X0 Y0 A0;	X-150, Y-210	
N7 G53 X0 Y0 Z0;	X0, Y0, Z0	Select 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 interpolation, F is 100 (mode value)
N11 X37.4;	X392.6	
N12 G00 Z0;	Z-120	
N13 G00 X0 Y0 ;	X-430, Y-330	

Seen from the above examples, we got to know that the purpose of G54~G59 instructions is to move the origin of coordinate system used by NC to the coordinate of preset value in machine tool coordinate system. For presetting methods, please refer to the operation parts of this manual.

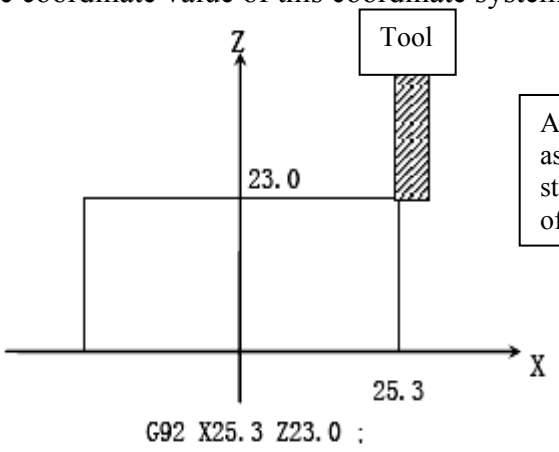
Switch on the machine and return to the origin of machine tool, the workpiece coordinate systems 1~6 are then created. G54 is the initial mode when electrified. The absolute position is the coordinate value of current coordinate system.

In NC programming of machine tool, except otherwise specified, the IP in interpolation instructions and other instructions related to coordinate value refers to the coordinate position in current coordinate system (the coordinate system used when the instructions are executed). In most conditions, the current coordinate system is one of that of G54~G59. It is rare to use the machine tool coordinate system directly.

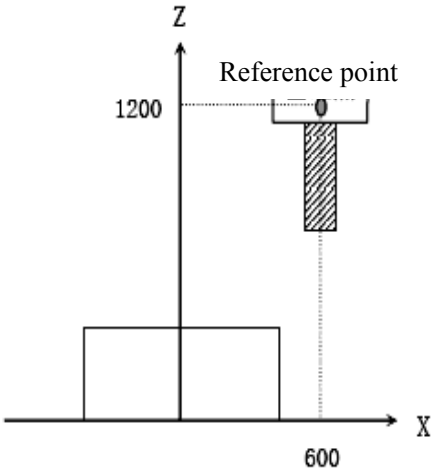
3) Programmable work piece coordinate system (G92)
Format: (G90) G92 X_Y_Z_ ;

This instruction builds a new work piece coordinate system, in which the coordinate value of current point where the tool is located is the value of IP_ instruction. G92 instruction is a modeless instruction, but the work piece coordinate system built by this instruction is of mode type. Actually, this instruction also gives an offset indirectly, which is the coordinate value of origin of new work piece coordinate system in original one. Viewed from functions of G92, we know that the offset is the difference of tool coordinate value in original work piece coordinate system and the IP_ instruction value. If G92 instruction is used for many times, the offset will be added for each using of G92 instruction. For each preset work piece coordinate system (G54~G59), this added offset is valid.

New coordinate system of parts is set by using the above instructions, for example, the coordinate value of tool tip is IP_. Once the coordinate is confirmed, the position of absolute value instruction is the coordinate value of this coordinate system.



As the picture shows, take the tool tip as the starting point of program, and start G92 instruction at the beginning of program.



As the picture shows, take a reference point on tool holder as the tool start point, and start G92 instruction at the beginning of program. If moved in accordance with absolute value instruction in program, the reference point will move to the specified position. Tool length compensation should be added and the value is the distance between reference point and tool tip.

Use G92 X600.0 Z1200.0 instruction to set the coordinate system (take a reference point on tool holder as the tool start point).

Note: a. If G92 is used in tool offset to set the coordinate system, the tool length compensation is the coordinate system set by G92 before adding tool offset.
b. For tool radius compensation, tool offset should be cancelled when using G92 instruction.

For example:
Preset offset of 1# work piece coordinate system: X-150.000 Y-210.000 Z-90.000
Preset offset of 4# work piece coordinate system: X-430.000 Y-330.000 Z-120.000

Content of program segment	Coordinate value of end point at the machine tool coordinate system	Explanation
N1 G90 G54 G00 X0 Y0 Z0;	X-150, Y-210, Z-90	Choose 1# coordinate system and position quickly to the origin
N2 G92 X70. Y100. Z50.;	X-150, Y-210, Z-90	Tool does not move, build up new coordinate system, and the coordinate value of current point in new coordinate system is X70, Y100, Z50
N3 G00 X0 Y0 Z0;	X-220, Y-310, Z-140	Quick position to the origin of new coordinate system
N4 G57 X0 Y0 Z0;	X-500, Y-430, Z-170	Choose 4# coordinate system and position quickly to the origin (offset)
N5 X70. Y100. Z50.;	X-430, Y-330, Z-120	Position quickly to the origin

4) Local coordinate system (G52)

G52 can build a local coordinate system, which is equal to sub coordinate system of G54~G59 coordinate systems.

Format: G52 X_Y_Z_ ;

In this instruction, IP_ gives a offset relatively to the current G54~G59 coordinate system, which means that IP_ specifies the position coordinate of origin of local coordinate system in the current G54~G59 coordinate system, even when a G52 instruction has created a local coordinate system before the G52 instruction is executed. It is also very simple to cancel the local coordinate system, just use the G52 IP0.

1.2.6 Reference point related instructions (G27, G28, G29)

The establishment of machine tool coordinate system is done by operation of returning to reference point every time when NC is electrified. Reference point is a fixed point on machine tool, and its position is determined by the mounting position of block switches of all axes and the origin position of all axes servo motor. After the machine tool returned to reference point, the coordinate value of reference point in machine tool coordinate system is X0, Y0, Z0.

Return to reference point automatically (G28)

Format: G28 IP_;

This instruction makes the instruction axis return to reference point of machine tool through the intermediate point specified by IP at quick positioning feed rate. The intermediate point can be specified in absolute value mode or increment value mode, which is determined by the current mode. Generally, this instruction is used to move the work piece out of the processing area after the program with the purpose of offloading the done parts and feeding the parts waiting to be processed.

When executing G28 instruction before returning to reference point manually, the motion from intermediate point for each axis is the same as that of returning to reference point manually, and the direction of motion from the intermediate point is positive.

The coordinate value in G28 instruction is saved by NC as intermediate point. On the other hand, if an axis is not included in G28 instruction, the intermediate point coordinate value of this axis saved by NC will use the previous value specified in G28 instruction.

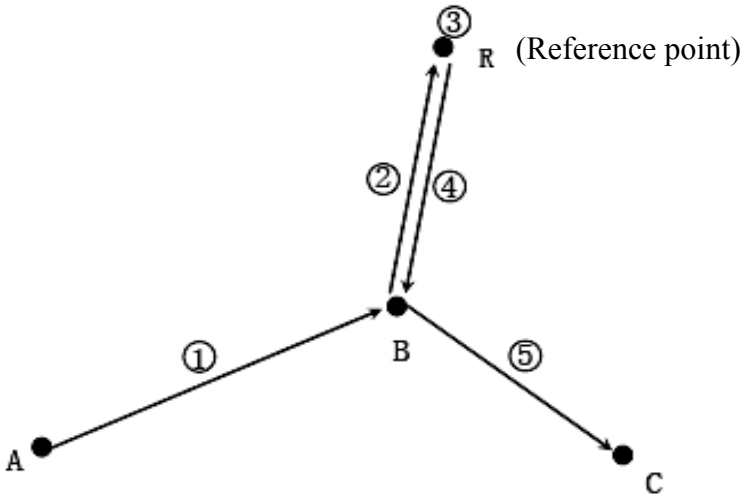
For example:

N0010 X20.0 Y54.0;

N0020 G28 X-40.0 Y-25.0; Coordinate value of intermediate point (-40.0,-25.0)

N0030 G28 Z31.0; Coordinate value of intermediate point (-40.0,-25.0,31.0)

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



☞Note:

In tool offset mode, tool offset is also effective to G27 instruction. For the safety, it is usually to cancel the tool offset (radius offset and length offset) before executing G28 instruction.

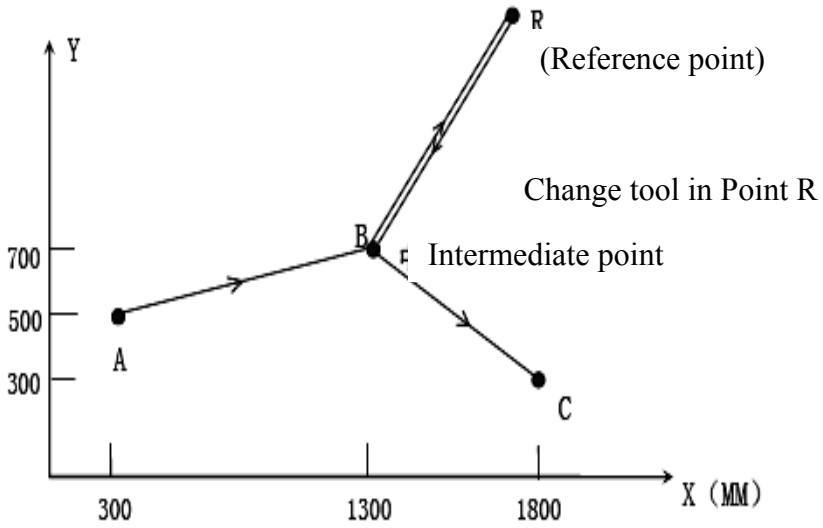
Return from reference point automatically (G29)

Format: G29 IP-;

This instruction makes the instruction axis return from reference point through the intermediate point to appointed position at quick positioning feed rate. The position of intermediate point is determined by previous G28 instruction. Generally, this instruction is used after G28 when the instructed axis is located at reference point or the second reference point.

In increment value mode, the instruction value is the distance between intermediate point and end point (instruction position).

G28, G29 application examples:



G28 X1300.0 Y700.0 ; (program of A→B)

……

G29 X1800.0 Y300.0 ; (program of B→C)

The above examples clearly show that in program, it is not required to calculate the detailed movement from the intermediate point to reference point.

Note: when changing the coordinate system of parts after passing through the intermediate point to reference point via the G28 instruction, the intermediate point is also moved to the new coordinate system. After that, when execute the G29 instruction, the positioning is done in appointed position via intermediate point in new coordinate system.

Reference point return check (G27)

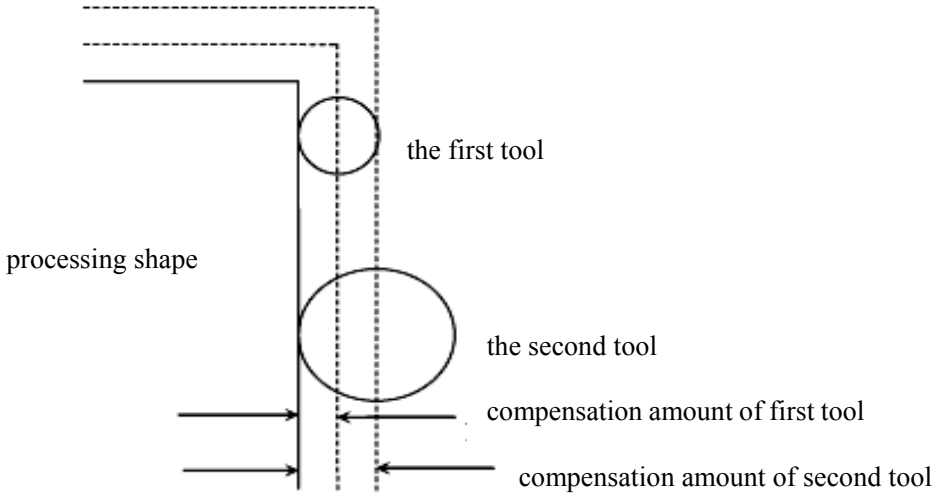
Format: G27 IP_;

This instruction makes the instruction axis move to the position specified by IP at quick positioning feed rate, and then checks whether the point is the reference point. If so, send out complete signal for the return of reference point of this axis (light the indicator for reference point arrives). If not, send out an alarm and stop the program.

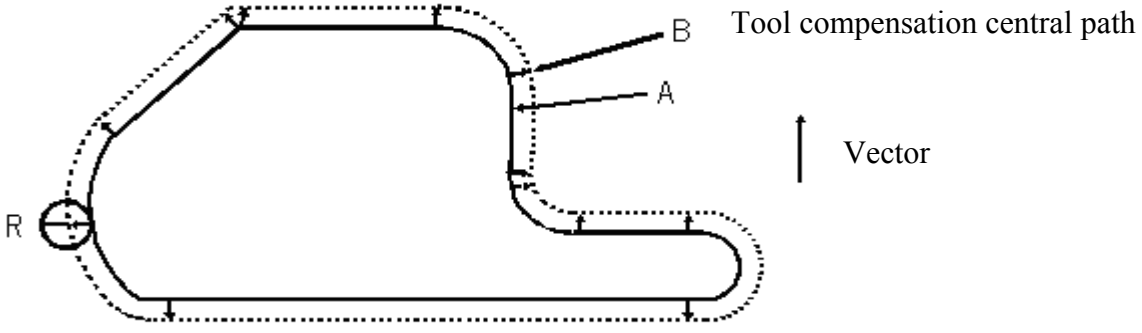
1.2.7 Tool compensation (G40, G41, G42, G43, G44, G49)

1) Tool radius compensation

The tool has a size (length, diameter). When processing a part of certain shape, the moving track of tool will be different due to the difference of tool. If the size data of tool is preset in CNC, the tool track will then be generated by CNC automatically in the same program, even for different tools. The tool size data are called as compensation amount (or offset).



As the following picture, use the tool of radius R to cut work piece A, the central path of tool is B, and the distance between B and A is R. tool leaves a certain distance away from work piece A, this is called as compensation. The programmers build work programs with the tool radius compensation mode. During the processing, they determine the tool radius and set it in CNC, and the tool path will be changed to compensation path B.



Compensation and

2) Compensation amount (D code)

This system can set as many as 18 D00-D18 compensations. Compensation refers to the two digits after the D code in program. The compensation should be set in [Tool compensation] menu.

The range of compensation is set as follows:

	Input in mm	Input in inch
Compensation	0-±999.999mm	0-±999.999inch

3) Compensation vector

The compensation vector is the 2-dimensional vector, equal to the compensation specified by D code. The calculation of compensation vector is done within the control unit, and in every program segment its direction is changed according to the tool path. This compensation vector is done in control unit so that it is convenient to calculate how much compensation should be given for the tool movement. Compensation path (central track of tool) is the result of programming path adding or subtracting (determined by compensation direction) the tool radius.

Compensation vector is always related to the tool. During the programming, it is very important to know the status of vector.

4) Plane selection and vector

Calculation of compensation is done in plane selected by G17, G18, and G19, which is called compensation plane. For example, when choosing XY plane, the program uses (X, Y) or (I, J) to execute the compensation calculation and vector calculation. The coordinate value of axis not in compensation plane is not influenced.

When using controller of three axes at the same time, only the tool path projected to the compensation plane is compensated.

The change of compensation plane should be done after canceling the compensation mode. If it is done in compensation mode, the system will give an alarm and the machine will stop at the same time.

G code	Compensation plane
G17	X-Y plane
G18	Z-X plane
G19	Y-Z plane

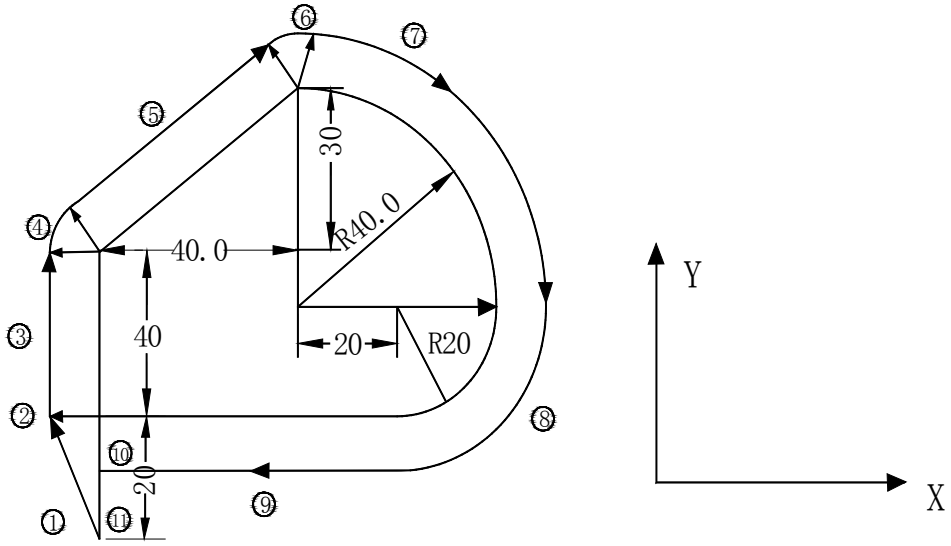
5) G40, G41 and G42

Use G40, G41, and G42 instructions to cancel or execute the tool radius compensation vector. These instructions are combined with G00, G01, G02, and G03 instructions, defining a mode to determine the value of compensation vector, direction, and the moving direction of tool.

G code	Function
G40	Cancel tool radius compensation
G41	Tool radius left compensation
G42	Tool radius right compensation

G41 or G42 allows the system to enter the compensation mode, and the G40 allows the system to cancel the compensation mode.

the compensation program is as follows:




```
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 called as start-up, and the G41 instruction in this segment turns the compensation cancellation mode to compensation mode. In the end of this segment, the tool center is compensated at the direction of tool radius perpendicular to the next program path. The tool compensation is specified by D07, which means the compensation number is set as 7, and the G41 represents the tool path left compensation.

6) Details of tool radius compensation C

This section is to describe the tool radius compensation C in details.

a. Cancellation mode

when the system is electrified/reset or the program has executed M02, M30 instructions, the system is in tool compensation cancellation mode.

Vector in this mode is always 0, and the central path of tool and the programming path are consistent. In cancellation mode, G40 should be specified before the end of program.

b. Starting compensation

In cancellation mode, the system enters the compensation mode when the program segment satisfying the following conditions starts to run.

- Contain G41 or G42 instruction, or control to enter the G41 or G42 mode
- Offset number of tool compensation is 0.
- For the movement of any axis (except I, J, K) on compensation plane, the movement should not be zero.

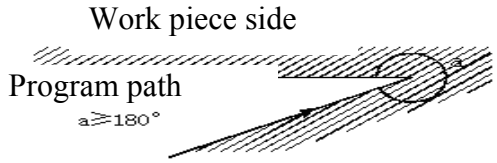
In program segments at the beginning of compensation, there should be no circular instruction G02 and G03; otherwise, it will have an alarm (P/S34). In the starting segment of compensation, read into two program segments, the first one of which is read and executed, and the second one is read into the tool compensation buffer area.

Read into two program segments in single program segment mode, execute the first one, and then stop.

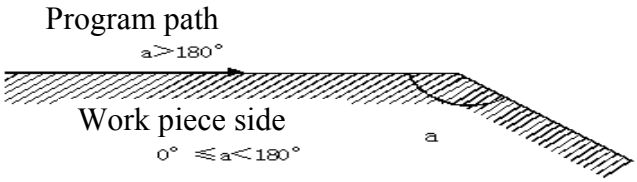
In continuous execution, it is usually pre-read into two program segments, so there are three program segments in CNC. One is the program segment being executed, and the other two as below are entering into the buffer area.

Note: the definition of following common glossaries “inside” and “outside” is that: when the inclination at the crossing point of two moving program segments is larger or equal to 180°, it is called as “inside”, and when the inclination is within 0-180°, it is called as “outside” (see the following picture):

Inside

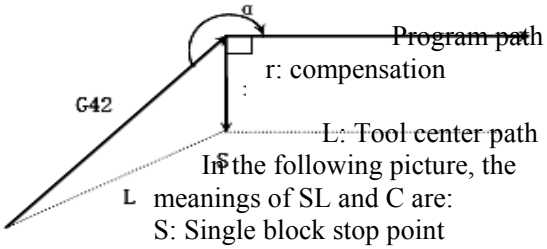


Outside



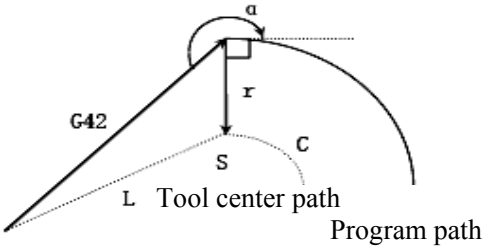
(a) Move along the inside of corner (a≥180°)

(i): Line →Line



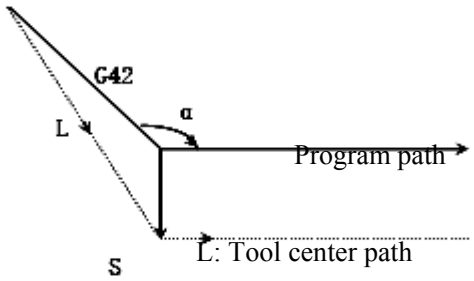
In the following picture, the meanings of SL and C are:
S: Single block stop point
L: Line
C: Circular arc

(ii): Line → Circular arc

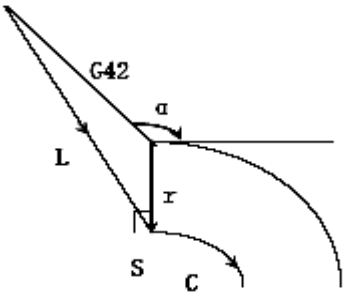


(b) Move along the outside of corner at obtuse angle (180°>a≥90°)

(i): Line →Line

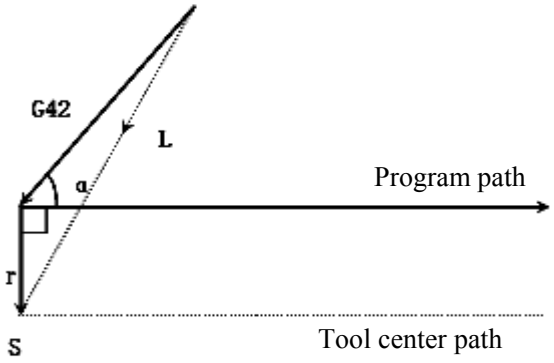


(ii): Line → Circular arc

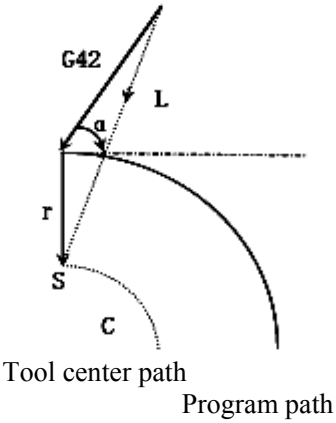


(c) Move along the outside of corner at acute angle (a<180°)

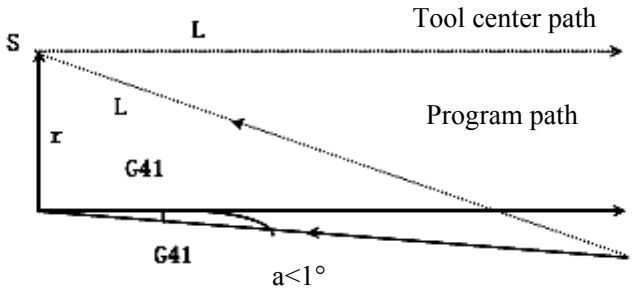
(i): Line →Line



(ii): Line → Circular arc



(d) Move along the outside of corner at less than 1°, line →line (a<1°)

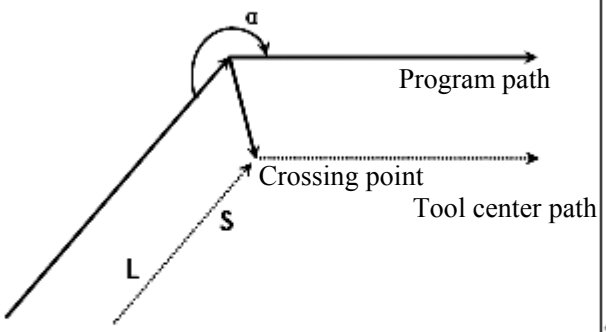


c. Compensation mode

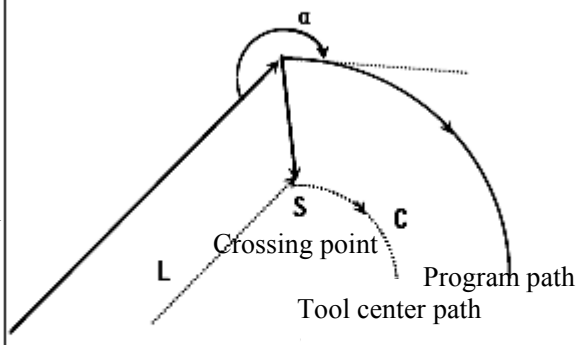
In compensation mode, if you do not appoint two or more non-moving instructions (auxiliary function or pause, etc.) continuously, the compensation will be executed properly; otherwise, there will be over cutting or short of cutting. Compensation plane cannot be modified when in compensation mode; otherwise, it will give out alarm and the tool will stop.

(a) Move along the inside of corner (a≥180°)

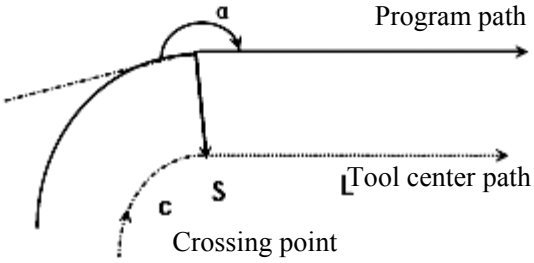
(i) Line →Line



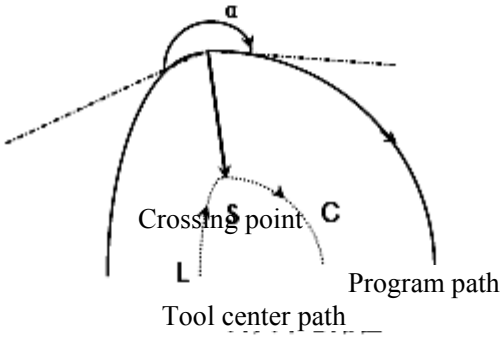
(ii) Line →Circular arc



(iii) Circular arc →Line

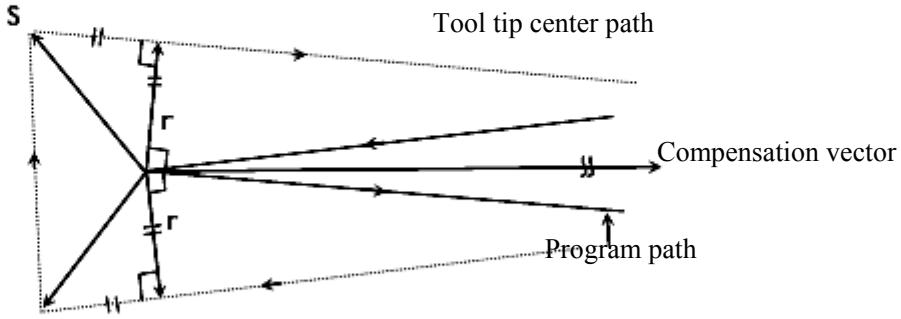


(iv) Circular arc →Circular arc



(v) processing <1° inside and enlarging compensation vector

(I) Line →Line



Considering the following conditions with the same method

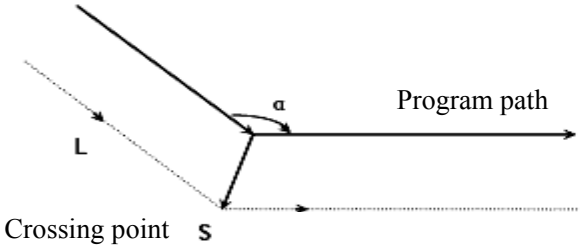
(II) Circular arc →Line

(III) Line →Circular arc

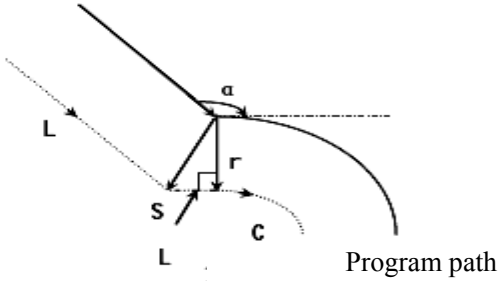
(Iv) Circular arc →Circular arc

(b) Move along the outside of corner at obtuse angle (180°>a≥90°)

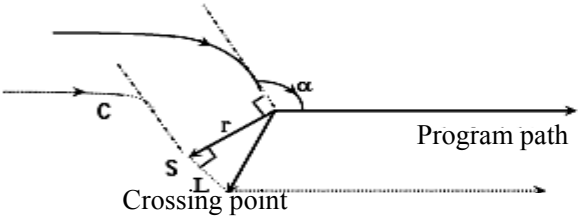
(i) Line →Line



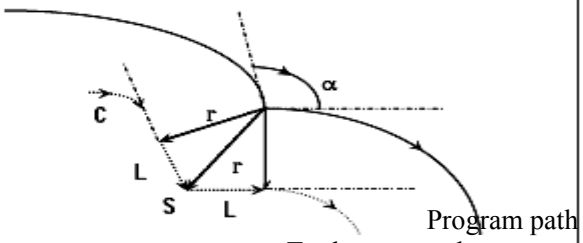
(ii) Line →Circular arc



(ii) Circular arc →Line

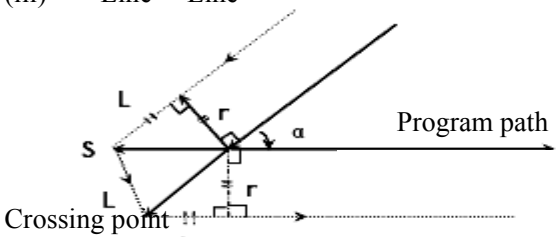


(iv) Circular arc →Circular arc

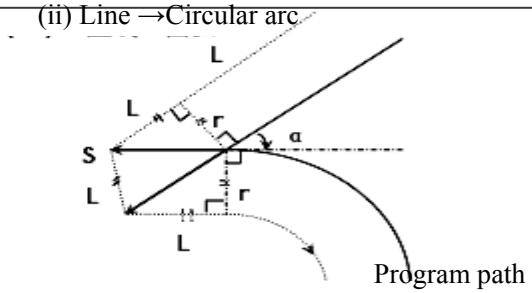


(c) move along the outside of corner at acute angle (a<90°)

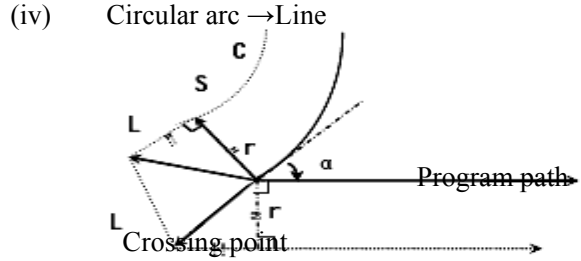
(iii) Line →Line



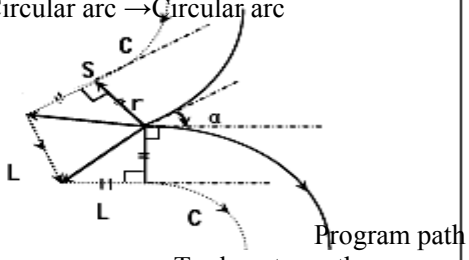
(ii) Line →Circular arc



(iv) Circular arc →Line

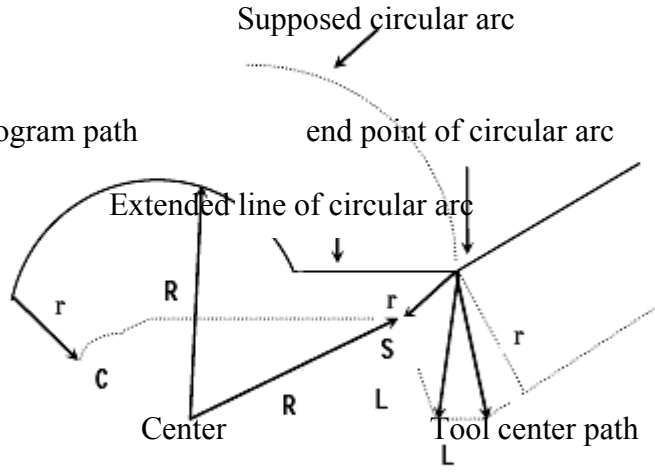


(iv) Circular arc →Circular arc



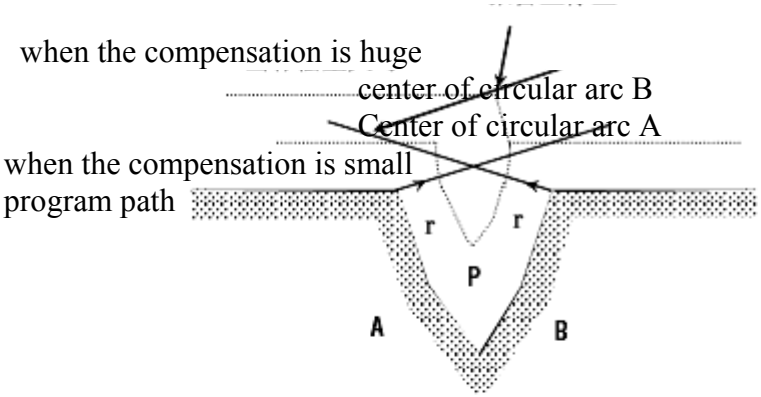
(d) special conditions

(i) end point of circular arc is not on circular arc



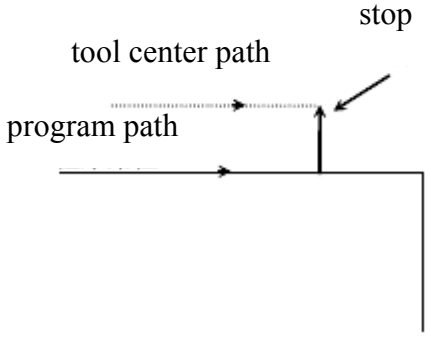
When the circular arc is not on end point, the extended line is as shown in the left picture. Suppose a circular arc comes across the end point, and the compensation takes the supposed circular arc as vector, the tool center path is different from the offset path while considering the extended line of circular arc.

(ii) when without crossing point



In the left picture when the tool radius is small, there will be a crossing point for the compensation path of circular arc. However, when the radius become bigger, the crossing point may disappear. The tool will stop at the end point of previous program segment and an alarm may occur.

(iii) the center of circular arc is consistent with starting point of end point



As shown in the left picture, an alarm may occur and the tool will stop at the end point of previous program segment.

```
(G41)
N5 G01 X1000;
N6 G02 X1000 I0 J0;
N7 G03 Y-1000 J-1000.;
```

d. Compensation mode

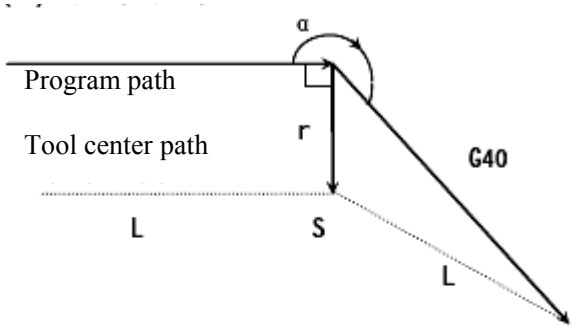
In compensation mode, the system enters the compensation cancellation mode when the program satisfying any following conditions is executed, and the action of this program segment is called as compensation cancellation.

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

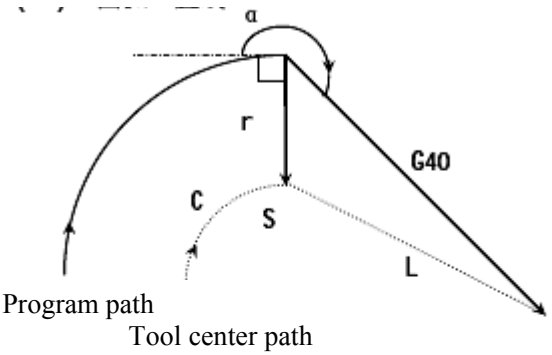
When executing the compensation cancellation, circular arc instruction (G03 and G02) cannot be used; otherwise, it will give out an alarm (P/S34) and the tool will stop.

(a) Move along the inside of corner (a>180°)

(i) Line →Line



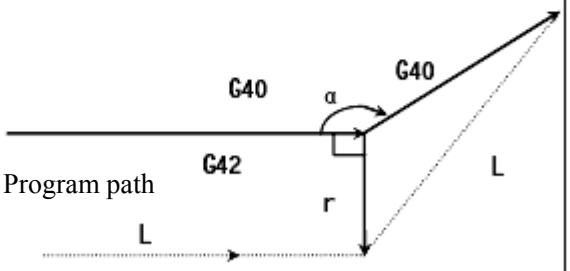
(II) Circular arc →Line



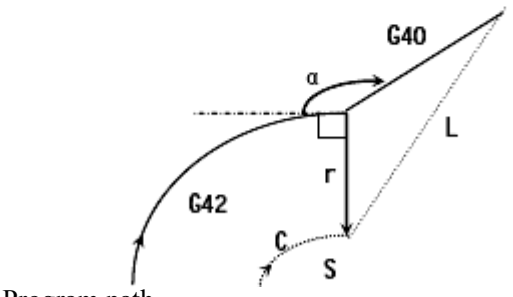
(b) Move along the inside of corne

(90˚≤a≤180°)

(I) Line →Line

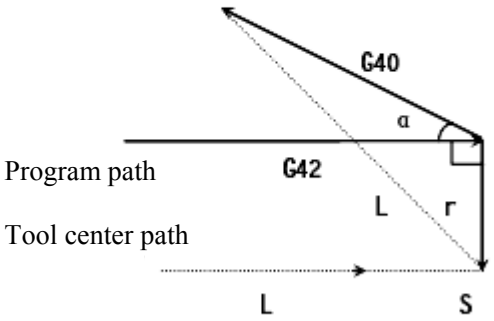


(II) Circular arc →Line

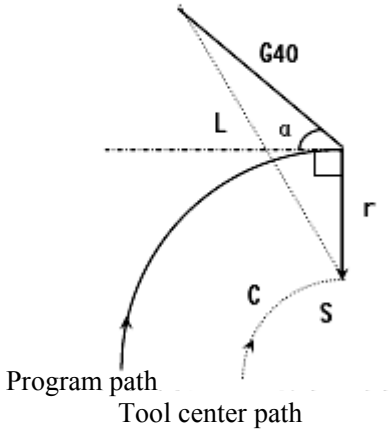


(c) Move along the outside of acute angle corner (a<90°)

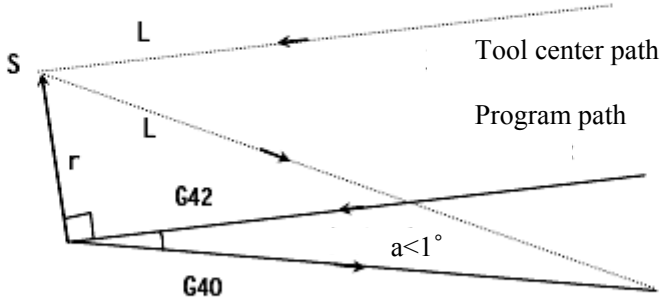
(i) Line →Line



(II) Circular arc →Line



(d) Move along the outside of acute angle less than 1°, line →line (a<1° =



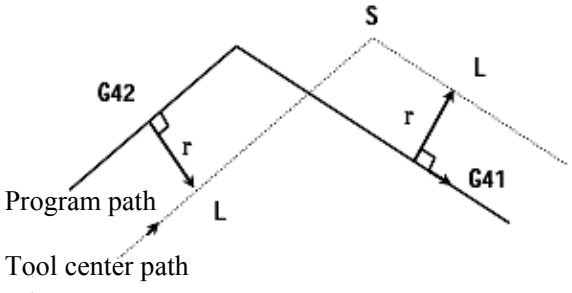
e. Change compensation direction in compensation mode

Tool radius compensation G codes (G41 and G42) determine the direction of compensation. The sign of compensation is as follows:

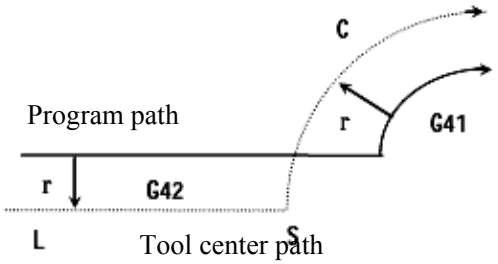
Sign of compensation / G 码		
G41	Left compensation	Right compensation
G42	Right compensation	Left compensation

In special occasion, it is able to change the compensation direction in compensation mode, but it is unable to change the starting program segment and the later program segments. When changing the compensation direction, there is no way of saying inside and outside. The following compensation is supposed to be positive.

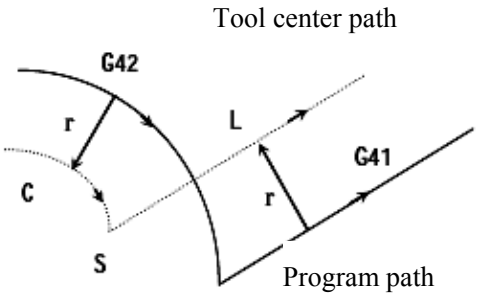
(i) Line →Line



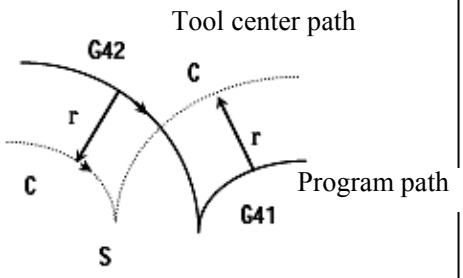
(II) Circular arc →Line



(iii) Circular arc →Line



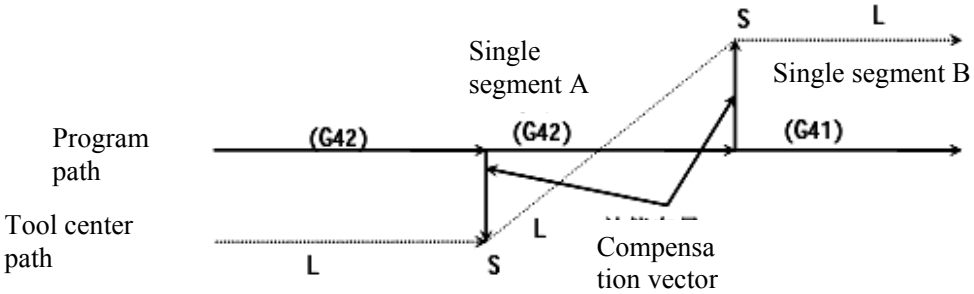
(iv) Circular arc →Circular arc

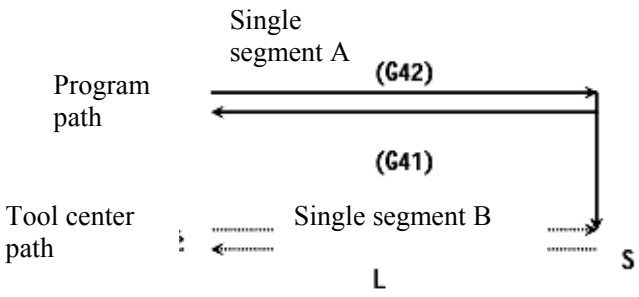


➢ If the compensation is executed normally, but there is no crossing point

When using G41 and G42 to change the offset direction from program segment A to B, if it is not required to compensate the crossing point of path, make a vector at the starting point of program segment B that is vertical to program segment B.

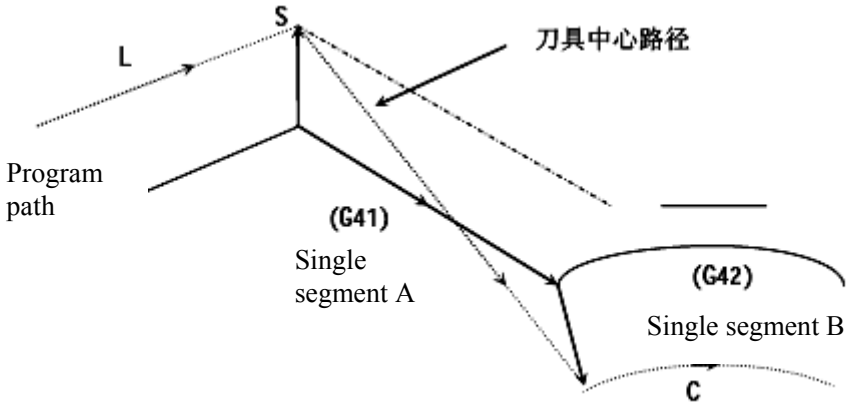
✧ Line----Line



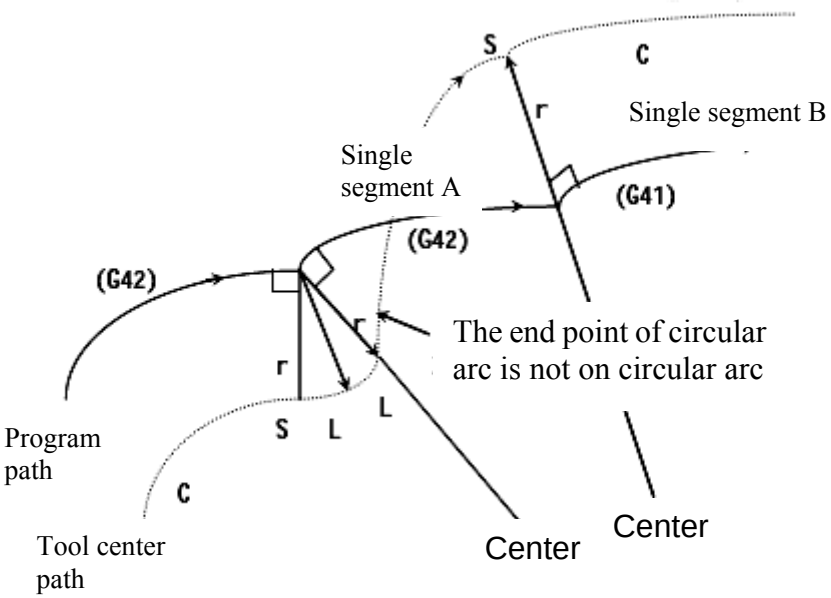


✧ Line----Circular arc

Tool center
path

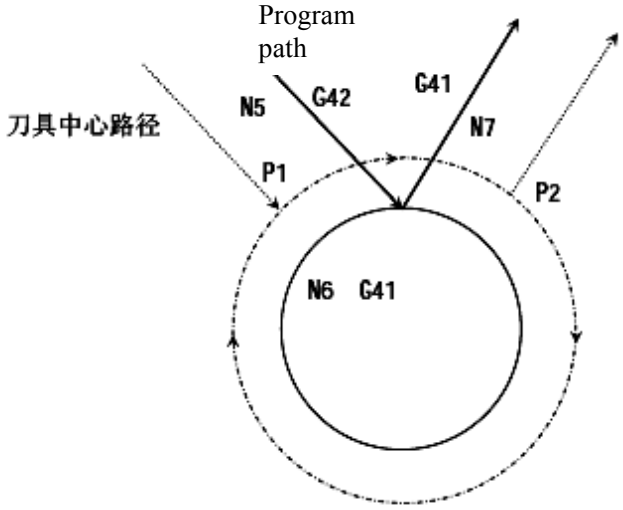


✧ Circular arc ---- Circular arc



➢ In tool radius compensation, when the length of tool center path is over a circle

Usually, this situation would not happen. However, when G41 and G42 are changed, or when I, J, or K instruction G40 is used, this situation may happen.



(G42)
N5 G02G91X5000Y-7000;
N6 G41G02J-5000;
N7 G42G01X5000Y7000;

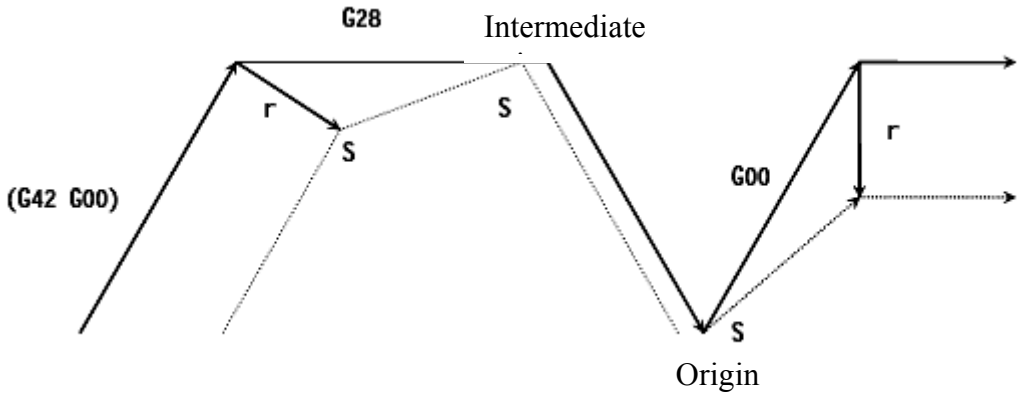
At this time, the tool center path is not a circular arc but a section of arc between P1 and P2
In some conditions, it may give an alarm may because of affecting the check.
If you want the tool to move along the circle, it should be instructed by segment.

f. Temporary compensation cancellation

In compensation mode, if the following instructions are appointed, the compensation will be cancelled temporarily. The system will resume the compensation mode automatically later. For detailed operations, please refer to details of compensation cancellation and compensation starting.

➢ G28 returns to reference point automatically

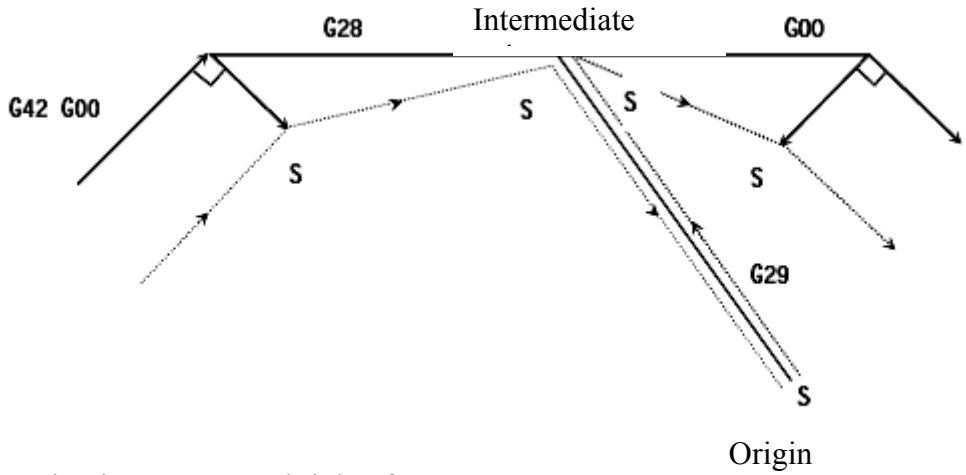
In compensation mode, if it is instruction G28, the compensation will be cancelled at intermediate point and the compensation mode is resumed automatically after the reference point is returned.



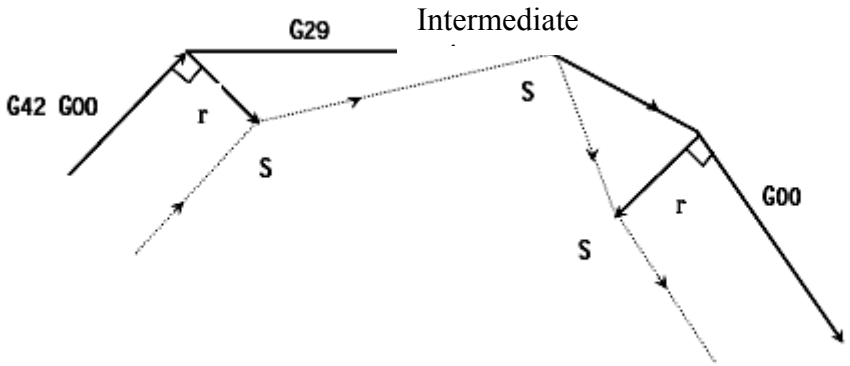
➢ G29 returns from reference origin automatically

In compensation mode, if it is instruction G29, the compensation will be cancelled at intermediate point, and the compensation mode will be resumed automatically in the next program segment.

When executing instruction immediately after G28



If the instruction is not executed right after G28

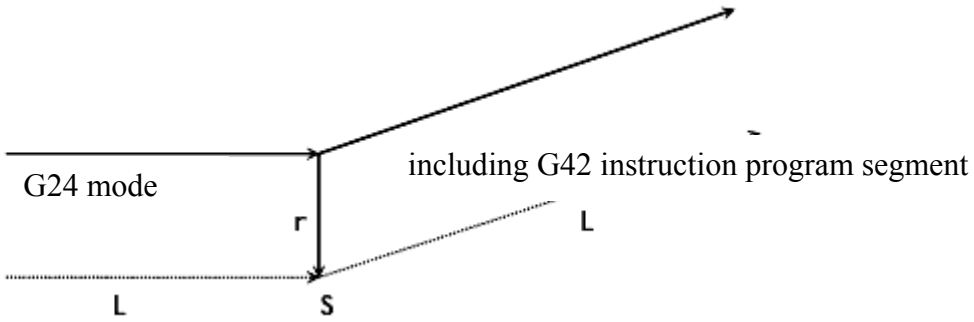


g. Tool radius compensation G code in compensation mode

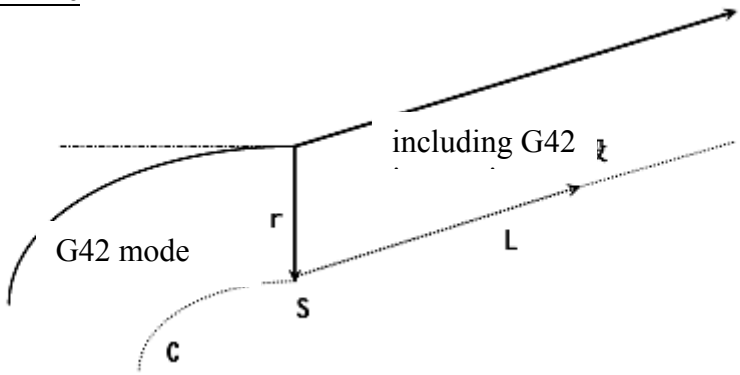
In compensation mode, when appointing tool radius compensation G code (G41, G42), it will form a vector that may form right angle with previous program segment, and it is nothing to do with the processing inside and outside. However, if you appoint this G code in circular arc instruction, you will not get the correct circular arc.

When using tool radius compensation G (G41, G42) to change the compensation direction, please refer to (5).

Line---line



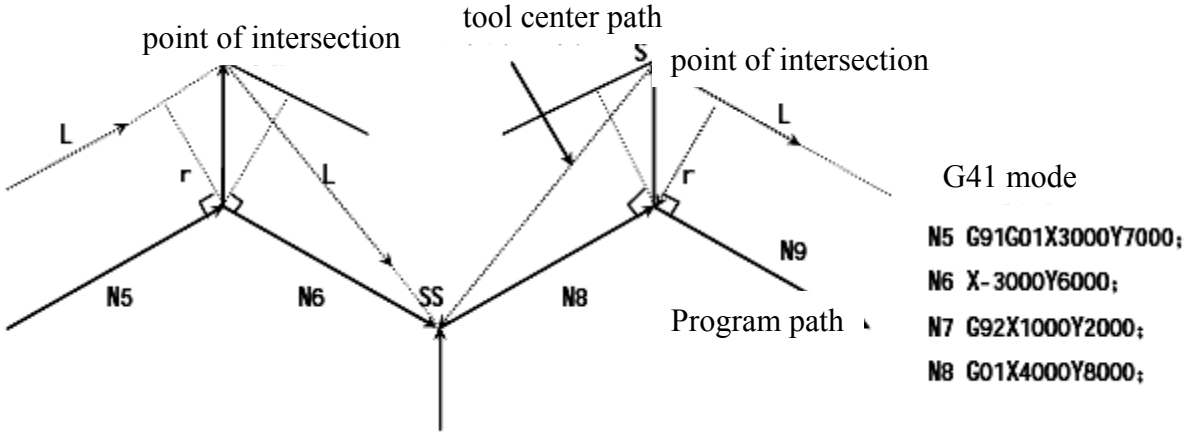
Circular arc---line



h. Instructions for canceling compensation vector temporarily

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

It is different from compensation cancellation mode, the tool is moved from the crossing point directly to instruction point of compensation vector cancellation. When resuming in compensation mode, the tool is also moved directly to the crossing point.



G50 program segment N7

Note: SS represents the point when the tool stops twice in single segment mode.

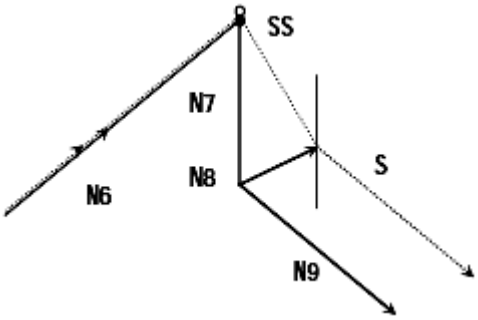
i. Program segment in which the tool does not move

There is no tool movement in the following program segments. In these segments, the tool would not move even there is crossing point in tool radius compensation mode.

(1) M05:…… M code output
(2) S21:…… S code output
(3) G04 X10000:…… Pause
(4) （G17）Z100:…… No moving instruction in compensation plane Not move
(5) G90:…… Only G code
(6) G01 G91 X0:…… Movement is 0

➢ Instructions at the beginning of compensation

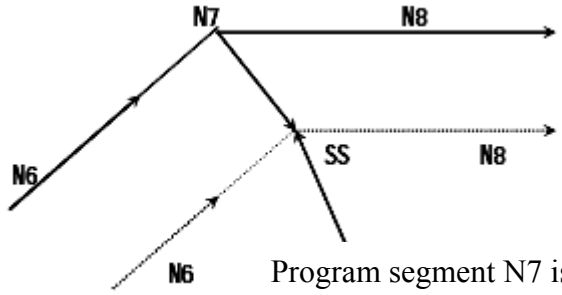
If there is no tool movement for the instructions at the beginning of compensation, it will not generate the compensation vector.




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

➢ Instructions in compensation mode

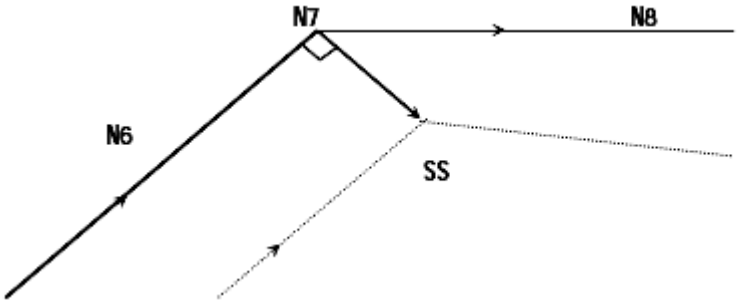
When only one program segment without tool movement is instructed in compensation mode, the vector and tool center path are the same as those when without this program segment.(refer to item (3) Compensation mode) At this time, the tool movement program segment is executed at the stop point of single program segment.




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

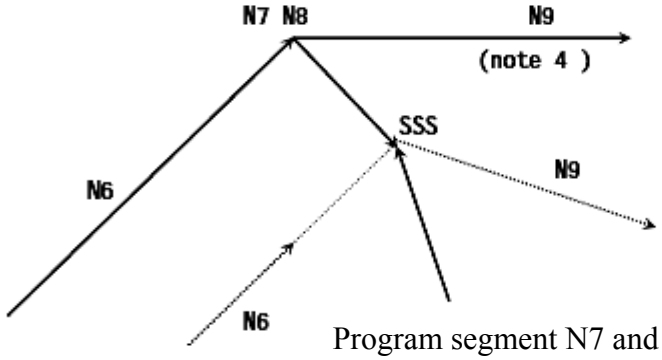
Program segment N7 is executed here

However, when the movement of program segment is 0, even only one program segment is appointed, the tool is still as though having no moving instruction. This will be described in details later.




```
N6 G91X1000 Y2000 ;
N7 X0 ;
N8 X1000 ;
```

Two program segments without tool movement instructions should not be executed successively. Otherwise, it will form a vector with length as compensation and the direction vertical to the moving direction of previous program segment, which will lead to the over cutting.



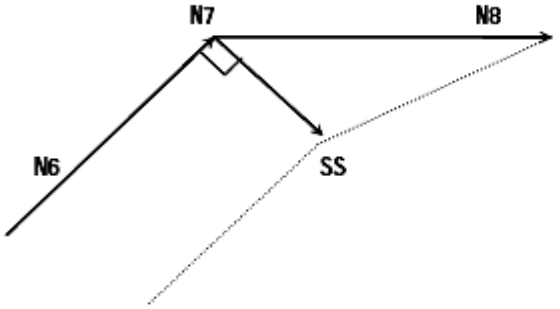

```
N6 G91X100.0 Y 200.0 ;
N7 S21 ;
N8 G04 X1.0 ;
N9 X100.0 ;
```

Program segment N7 and N8 are executed here

Note: SSS means to operate tool using program segment and stop for 3 times.

➢ When instructing with compensation cancellation

When the program segment instructing with compensation cancellation does not have tool movement instruction, it will form a vector with length as compensation and the direction vertical to the moving direction of previous program segment, and the vector will be cancelled in the next moving instruction.

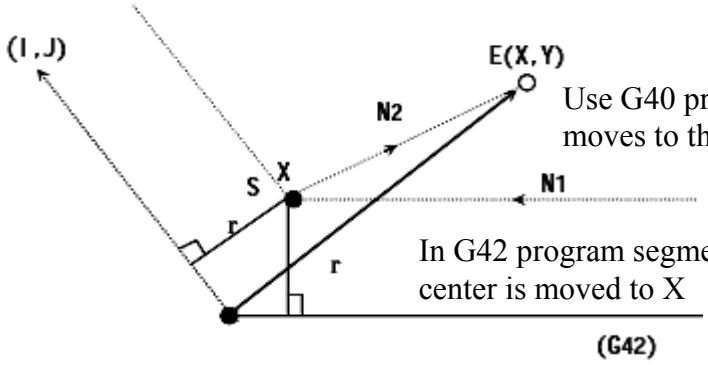



```
N6  G91X100.0 Y100.0 ;
N7  G40 ;
N8  X100.0 Y0 ;
```

j. In compensation plane, a program segment contains G40 and I-J-K instructions.

➢ the previous program segment is G41 or G42

Suppose the CNC has instructed the end point of previous program segment to execute movement at I, J or K direction.



Use G40 program segment, the tool moves to this position

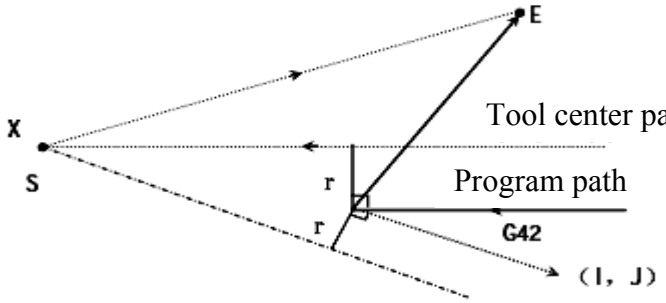
N1 (G42 mode)

N2 G40X□Y□I－J－;

In G42 program segment, the tool center is moved to X

Program path

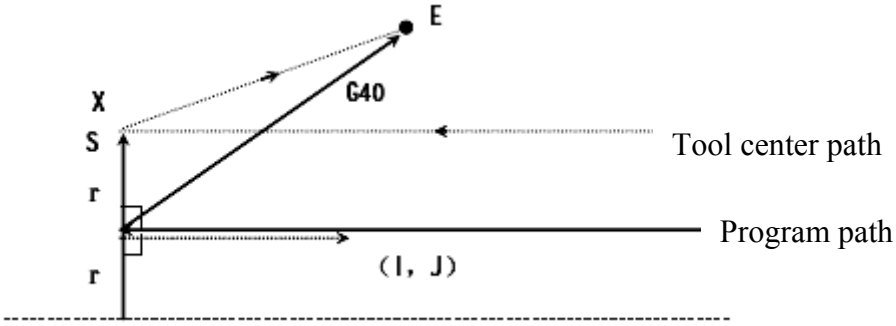
Note: the crossing point of tool path calculated by CNC has nothing to do with the appointed processing inside or outside.



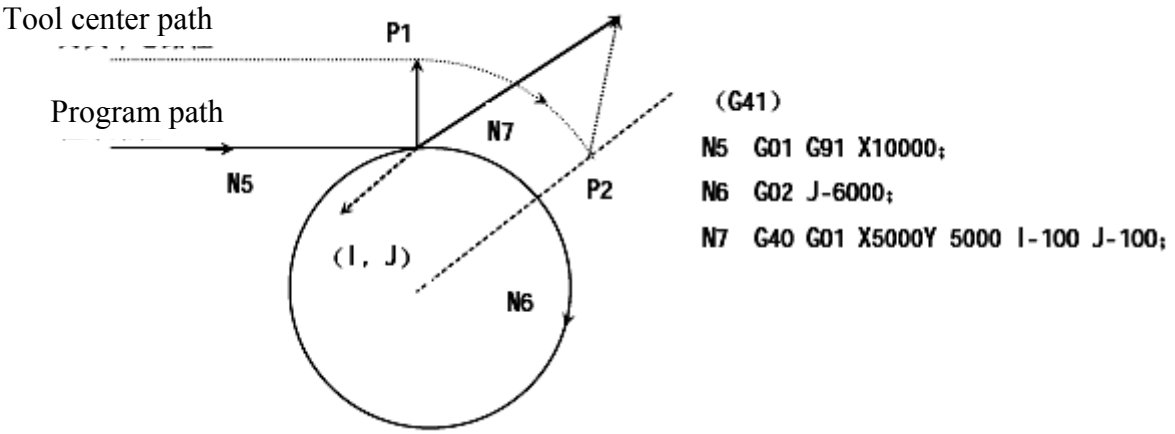
Tool center path

Program path

When crossing point cannot be calculated, the tool at end point of previous program segment is moved to a position that is vertical to the previous program segment.



➢ If length of tool center path is over one circle



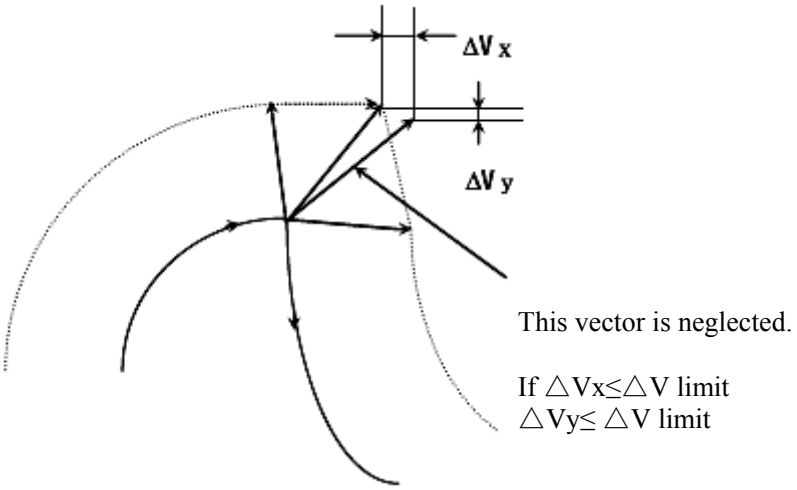
In the above picture, the tool center path is not moved along the circle but the circular arc from P1 to P2.

Under certain circumstances, it may as a result cause interference for the check and give an alarm (P/S41). This will be described later. (To move along the circle, the circular arc instruction should be separated.)

k. Corner moving

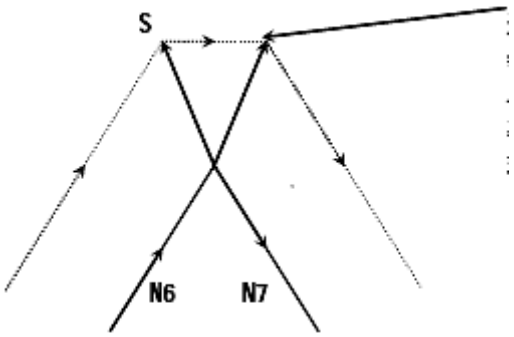
If there are two or more vectors at the end of a program segment, it means the tool is moved at straight line from a vector to the other vector, which is called corner moving.

If these vectors are almost the same, the corner moving is not executed, and the later vector can be neglected.



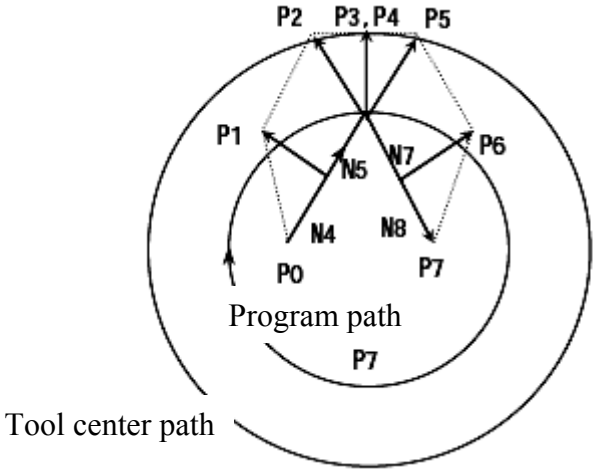
If △VX≤△V limit and △VZ≤ △V limit, the later vector is neglected. △V limit uses the parameter.

If these vectors are inconsistent and generate a movement along the corner, this movement is the later program segment.



the move belongs to program segment N7, and therefore the feed rate is equal to that of program segment N7. If the program segment N7 is G00 mode, the tool is moved at quick feed rate. If it is G01, G02 or G03 mode, the tool is moved at cutting feed rate.

However, if the path of next program segment exceeds the half circle, the above functions are not executed. The reason is that:



N4 G41G91X1500Y2000;
N5 X1500Y2000;
N6 G02J-6000;
N7 G01X1500Y-2000;
N8 G40X1500Y-2000;

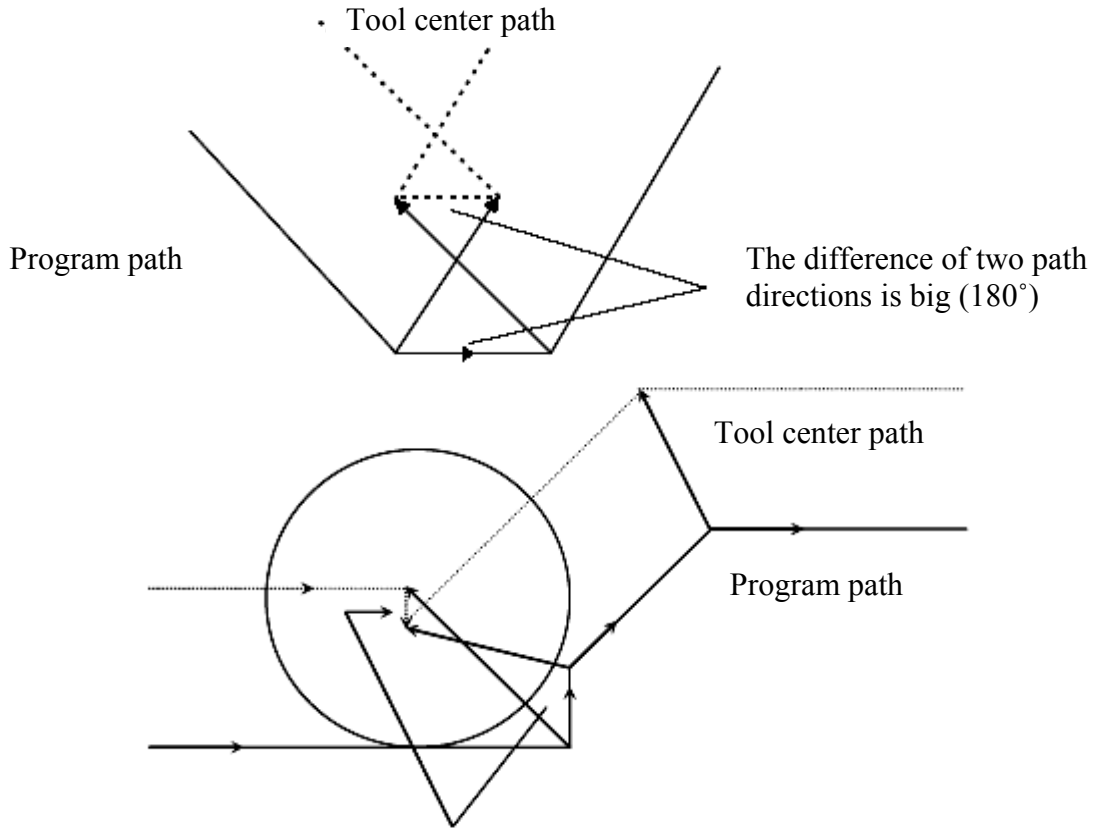
thIf the vector is not neglected, the tool path is as follows:
P0→P1→P2→P3 (Circular arc)→P4→P5→P6→P7
But if the distance between P2 and P3 is neglected, the P3 will be neglected. The tool path is as follows:
P0→P1→P2→P4→P5→P6→P7, circular arc cutting of program segment N6 is neglected.

l. Interference check

Over cutting of tool is called as “Interference”. Interference enable users to pre-check the over cutting of tool, but this function cannot check out all interferences. Interference check is also done even there is no over cutting.

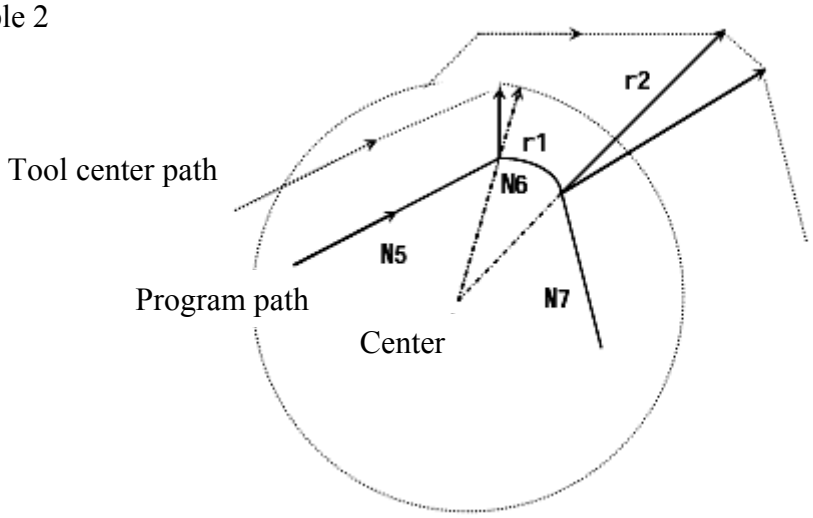
- Basic conditions of interference
  - The tool path direction is different from that of program path. (the inclination of paths is 90˚-270˚.)
  - When processing circular arc, apart from the above conditions, the inclination of starting point and end point of tool center path has a great difference from that of the program path (above 180˚)

Example 1



The difference of two path directions is big (180˚)

Example 2



(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 r1=2000)
(H02 tool radius compensation r2=6000)

In above examples, the circular arc in program segment N6 is within the first quadrant, but after the tool compensation, it is in the fourth quadrant.

- Pretreatment of interference
  - Interference incurred by vector movement

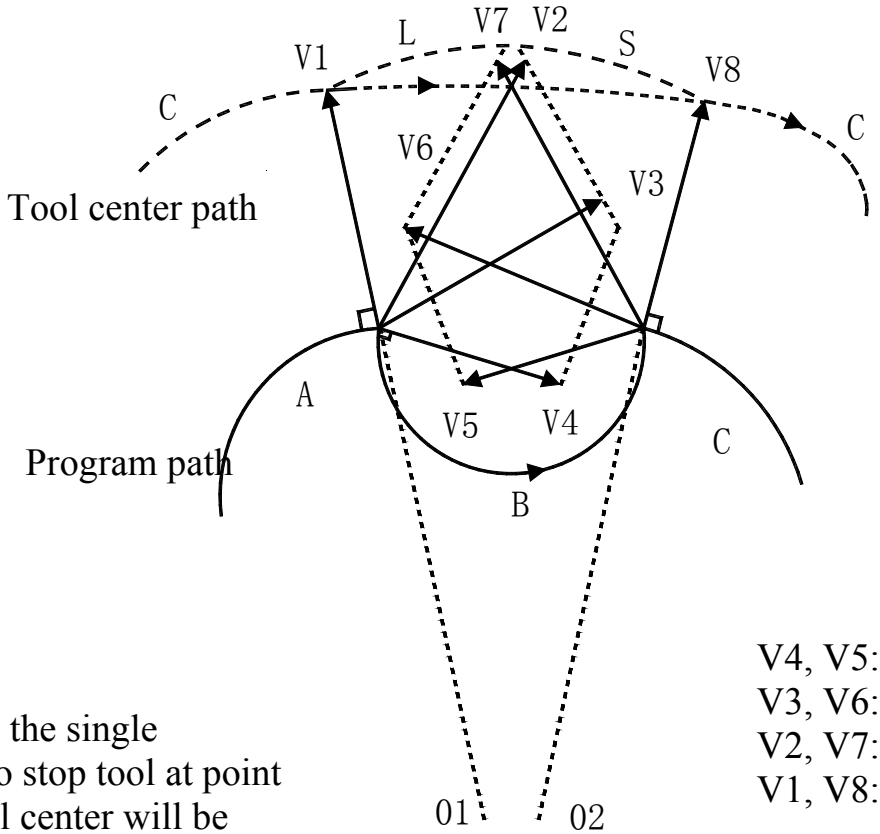
When tool compensation program segment A, B and C are executed, it will produce vector V1, V2,V3, and V4 between A and B, and V5, V6, V7, and V8 between B and C. First, check the latest vector. If there is interference, they will be cleared automatically. If the vector to be neglected is at the end of corner, they cannot be cleared.

Interference check:

Between V4 and V5——Interference——V4, V5 cleared
Between V3 and V6——Interference——V3, V6 cleared
Between V2 and V7——Interference——V2, V7 cleared
Between V1 and V8——Interference——V1, V8 cannot be cleared

If a vector has no interference during the check, the later vector is not checked. If the program segment B is moved along circular arc, the vector interference will produce straight line movement.

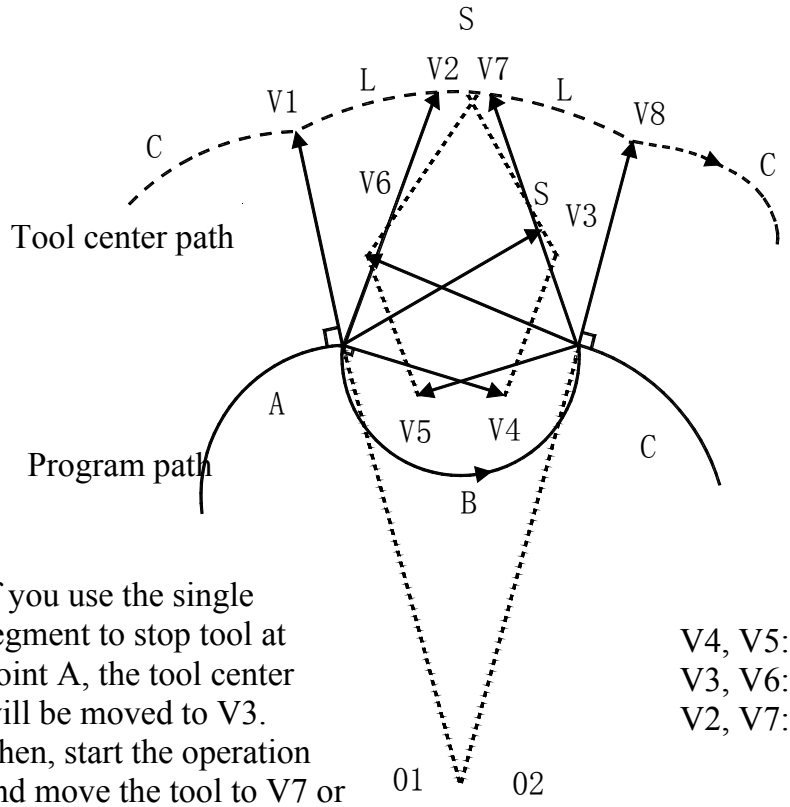
(Example 1) tool moves from V1 to V8 in straight line



If you use the single segment to stop tool at point A, the tool center will be moved to V3.

V4, V5: interfere
V3, V6: interfere
V2, V7: interfere
V1, V8: Not interfere

(Example 2) Tool straight line movement is as follows:
Tool path: V1→V2→VY→V8

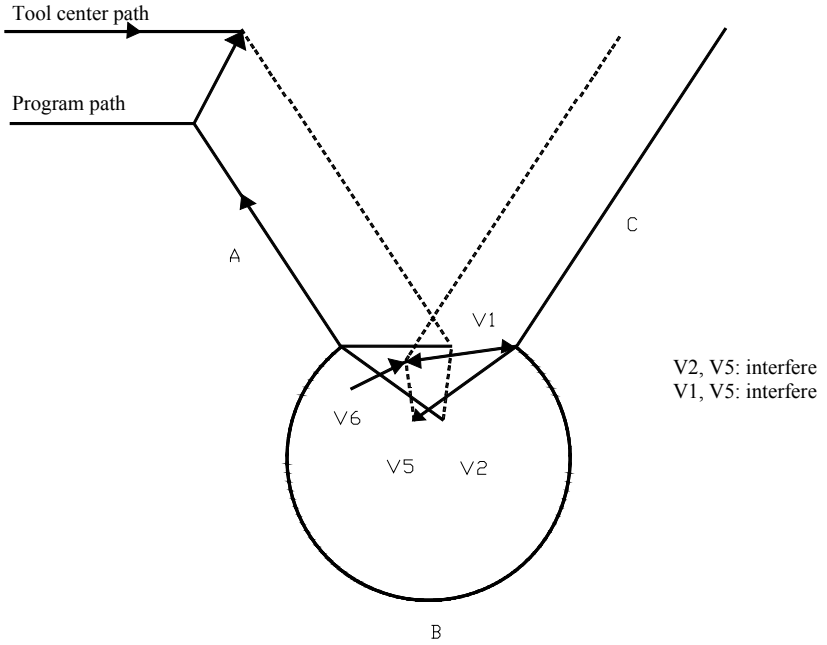


If you use the single segment to stop tool at point A, the tool center will be moved to V3. Then, start the operation and move the tool to V7 or V8.

V4, V5: interfere
V3, V6: interfere
V2, V7: Not interfere

- ✧ If there is still interference after treatment (1), the tool will stop and give an alarm. If interference occurs after treatment (1) or there is only one vector at the beginning of the check, the tool will stop after the execution of previous program segment, and give an alarm (P/S41).

(If executed by single program segment, the tool will stop at the end of the program segment.)

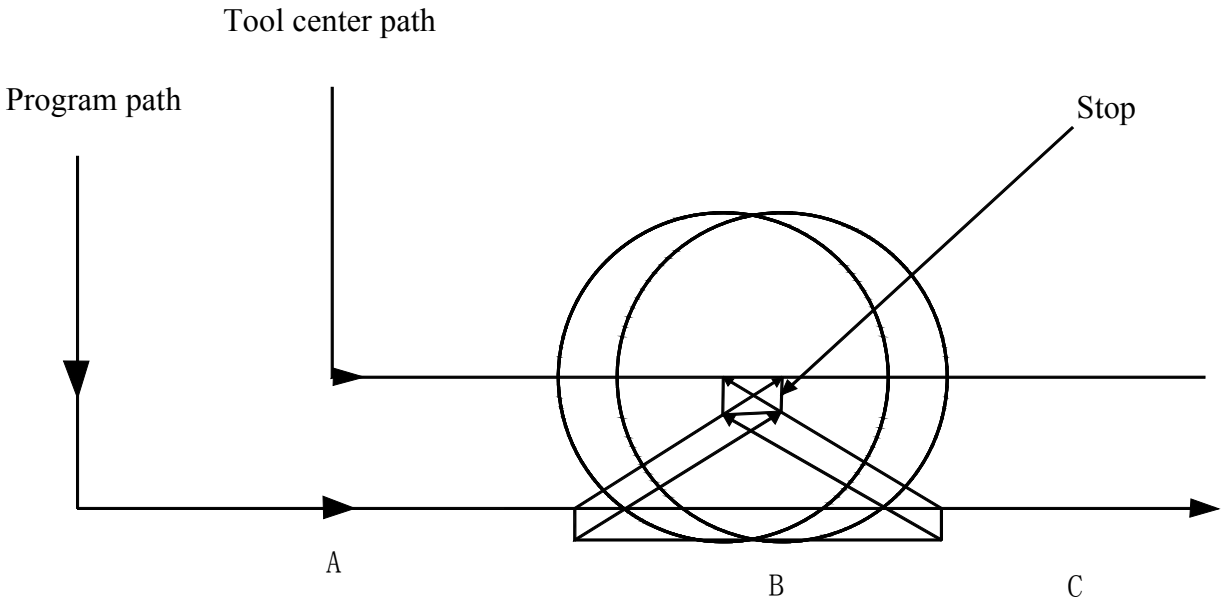


V2, V5: interfere
V1, V5: interfere

After the interference has neglected vector V2 and V5, the interference still occurs between V1 and V6. The alarm will be shown and the tool will be stopped immediately.

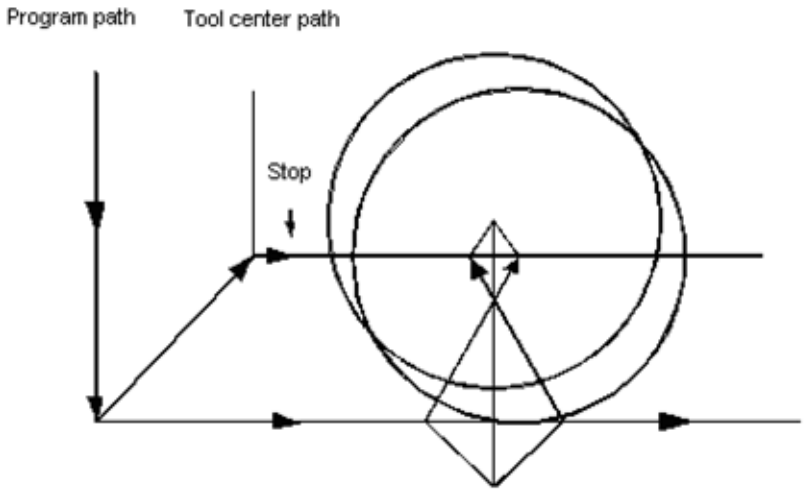
- ➢ In fact, there is no interference, but doing the interference check
  For example:

  - ✧ Recess depth is smaller than compensation



Actually, there is no interference, but because the tool is in program segment B, the program direction is opposite to the path of tool radius compensation, the tool stops and shows an alarm.

✧ Depth of cut-off trench is smaller than compensation

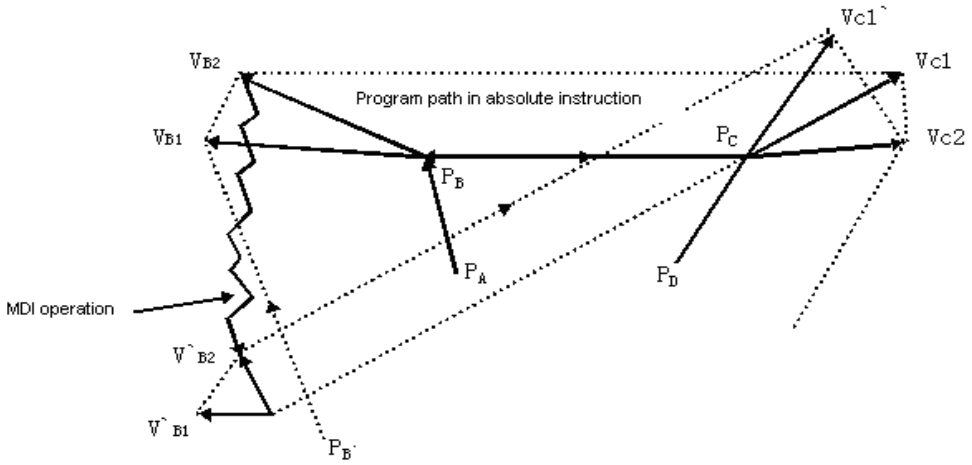


As example (1), the direction of tool path is opposite to that of program path.

m. It is unable to execute compensation by MDI

When using single segment to execute the stop during the auto execution of NC program (absolute instruction programming), insert the MD1 operation and then start the auto execution again. The tool path is as follows:

At this time, transmit the vector of starting point of the next program segment, and generate other vectors according to the next two program segments. Therefore, compensation from point Pc could be executed properly.



When Pa, Pb, Pc are programmed by absolute instruction, use single segment to execute the stop after the execution of program segment from Pa to Pb, and move the tool in MDI mode. Vector Vb1 and Vb2 are transferred to V ' b1 and V ' b2, so vector Vc1 and Vc2 of Pb→Pc and Pc→Pd are calculated again.

However, because vector Vb2 does not have re-calculation, compensation can be executed correctly after the Pc point.

n. Manual operation

For manual operation in tool tip radius compensation, please refer to the manual operation in operation chapter.

o. If tool length compensation is executed in tool radius compensation, the compensation of tool radius is regarded as the compensation change.

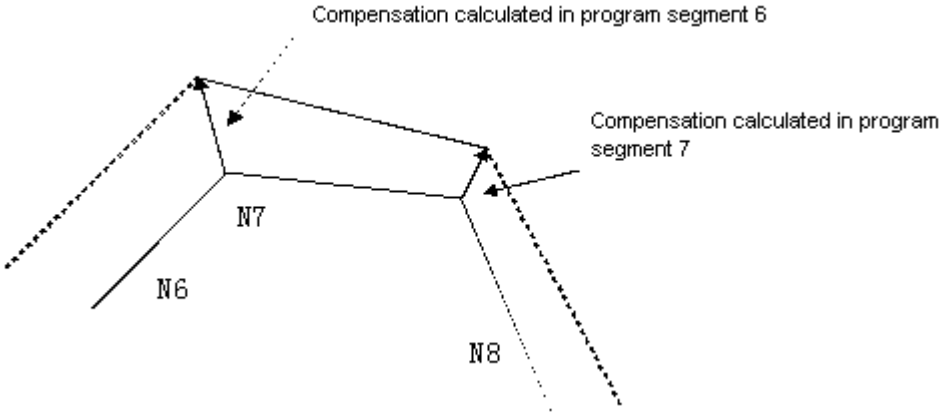
p. Notices for compensation

■ Instruction compensation

D code is used to specify the compensation number. Once specified, H code is valid until another H code is specified or the compensation is cancelled. Apart from specifying the compensation in tool radius compensation, H code can also be used to specify the offset of tool.

■ Change compensation

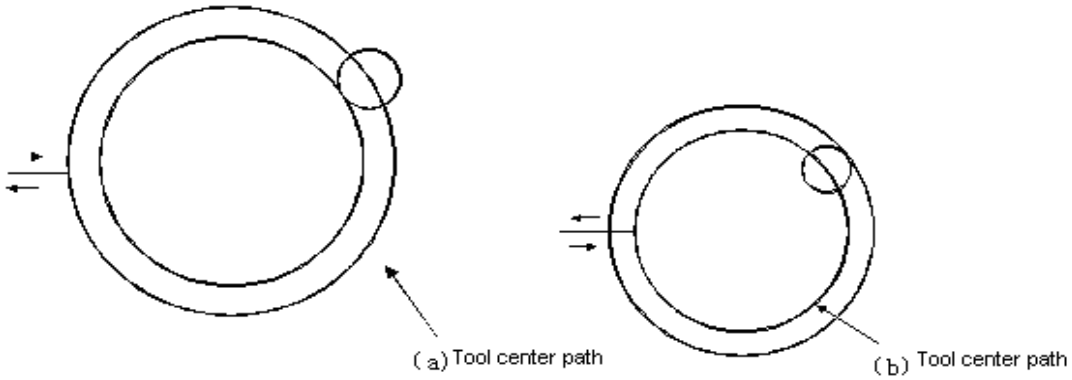
Usually when changing tool, the compensation should be changed in cancellation mode. If compensation is changed in compensation mode, calculate the new compensation at the end point of program segment.



Positive/Negative of compensation, and tool center path

If the compensation is negative (-), the G41 and G42 in program are exchanged. If the tool center moves along the outside of work piece, it will move along the inside, and vice versa.

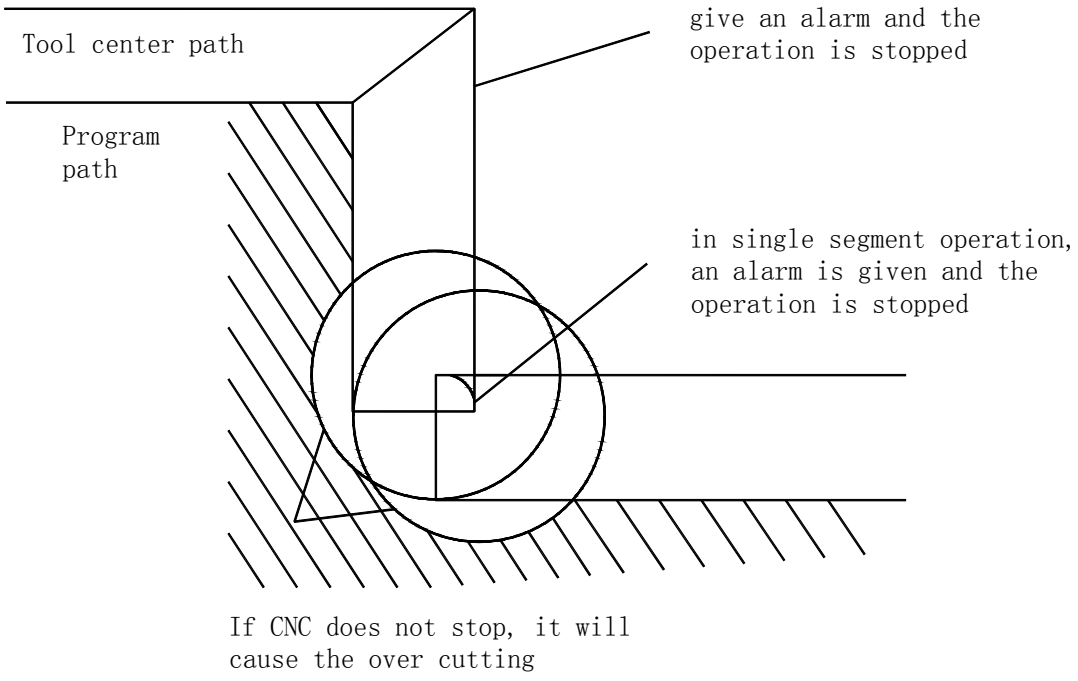
Generally, the compensation is (+) when programming. When tool path is programmed as picture (a), if the compensation is (-), the tool center movement is as picture (b), and vice versa. In this way, the same program can be cut into male or female type, and the gap between them can be adjusted by choosing the compensation. (applicable to the compensation start and cancellation is A type)



■ Compensate over cutting by tool radius

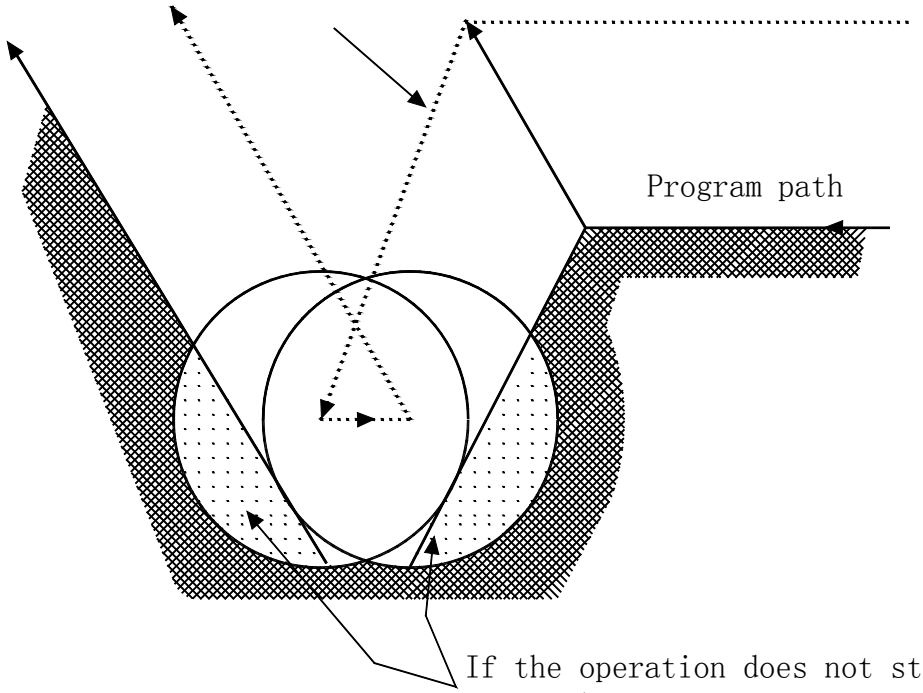
◆ when processing by circular inside of smaller tool radius

When the corner radius is smaller than tool radius, the inside compensation of tool will produce over cutting, giving the alarm. CNC stops at the starting position of single segment program.



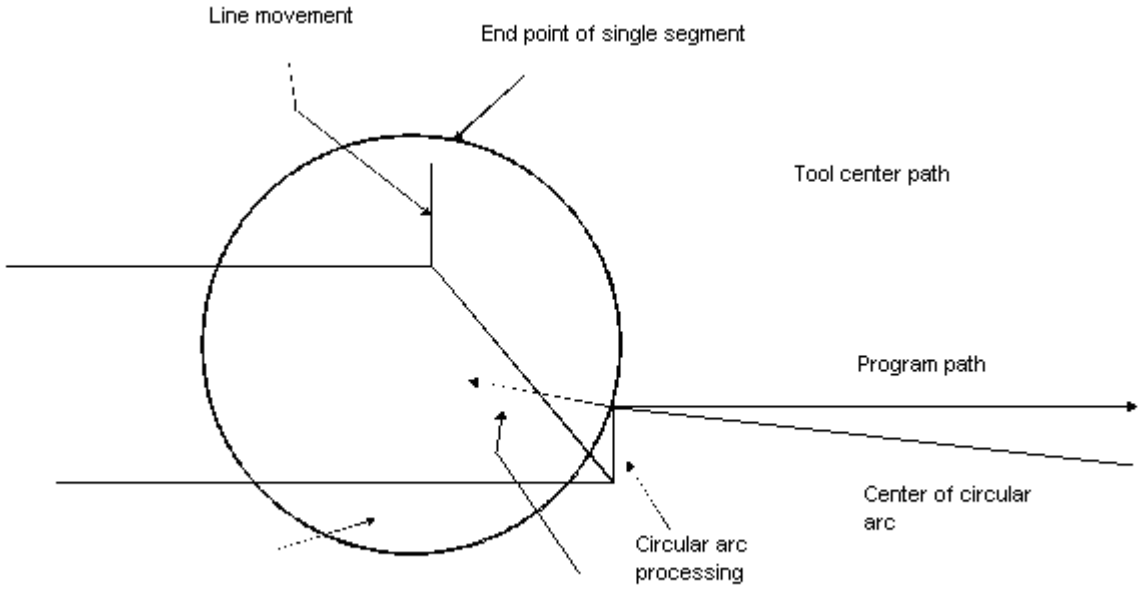
✧ when processing the cut-off trench smaller than radius of tool

Because the tool radius compensation forces the tool center path to move reversely to program path, it will then generate the over cutting.



✧ when processing in case of segment difference smaller than tool radius

If there is segment difference smaller than tool radius, use the circular processing instruction to process the segment difference, and the normal tool center path of compensation will be opposite to the program direction. At this time, neglect the compensation vector and the tool is moved to the second vector in straight line. The single program is stopped here. If it is not in single segment mode, the operation will continue. If the segment difference is a straight line, it will not give an alarm but executing the correct cutting, leaving the parts that are not cut.

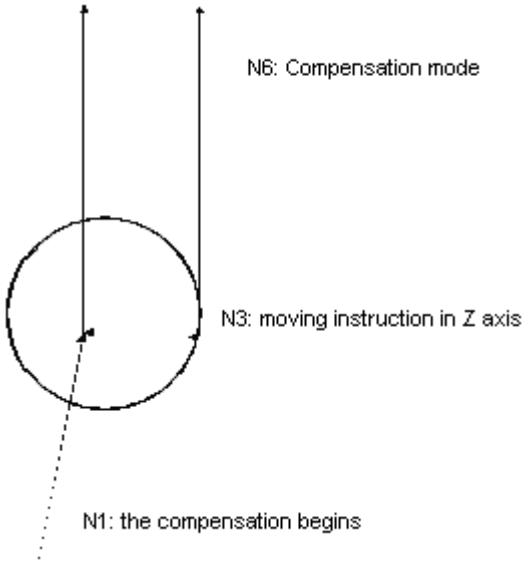


✧ If the initial vector of tool is not neglected, it will generate the over cutting

It is usually at the beginning of processing and when the tool radius compensative is effective, the tool move along Z axis a certain distance away from the workpiece. In this situation, if you want to divide the move along Z axis into quick feed and cutting feed, please follow the procedures as follows:

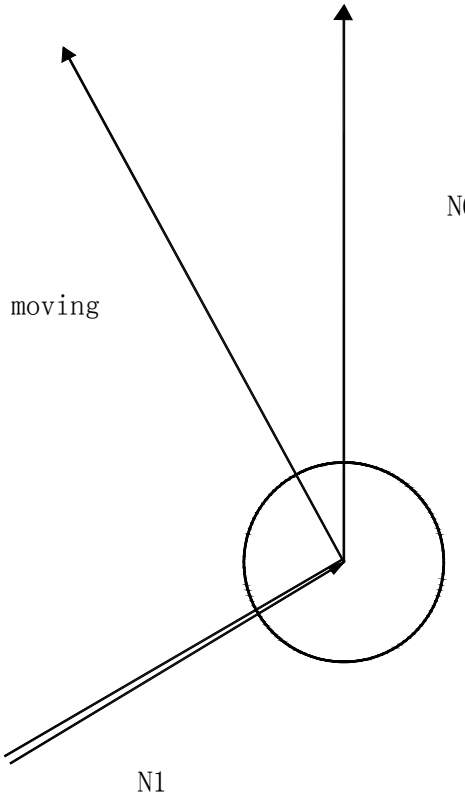
```
N1 G91G00X50000Y50000H01:
N3 G01Z-30000F1:
N6 Y10000F2:
```

When executing N3, N6 is also entered into the buffer area, and take advantage of their relationship to execute the correct compensation



Program segment N3 (Z axis moving instruction) is divided as follows:

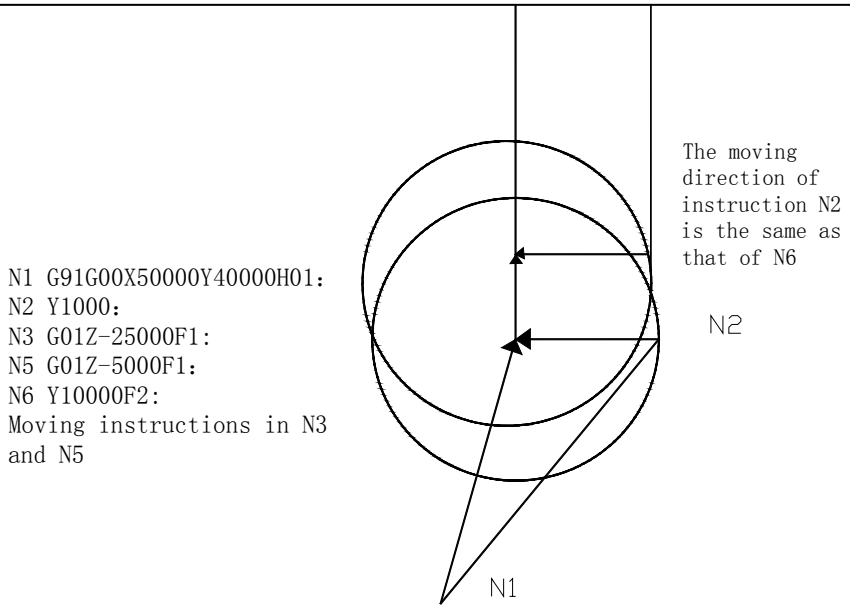
```
N1 G91G00X50000Y50000H01:
N3 Z-250000:
N5 G01Z-5000F1:
N6 Y10000F2:
```



Moving instructions in N3 and N5

If the selected plane does not contain the two moving instruction program segments, N6 cannot enter the buffer area and the tool center path is calculated by N1 as the above picture shows. If the compensation vector is not calculated at the beginning of compensation, it will then generate the over cutting. It is necessary to modify the above example as follows:

When executing N1, program segment N2 and N3 enter the buffer area, and take advantage of the relationship of N1 and N2 to execute the correct compensation.



Length compensation G43 G44 G49

G43 G43
Z_H_ or H_
G44 G44

In accordance with the above instruction, move the end position of Z axis an offset, and preset the difference of supposed tool length in programming and actual used tool length in offset storage; therefore, it is not required to change the program, users just need to change the compensation value to use tool of different length to process the parts.

G43, G44 specify the different offset directions, with H code for specifying the offset number.

- Offset direction

G43: Positive offset
G44: Negative offset

No matter it is absolute value instruction or increment value instruction, add the terminal coordinate value of Z axis moving instruction in program with offset specified by H code while in G43, or deduct offset specified by H code while in G44, and then take the calculated result as the coordinate value of end point.

In case the Z axis movement is omitted, it can be taken as the following conditions. When the offset is a positive value, G43 instruction is to move an offset in positive direction, and G44 is to move an offset in negative direction.

G43
G91 H_
G44

When the offset is a negative value, it is moved in reverse direction.

G43, G44 are mode G codes, and are valid before encountering other G codes of the same group.

- Designation of offset

H code specifies the offset number, and the corresponding offset is added or deducted with moving instruction value of Z axis in the program, forming the new Z axis moving instruction. The offset number can specify H00-H18.

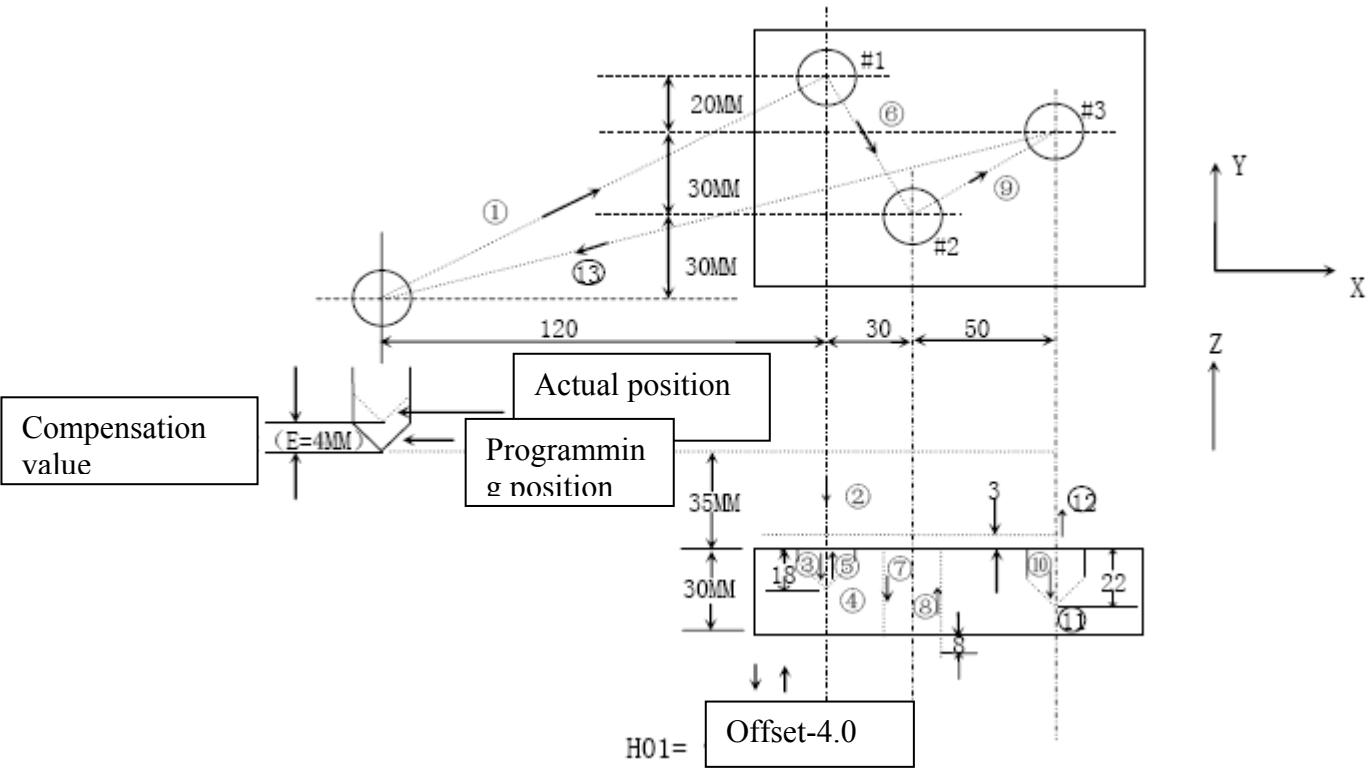
Enter the cuter compensation menu, and preset the corresponding offset number in offset storage.

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

Offset number 00, which means the corresponding offset of H00 is 0. The corresponding offset of H00 cannot be set.

- Cancel cutter length compensation; use G49 or H00 to cancel the cutter compensation. After G49 or H00 instruction, cancel the compensation immediately.
- Example of cutter length compensation

■ Cutter length compensation (Processing #1, #2, and #3 holes)



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 changing offset number to change the offset, it only changes for the new offset value, not adding new offset and old compensation value.

H01……Offset 20.0
H02……Offset 30.0
G90 G43 Z100 0 H01……Z moved to 120..0
G90 G43 Z100 0 H02……Z moved to 130.0

1.2.8 Hole processing cycle (G73~G89)

Fixed cycle of hole processing allows functions that should be done with many program segments in other methods to be done in just one program segment. Table 7.1 lists all fixed cycles of hole processing. Generally, one fixed cycle of hole processing finishes the following 5 operations (see Picture 7.1):

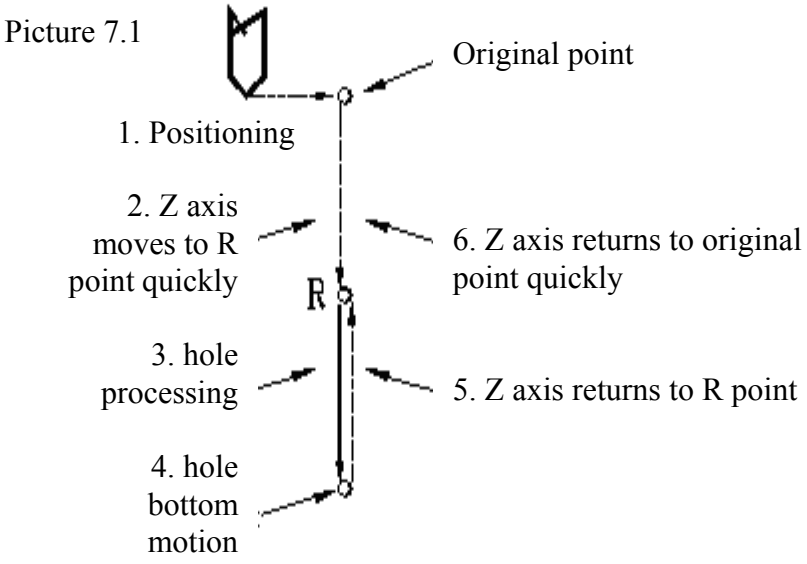
1. X, Y axis quick positioning
2. Z axis positioned to R point quickly
3. Hole processing

4. Down-hole motion
5. Z axis returns to R point
6. Z axis returns to original point quickly

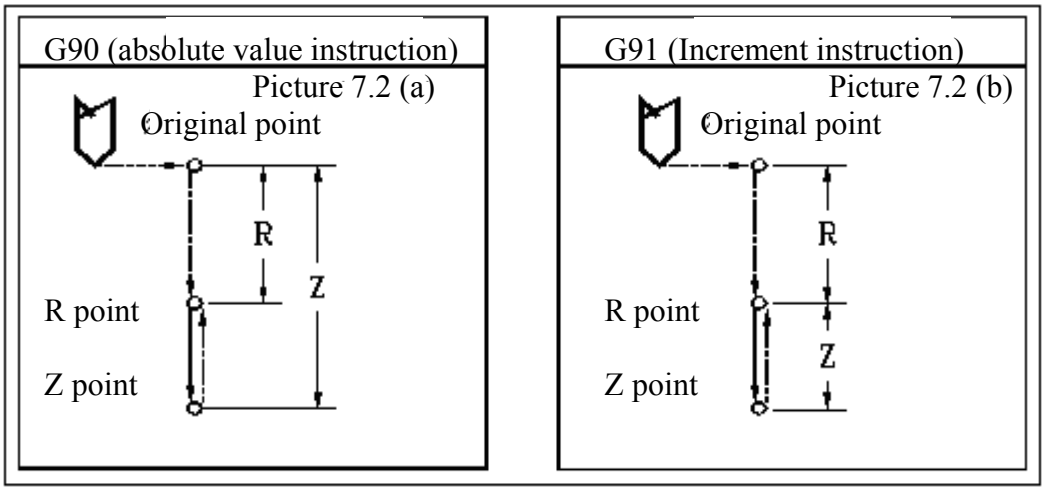
Table 7.1 Fixed cycle of hole processing

G code	Working Motion (Z negative)	Down-hole motion	Return Motion (Z positive)	Application
G73	By times, cutting feed	-	Quick positioning feed	High speed deep-hole drilling
G80	-	-	-	Cancel fixed cycle
G81	Cutting feed	-	Quick positioning feed	Common drilling cycle
G82	Cutting feed	Pause	Quick positioning feed	Drilling or rough boring
G83	By times, cutting feed	-	Quick positioning feed	Deep hole drilling cycle
G84	Cutting feed	Pause-Spindle reverse	Cutting feed	Right screw tapping
G85	Cutting feed	-	Cutting feed	Boring cycle
G86	Cutting feed	Spindle stop	Quick positioning feed	Boring cycle
G88	Cutting feed	Pause-Spindle stop	Manual	Boring cycle
G89	Cutting feed	Pause	Cutting feed	Boring cycle

Picture 7.1

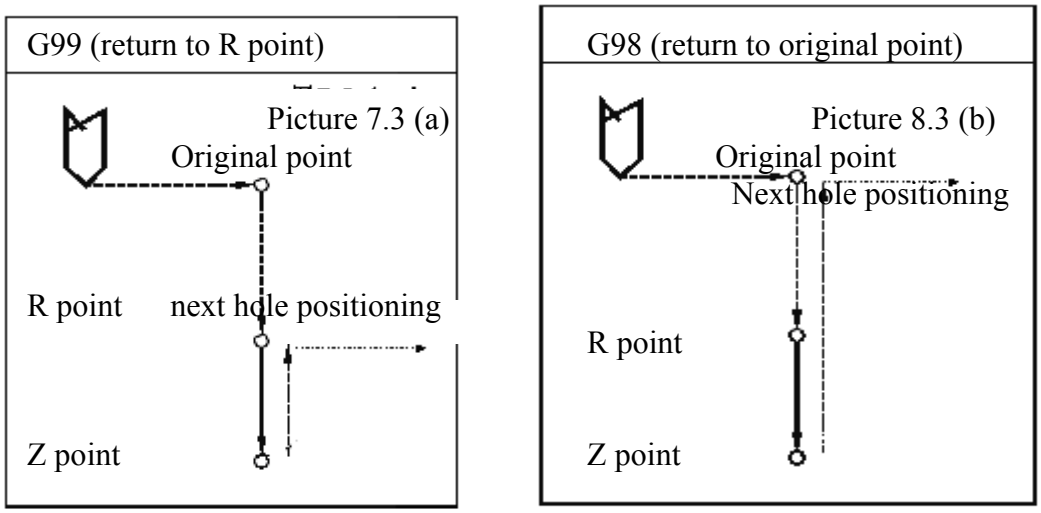


The instructions that influence the execution of instruction for hole processing fixed cycle include G90/G91 and G98/G99 instructions. Picture 7.2(a) and 7.2(b) show the influence of G90/G91 on instruction for hole processing fixed cycle.



G98/G99 determines whether the fixed cycle returns to R point or the original point after the hole processing. In G98 mode, Z axis returns to the original point after the hole processing, while in G99 mode, it returns to R point.

Generally, if the hole to be processed is on a flat plane, we can use G99 instruction, because in G99 mode, returning back to R point will perform the positioning of next hole. In general programming, R point is very close to the surface of work piece, which can shorter the time for processing the part. However, if the surface of work piece is higher than boss or bar of hole being processed, it is possible that the tool and work piece may collide if G99 is used. Therefore, G98 should be used to ensure that Z axis returns to original point and starts positioning the next hole. This way may be safer. See Picture 7.3 (a) and 7.3 (b).



Hole processing parameters are given after G73/G74/G76/G81~G89, and the format is:

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

G×× : Hole processing methods

X___ Y___ Z___ : position parameters of hole being processed

R___ Q___ P___ F___ : Processing parameters of hole

K___ : repeat time

Processing method G	See table 7.1

Location parameter X, Y	Specify the location of hole in way of increment value or absolute value, the track and speed of tool to hole being processed is the same as those of G00
Location parameter Z	Specify the location of hole bottom along the direction of Z axis in way of absolute value, and the distance from R point to hole bottom in way of increment value
Hole processing parameter R	Specify the location of R point along the direction of Z axis in way of absolute value, and the distance from original point to R point in way of increment value
Hole processing parameter Q	To specify the feed amount of deep hole drilling cycle G73 and G83, and the offset of fine boring cycle G76 and reverse boring cycle G87 (it is increment value instruction no matter in G90 or G91 mode)
Hole processing parameter P	Used in fixed cycle that has pause action in hole bottom operation to specify the pause time, unit in second
Hole processing parameter F	To specify the cutting feed rate of fixed cycle; in fixed cycle, the motion from original point to R point and from R point to original point is run at quick feed, the motion from R point to Z point is run at cutting feed speed specified by F, but the motion from Z point to R point may be run at rate specified by F or the quick feed rate in accordance with the fixed cycle
Repeat time K	Specify the repeat time of fixed cycle at the current positioning point. If K is not specified, NC will consider k=1; if K=0, the fixed cycle will not be performed at the current point.

Because the hole processing way specified by G×× is in mode way, the hole processing mode will continue if you do not change the current hole processing mode or cancel the fixed cycle. G instructing that uses G80 or 01 can cancel the fixed cycle. Hole processing parameters are also in mode way, it does not change before being changed or the fixed cycle is cancelled, even when the hole processing mode is changed. We can specify or change any of the hole processing parameters when specifying a fixed cycle or in any time when executing the fixed cycle. Repeat time K is not a mode value, it is only given when repeat is needed. Feed rate F is a mode value, which could be retained even when the fixed cycle is cancelled. If NC system is reset when executing the fixed cycle, the mode and parameters of processing hole, as well as the repeat time K would be cancelled.

The following examples would make you understand the above contents better:

S/N	Content of Program	Note
1	S____ M03	Specify the rotating speed and instruct the main axis to rotate clockwise
2	G81X__Y__Z__R__F__K__	Go to appointed point of X and Y quickly, and process in hole processing mode specified by G81 with processing parameters specified by Z, R, F for K times. At the beginning of executing fixed cycle, Z, R, and F are the necessary hole processing parameters.
3	Y__	X axis does not move, Y axis goes to appointed point quickly for processing the hole, the hole processing parameters and processing mode retain the mode value in 2. The K value in 2 does not work.
4	G82X__P__K__	Hole processing mode is changed. Hole processing parameter Z, R, and F remain the mode value. Give the value of hole processing parameter P and specify the repeat time K.
5	G80X__Y__	Fixed cycle is cancelled, and all hole processing parameters except F are cancelled.
6	G85X__Y__Z__R__P__	For fixed cycle is cancelled when executing 5, all necessary processing parameters except F should be specified again, even they have no any change comparing with the original value.
7	X__Z__	X axis is located to instruction point for processing the hole, and the hole processing parameter Z is changed in this program segment.
8	G89X__Y__	Locate to XY instruction point to process the hole, and the hole processing mode is changed to G98. R and P are specified by 6, while Z is specified by 7.
9	G01X__Y__	Fixed cycle mode is cancelled, all hole processing parameters except F are cancelled.

In the following diagrams, we use the following modes to show the feed of each segment:

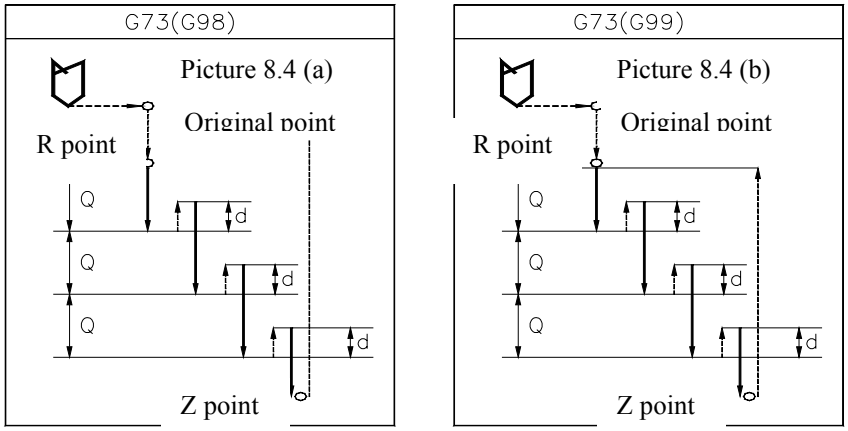
Move at quick feeding rate ·····→

Move at cutting feeding rate ——→

Manual feed - - - - - - - - - - →

➢ G73 (High-speed deep hole drilling cycle)

Format: G73 X_ Y_ Z_ R_ Q_ F_

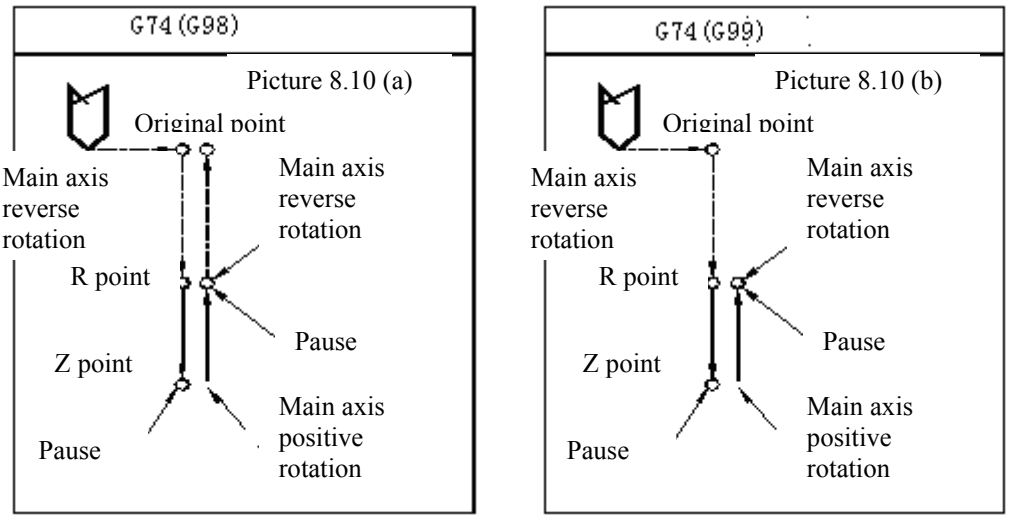


In high-speed deep hole drilling cycle, the feed from R point to Z point is done by segment. After each cutting feed, Z axis will be uplifted a certain distance before performing the cutting feed of next segment. The uplift distance of Z axis is d, which is specified by 531# parameter. The depth of each feed is specified by hole processing parameter Q. This fixed cycle is mainly used in processing the hole with small Calibre-Depth Ratio (such asΦ5, 70 in depth). After the cutting feed of each segment, the action of Z axis uplifting is to cut the scraps.

➢ G74 (Reverse thread tapping cycle)

Format: G74 X_ Y_ Z_ R_ F_(D_)

X_Y_: thread position
Z_: depth of thread
R_: original point of feeding and cutter withdrawal
F_(D_): Calculate the feed rate according to the pitch or give out the pitch distance directly by D_



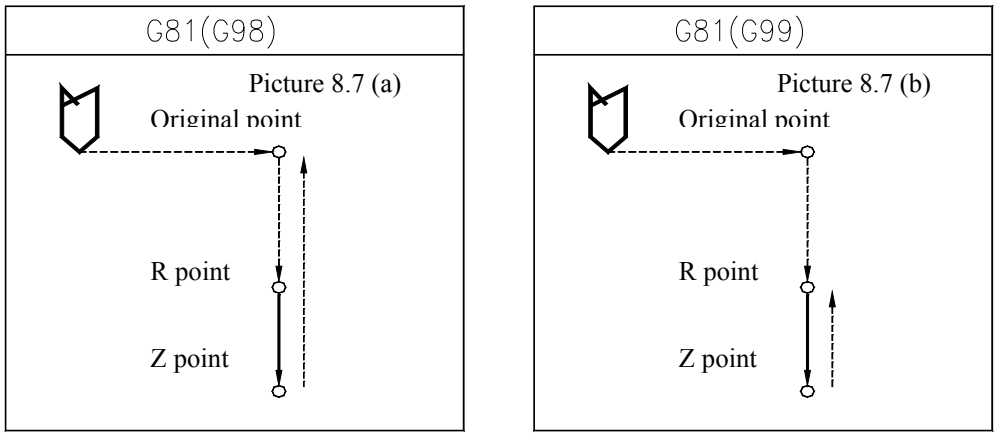
Note: In G74 and G84 cycles, the function of feed rate override and feed hold will be neglected, which means the feed rate will be kept at 100%. It should not be stopped in midway before a fixed cycle is executed. Before the cycle, it is required to instruct to rotate in main axis tapping direction.

G80 (Cancel fixed cycle)

Once G80 instruction is executed, fixed cycle (G73, G74, G76, G81~G89) will be cancelled, parameters of R point and Z point, as well as all hole processing parameters except F will be cancelled. In addition, G code in 01 group will also have the same function.

G81 (Drilling cycle)

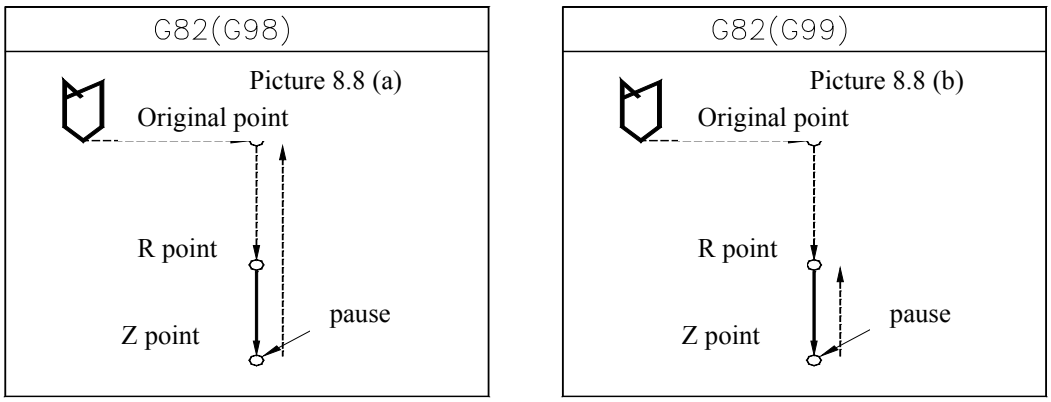
Format: G81 X_ Y_ Z_ R_ F_



G81 is the most simple fixed cycle, it is executed as follows: X, Y locating, Z axis moved to R point quickly and fed to Z point at F speed, and then returned to original point (G98) or R point (G99) quickly, without hole bottom action.

G82 (Drilling cycle, rough boring cycle)

Format: G82 X_ Y_ Z_ R_ P_F_

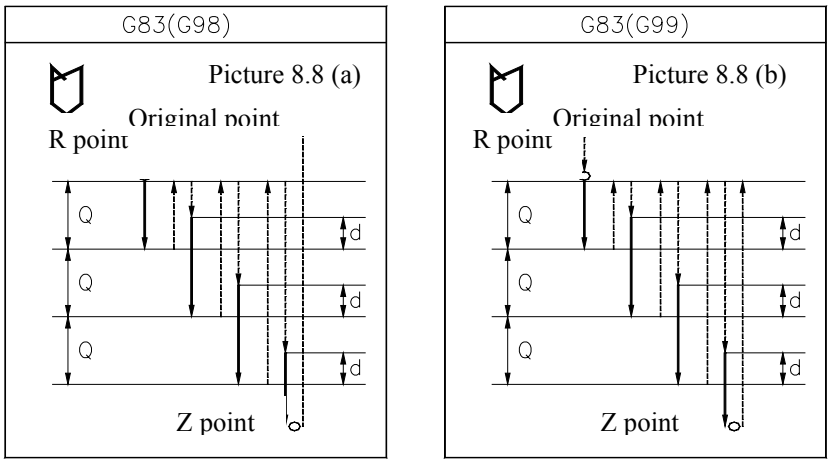


G82 has a pause action at the hole bottom, and apart from this, it is the same as G81. The pause at the bottom of hole can improve the precision of hole depth.

G83 (Deep hole drilling cycle)

Format: G83 X_ Y_ Z_ R_ Q_ F_

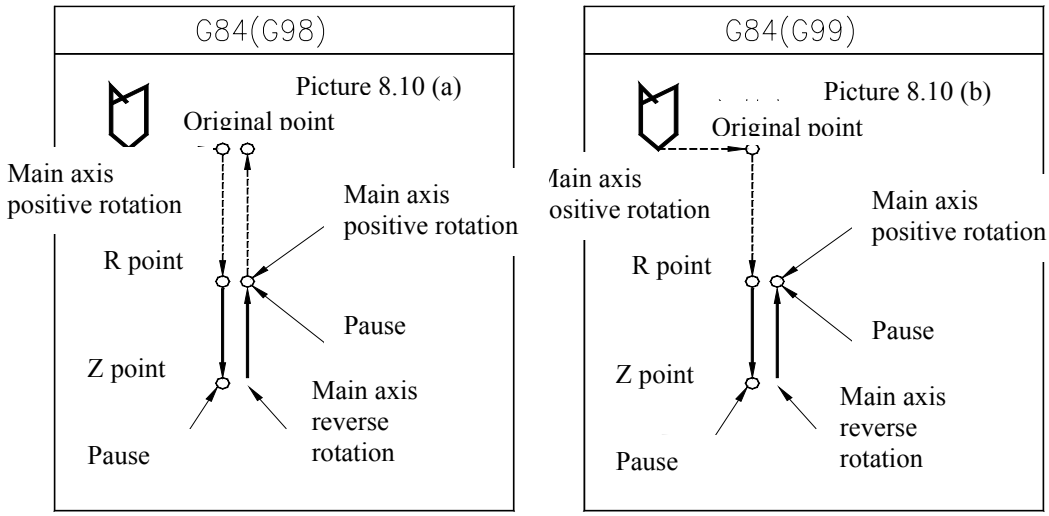
Similar to G73 instruction, the feeding from R point to Z point in G83 instruction is also done by segment. The difference is that after the feed of each segment, Z axis is returned to R point, and then moved at quick feeding rate to d above the feeding origin of the next segment and start the feeding motion of next segment. The feeding distance of each segment is specified by hole processing parameter Q, which is always the positive value. The value of d is given by 532# machine parameters. See Picture 8.9 :



➢ G84 (Tapping cycle)

Format: G84 X_ Y_ Z_ R_ F_(D_)

X_Y_: thread position
Z_: depth of thread
R_: original point of feeding and cutter withdrawal
F_(D_): Calculate the feed rate according to the pitch or give out the pitch distance directly by D_

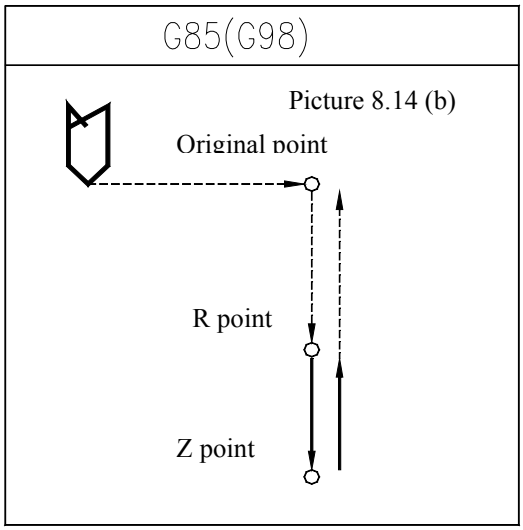


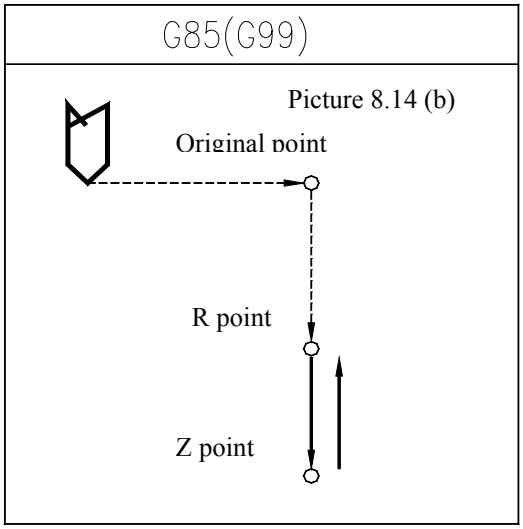
Note: In G74 and G84 cycles, the function of feed rate override and feed hold will be neglected, which means the feed rate will be kept at 100%. It should not be stopped in midway before a fixed cycle is executed. Before the cycle, it is required to instruct to rotate in main axis tapping direction.

7）G85 (Boring cycle)

Format: G85 X_ Y_ Z_ R_ F_

This fixed cycle is very simple, it is executed as follows: X, Y locating, Z axis moved to R point quickly and fed to Z point at speed specified by F, and then returned to R point at specified speed, or if in G98 mode, returned to R point and then to the original point quickly.

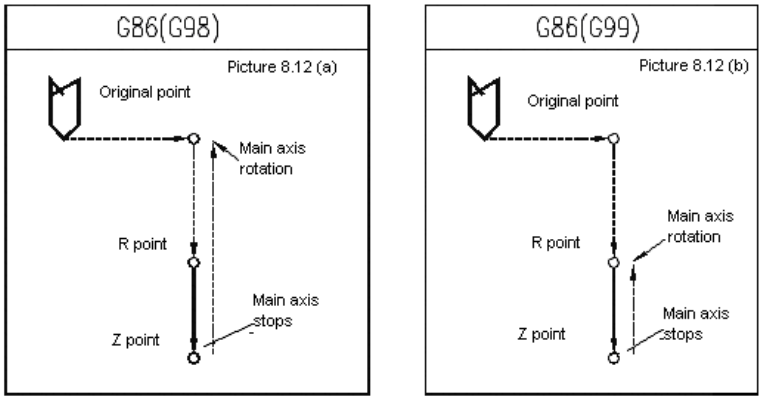




G86 (Boring cycle)

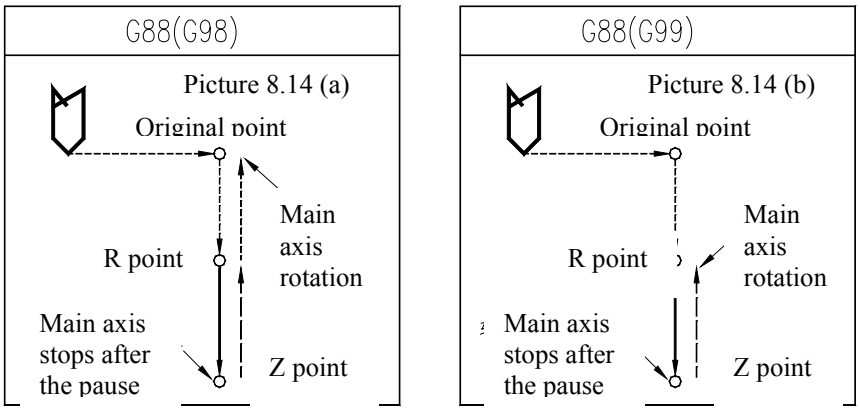
Format: G86 X_ Y_ Z_ R_ F_

This fixed cycle is executed similarly to G81. what is different is that in G86 when the tool is fed to hole bottom, the main axis will stop, and when it returns to R point or original point quickly, the main axis will rotate at the original speed in the same direction.



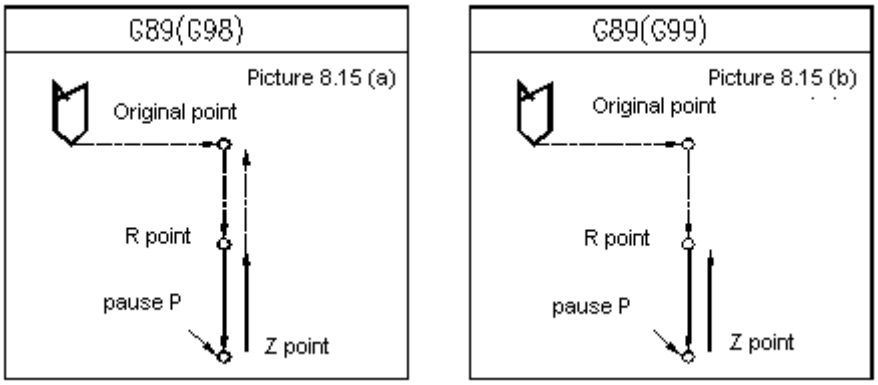
G88 (Boring cycle)

Fixed cycle G88 is provided with manual return function for fixed cycle of drilling (as picture shows):



➢ G89 (Boring cycle)

Pause at hole bottom is added in this fixed cycle basing on G85. See Picture 8.15:



➢ Notices for using fixed cycle of hole processing

a. When programming, it is required to make S and M codes to instruct the main axis to rotate before the fixed cycle instruction.

M03 ; main axis is rotated clockwise
.
.
G□□…… ; Correct
.
.
M05 ; main axis stops
G□□……; incorrect (it is required to have instruction M03 or M04 before this program segment)

b. In mode of fixed cycle, the program segment included X, Y, Z, R will execute the fixed cycle. If a program segment does not include any of the above address, the fixed cycle is not executed in this program segment, address X in G04 is excluded. In addition, address P in G04 could not change the P value in hole processing parameters.

G00 X__ ;
G81 X__ Y__ Z__ R__ F__ K__ ;
; (do not execute the hole processing)
F__; (do not execute the hole processing, F value is upgraded)
M__; (do not execute the hole processing, only execute the auxiliary function)
G04 P__; (do not execute the hole processing, use G04 P__ to change the hole processing data P)

c. Hole processing parameter Q and P should be specified in program segment, in which the fixed cycle is executed; otherwise, the Q and P values of instruction would be invalid.

d. When executing the fixed cycle (such as G76 and G84) that contains main axis control, as the tool starts cutting and feeding, it is possible that the main axis may not have achieved the appointed rotation speed. In this situation, it is required to add G04 pause instruction during the hole processing.

e. As it is described, the G code of 01 group can also be used to cancel the fixed cycle. Therefore, do not write the fixed cycle instruction and G code of 01 group in the same program segment.

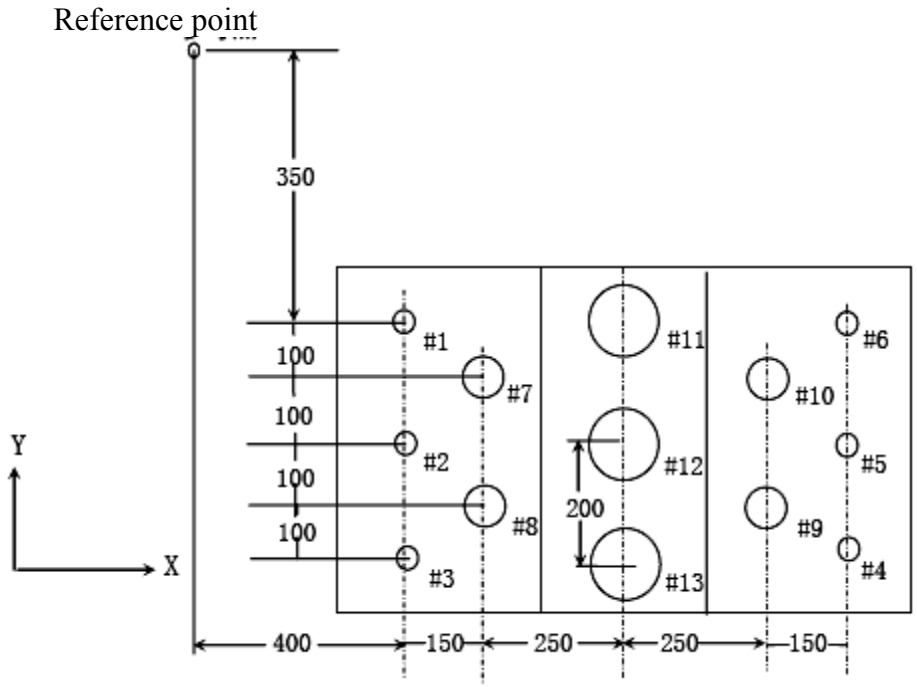
f. If an M code is specified in program segment for executing fixed cycle, M code will be executed at the same time when the fixed cycle is executing the positioning. The signal of completion of M instruction execution will be given out as Z axis returns to R point or the original point. When using K parameter to instruct repeating the execution of fixed cycle, the M code in the same program segment is executed when executing the fixed cycle at the first time.

g. In mode of fixed cycle, tool offset instruction G45~G48 will be neglected (not executed).

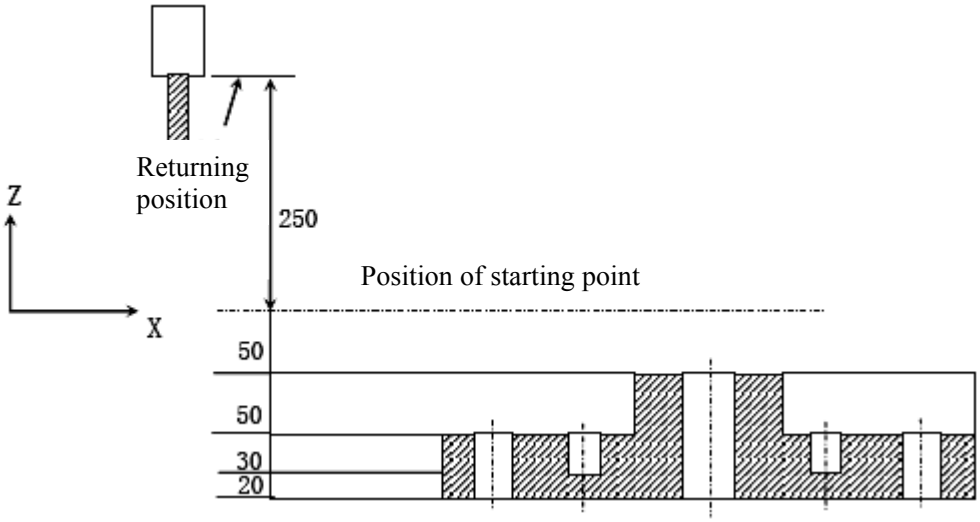
h. When the switch of single program segment is set in upper position, the fixed cycle will stop after executing X, Y positioning, feeding to R point quickly and returning from hole bottom (to R point or original point). This is to say you have to press the cycle activate button for 3 times to complete the hole processing. Within these 3 times of pause, the first two times are in feed holding state, while the last one is in stop state.

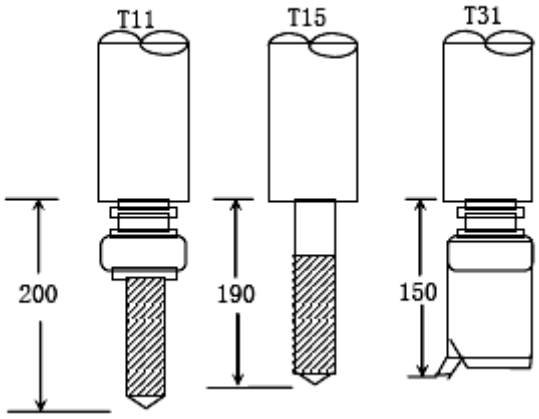
i. When executing G74 and G84 cycles, if you press feed holding button when Z axis is moving from R point to Z point and from Z point to R point, the feed holding indicator will be on immediately, but the machine would not stop and go to holding state until Z axis returns to R point. In addition, in G74 and G84 cycles, feed rate switch is not valid, and the feed rate is fixed at 100%.

➢ Examples of using tool length compensation and fixed cycle



#1~6 Drill ¢10 hole
#7~10 Drill ¢20 hole
#11~13 Bore ¢95 hole (50MM in depth)





The value of offset in number 11 is 200.0, number 15 is 190.0, and number 31 is 150.0. The program is as follows:

N001 G92 X0 Y0 Z0 ; the coordinate system is set at the reference point.
N002 G90 G00 Z250.0 T11 M6; change the tool.
N003 G43 Z0 H11 ; perform plane tool length compensation at the original point.
N004 S30 M3 ; main axis is started.
N005 G99 G81 X400.0 Y-350.0
Z-153.0 R-97.0 F120.0 ; process #1 hole after the positioning.
N006 Y-550.0 ; process #2 hole after the positioning, and then return to plane of R point.
N007 G98 Y-750.0 ; process #3 hole after the positioning, and then return to plane of original point.
N008 G99 X1200.0 ; process #4 hole after the positioning, and then return to plane of R point.
N009 Y-550.0 ; process #5 hole after the positioning, and then return to plane of R point.
N010 G98 Y-350.0 ; process #6 hole after the positioning, and then return to plane of original point.
N011 G00 X0 Y0 M5 ; return to reference point and the main axis stops.
N012 G49 Z250.0 T15 M6 ; Cancel the tool length compensation, and change the tool.
N013 G43 Z0 H15 ; perform tool length compensation on plane of original point.
N014 S20 M3 ; main axis starts.
N015 G99 G82 X550.0 Y-450.0 ;
Z-130.0 R-97.0 P30 F70; process #7 hole after the positioning, and then return to plane of R point.
N016 G98 Y-650.0 ; process #8 hole after the positioning, and then return to plane of original point.
N017 G99 X1050.0 ; process #9 hole after the positioning, and then return to plane of R point.
N018 G98 Y-450.0 ; process #10 hole after the positioning, and then return to plane of original point.
N019 G00 X0 Y0 M5 ; return to reference point and the main axis stops.
N020 G49 Z250.0 T31 M6 ; Cancel the tool length compensation, and change the tool.
N021 G43 Z0 H31 ; perform tool length compensation on plane of original point.
N022 S10 M3 ; main axis starts.
N023 G85 G99 X800.0 Y-350.0 ;
Z-153.0 R47.0 F50 ; process #11 hole after the positioning, and then return to plane of R point.
N024 G91 Y-200.0 ; process #12, #13 hole after the positioning, and then return to plane of R point.
Y-200.0 ;
N025 G00 G90 X0 Y0 M5 ; return to reference point and the main axis stops.
N026 G49 Z0 ; Cancel the tool length compensation.
N027 M30 ; % Program stops.

1.3 Auxiliary function (M, S, T)

The machine tool uses S code to program the rotation speed of main axis, and T code to program the tool selection. Other programmable auxiliary functions are achieved by M code.

1.3.1 M code

List of M code:

M code	Function
M01	Stop program
M03	Rotate main axis clockwise
M04	Rotate main axis anti-clockwise
M05	Stop main axis
M06	Change tool instruction
M08	Open the cooling
M09	Close the cooling
M32	Enable the lubrication
M33	Disable the lubrication
M30	Program ended and returned to the beginning
M98	Call for sub-program
M99	Sub-program ended and returned/repeat the operation
M56	Expansion M code output control (refer to the definition of output interface connection)
M57	Expansion M code output control (refer to the definition of output interface connection)
M58	Expansion M code output control (refer to the definition of output interface connection)
M59	Expansion M code output control (refer to the definition of output interface connection)
M10	Expansion M code output control (refer to the definition of output interface connection)
M11	Expansion M code output control (refer to the definition of output interface connection)
M20	Expansion M code output control (refer to the definition of output interface connection)
M21	Expansion M code output control (refer to the definition of output interface connection)
M12	Expansion M code output control (refer to the definition of output interface connection)
M13	Expansion M code output control (refer to the definition of output interface connection)
M14	Expansion M code output control (refer to the definition of output interface connection)
M15	Expansion M code output control (refer to the definition of output interface connection)
M16	Expansion M code output control (refer to the definition of output interface connection)
M17	Expansion M code output control (refer to the definition of output interface connection)
M18	Expansion M code output control (refer to the definition of output interface connection)
M19	Expansion M code output control (refer to the definition of output interface connection)
M40	Expansion M code output control (refer to the definition of output interface connection)
M41	Expansion M code output control (refer to the definition of output interface connection)
M42	Expansion M code output control (refer to the definition of output interface connection)
M43	Expansion M code output control (refer to the definition of output interface connection)
M44	Expansion M code output control (refer to the definition of output interface connection)
M45	Expansion M code output control (refer to the definition of output interface connection)
M46	Expansion M code output control (refer to the definition of output interface connection)
M47	Expansion M code output control (refer to the definition of output interface connection)
M48	Expansion M code output control (refer to the definition of output interface connection)
M49	Expansion M code output control (refer to the definition of output interface connection)
M50	Expansion M code output control (refer to the definition of output interface connection)
M51	Expansion M code output control (refer to the definition of output interface connection)
M66	Expansion M code output control (refer to the definition of output interface connection)
M67	Expansion M code output control (refer to the definition of output interface connection)

M64	Expansion M code output control (refer to the definition of output interface connection)
M65	Expansion M code output control (refer to the definition of output interface connection)
M62	Expansion M code output control (refer to the definition of output interface connection)
M63	Expansion M code output control (refer to the definition of output interface connection)
M60	Expansion M code output control (refer to the definition of output interface connection)
M61	Expansion M code output control (refer to the definition of output interface connection)
M88 Pn Lm	Test whether the level signal of waiting IO (IN n) is m (high, low)
M89 Pn Lm Qt	Output OUT n, level is m, output in t milliseconds delay

In machine tool, the function of M code can be classified as two categories: one is to control the execution of program, and the other is used for IO operation to control the execution of auxiliary devices such as main axis and cooling system.

M code for controlling program

The M codes for controlling program include M00, M30, M98, and M99. Their functions are explained as follows:

M00……program stops. NC stops the execution of program when executing to M00. After the reset, you can press start button to continue executing the program.

M30……program ends and returns to the beginning of program.

M98……Call the subprogram.

M99……Subprogram ends and returns to the main program.

Other M codes

M03……main axis is rotated clockwise. Use this instruction to make the main axis to rotate anti-clockwise (CCW) at the current appointed rotation speed.

M04……main axis reversal; use this instruction to make the main axis to rotate clockwise (CW) at the current appointed rotation speed.

M05……main axis stops.

M06……start changing tool; M06 T02 is to change the No. 2 tool

M08……Cooling open

M09……Cooling closed

M32……lubrication opened

M33……lubrication closed

M88……specify input IO to judge the level; if it's the same, the execution will continue; or otherwise stop and wait. If level signal is not specified, it's default as low level signal. For example, M88 P0 L1 waiting IN0 is high level; otherwise, wait all the time.

M89……specify output IO to judge the appointed level; if level signal is not specified, it's default as low level. If Q value is specified, this operation will output IO signal with Q milliseconds delay. For example: M89 P5 L0, specifies OUT5 to output low level.

Note:

- **When the moving instruction and M are in the same program segment, M instruction is executed first.**
- **If the program has many M codes in the current line, only one code is valid, which is the one defined at the very end.**

1.3.2 S code

The rotation speed instruction of main axis is given by S code, which is mode state, meaning once the rotation speed is specified, it will be effective all the time until the mode value of the other S code is changed.

The maximum value of S instruction is restricted by the maximum main axis rotation speed set by parameter P5.020.

S instruction has three output modes, which are influenced by parameter P2.049 (main axis specifies the interface axis number), P1.061 (frequency-changing control mode), as follows:

P2.049 set as non 0:

It means the current main axis is at AB-phase pulse control mode. At this time, the S value is set by main axis coder to determine the pulse frequency.

P2.049set as 0, P2.061set as 1:

Frequency-changing control mode, the communication with frequency-changing uses 4 IO (OUT23~OUT20) shifts. Four shifts form 16 codes, which means the S instruction value is S00~S15;

P2.049 set as 0, P2.061 set as 0:

Frequency-changing analog control mode, the ratio of S value and the maximum rotation speed set by parameter P5.020 plus 10V, and change over to get the analog voltage value; S instruction can output analog value after having specified to execute M03 or M04;

1.3.3 T code

Tool library of machine tool uses arbitrary way of tool selection. The two digit T code (T××) specifies the tool number, and it is not necessary to know which tool set the tool is in. The range of address T can be any integer among 1~99.

Warning:

Tool table should be set correctly. If it is inconsistent with the actual condition, it will damage the machine tool and bring unexpected results.

1.4 Macro

1.4.1 Variable instruction

The address values in program are not described in fixed values but in variable. When running the program, variable is quoted with the purpose of increasing the universal property of program. This is called the variable instruction.

Instruction format:
#△△△=○○○○○○○○○ or #△△△= [Expression]

Details:
(1) Expression of variable:

(a) # m	M=value formed by 0~9	#100
(b) # [f]……	f represents the following meanings	
	Numerical value m	123
	Variable	#543
	Expression	#110+#119
	-(symbol) expression	-#120
	Function expression	SIN [#110]

- Standard operational signs include +, -, ×, /
- When function expression is neglected, the function cannot be executed.
- The sign of variable should not be negative, for example,#-100 is illegal.
- The following are the wrong expressions of variable:

Wrong		Correct
#6/2	→	# [6/2]
#-[#1]	→	# [-#1]
#——5	→	# [- [-5]]

(2) Kind of variable

Kind	Variable address	Description of function
Global variable	#100~#199 #500~#999	➢ Can be called by main program and sub program ➢ #100~#199 is non-maintained variables and will be cleared to 0 once the system is electrified again. ➢ #500~#999 is maintained variable, and the value will be retained after the system power down.
Local variable	#1~#32	can be called within the same program
System variable	Not available	

(3) quotation of variable

(a) Except O, N and / (slash)

(b) specified as variable directly
G01X#1Y#100

(c) take complementary number for the variable directly
G01X-#2

(d) variable defines the variable
#3=-#105; take the complementary number of #105 value and assign it directly to #3
#4=1000; assign 1000 to #4 directly

(e) use expression to assign the value
#1=#3+#2-100; the value of #1 is the result of #3+#2-100
X[#1+#3+1000]; the value of X is the calculation result of expression [#1+#3+1000]

- Assigning value by function and by expression should not be in the same line, they should be written separately.

Wrong		Correct
X#1=#3+100	→	#1=#3+100 X#1

- For [ ] (bracket) calculation, as many as 5 layers can be embedded.
#543=-[[[[[#120]/2+15.]*3-#100]/ #520+#125+#128]* #130+#132
- The value of variable should be within 0~±9999999 (7-digit effective figure). If it exceeds the maximum value, the error of calculation will be big.

1.4.2 Macro program call

using calling function of macro

Function and purpose

The call of macro program is the same as that of sub program. When the macro program is calling, it can transfer some variable values to sub program. This is different from the call of M98 sub-program.

The following G codes are instructions for calling macro programs:

G code	Function
G65	simple call of macro program
G66	macro program calling mode A (Moving instruction call)
G661	macro program calling mode B (each segment call)
G67	cancel the macro program calling mode

Details:

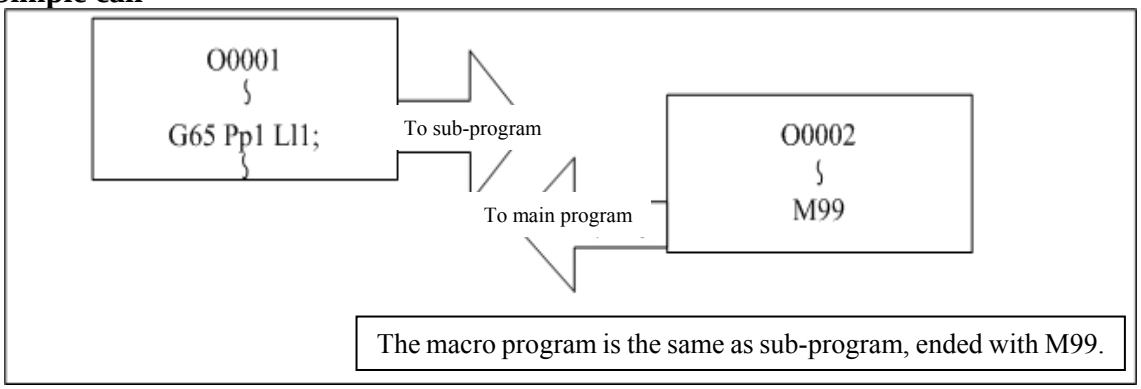
1) Specified by G66 (or G661) instruction, and before G67 (Cancel) instruction, the macro program specified after the execution of single segment with move instruction (or each single segment) will be called.
2) In the same program, G66 (or G661), G67 instructions should be used correspondingly in pair.

Calling command of macro program

Function and purpose

Calling command of macro program includes simple call, and call mode (A&B) of single segment fixed call.

1. Simple call



Format specification:

G65 P_ L_ < argument >;
P_ : No. of sub program
L_ : Repeat time

The function of < argument > in G65 is one of the methods that the main program uses address to transfer parameters to sub program. This method uses local variable to transfer, and the description of argument is as follows:

Argument format:

Format description:

A_B_C_...X_Y_Z_

Details:

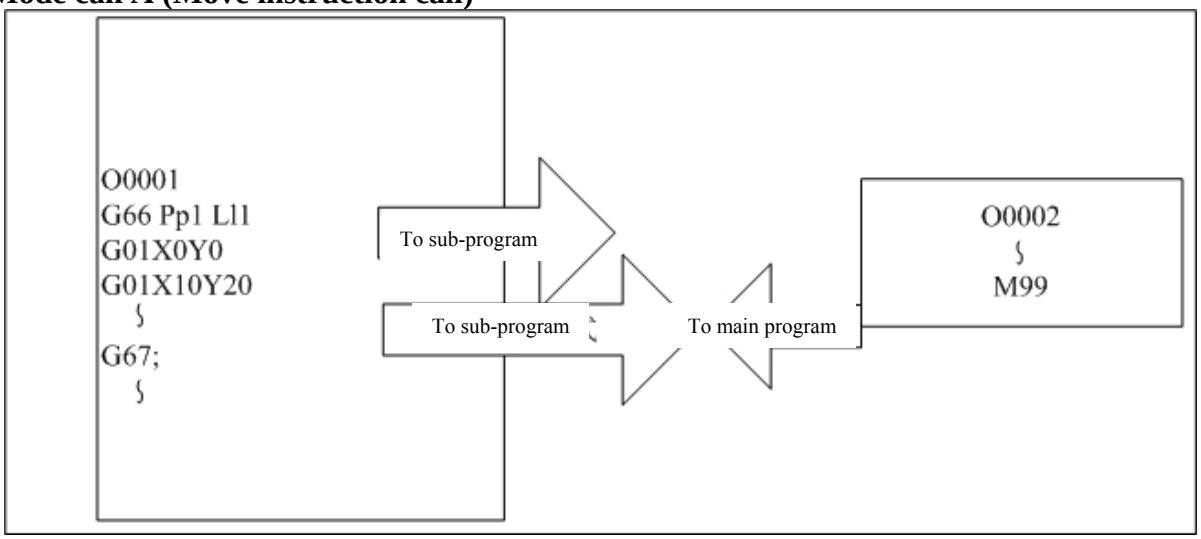
1) Except G,L,N,O,P, all addresses can be specified as argument.
2) Addresses that are not required to be transferred can be neglected.
3) The address information occurs in G65 instruction is considered as the argument of G65. For example: G65P0002N100G01G90X100.Y200.F400R1000,G01 instructions are not executed, and all addresses are considered as the argument of G65.
4) The comparison of addresses specified by argument and the local variable number is as follows:

Address	Variable No.	G65, G66, G661
A	#1	○
B	#2	○
C	#3	○
D	#7	○
E	#8	○
F	#9	○
G	×	×
H	#11	○
I	#4	○
J	#5	○
K	#6	○
L	×	×
M	#13	○
N	×	×
O	×	×
P	×	×
Q	#17	○
R	#18	○
S	#19	○
T	#20	○
U	#21	○
V	#22	○
W	#23	○
X	#24	○
Y	#25	○
Z	#26	○

○: Available

×: Not available

2. Mode call A (Move instruction call)



Between G66 and G67, when the single segment with move call is executed, the appointed macro sub-program will be called and executed. The time of execution is the time specified by L.

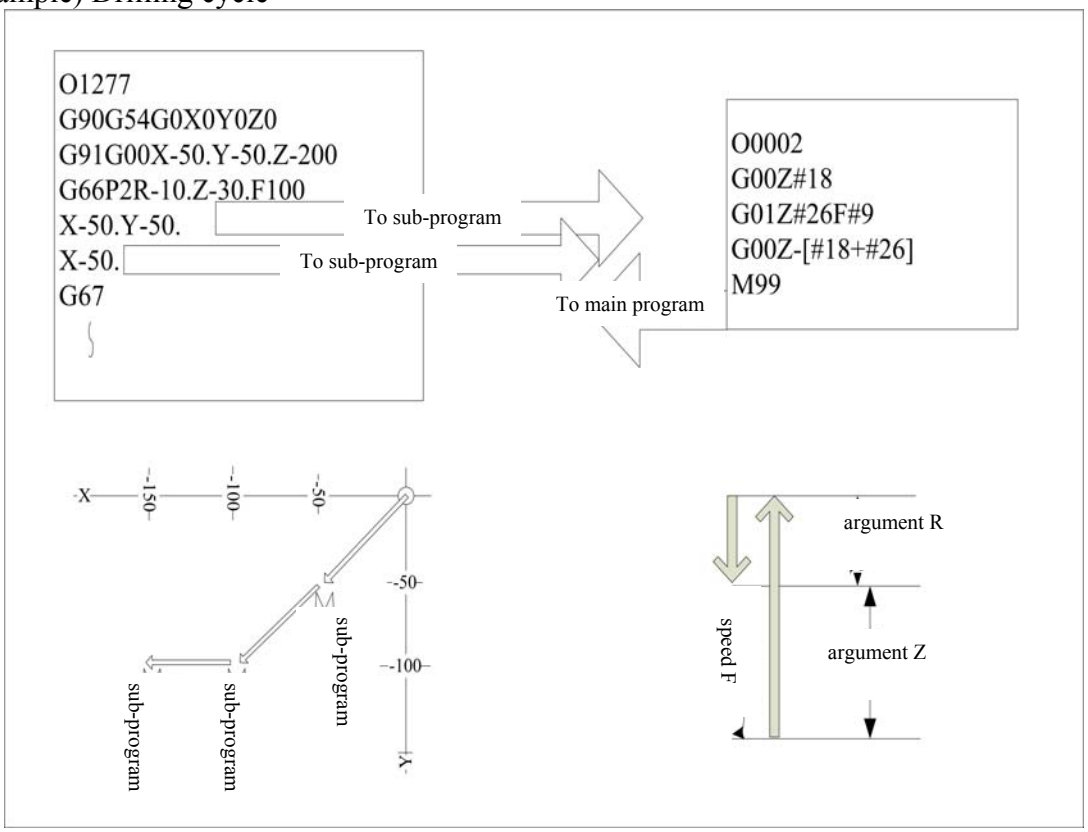
Format description:

G66 P_ L_< argument >;	
P_	: No. of sub program
L_	: Repeat time

Details:

1) Specified by G66 instruction, and before G67 (Cancel) instruction, the macro sub-program specified by G66 will be called automatically after the execution of program segment with move instruction.
2) In the same program, G66 and G67 instructions should be specified in pair. If G66 instruction is not executed first before G67, the system will give a warning.
3) The address information occurs in G66 instruction segment is considered as the argument of G65. For example: G66P0002N100G01G90X100.Y200.F400R1000,G01 instructions are not executed, and all addresses are considered as the argument of G66.

(Example) Drilling cycle



- G66 instruction executes the sub-program for the first time, and the later move instruction will call and execute the sub-program automatically.
- Once the G67 instruction takes effect, the sub-program will not be executed any more.

3. Mode call B (Call in every single segment)

Between G661 and G67 instructions, each single segment of instruction will call the appointed macro sub-program unconditionally.
Format description:

G661 P_ L_< argument >;	
P_	: No. of sub program
L_	: Repeat time

Details:

1) In G661 mode, except O, N and G codes of single segment of each instruction, all are used as argument.
2) The address information occurs in G661 instruction segment is considered as the argument of G661. For example:
G661P0002N100G01G90X100.Y200.F400R1000,G01 instructions are not executed, and all addresses are considered as the argument of G661.

1.4.3 Variable

Function and purpose

Variable is a useful function for macro, which is divided into four kinds, local variable, global non-maintained variable, global maintained variable, and system variable. These variables make it convenient and universal when compiling the macro.

Use of multivariable

- Macro calls the variable, and the variable number can be multiple or specified by expression. As the following examples show:

#1＝10 #10＝20 #20＝30 #5＝#[#[#1]];	For #1=10,#[#[#1]]=#[#10] For #10=20,#[#10]=#20 Hence #5=#20 or #5=30

#10=5 #10=20 #20=30 #5=1000 #[#[#1]]=#5	For #1=10,#[#[#1]]=#[#10] For #10=20,#[#10]=#20 Hence #20=#5 or #20=1000

- Examples of specifying the multivariable:

#10=5 #5=100 #6=##10	##10 is the same as #[#10]

- use expression to replace as number:

#10=5 #[#10+1]=1000 #[#10-1]=-1000 #[10*3]=100 #[#10/2]=-100	 #6=1000 #4=-1000 #15=100 #2=-100

Undefined variable

Once the system is started, the undefined variable is default as null. The local variables that argument does not specify are also considered as null variable. The #0 of system is also the null variable. The null variable is considered as 0 in calculation, and #0 is generally not allowed to act as the left value of expression to join in the calculation. However, if the programmers make a mistake, the program will not report the error, but it will not have any effect.

- Calculation

#1=#0; ……#1=<Null> #2=#0+1; ……#2=1 #3=1+#0; ……#3=1 #4=#0*10;……#4=0 #5=#0+#0;……#5=0	It should be noted that <Null> in calculation is equal to 0. <Null> + <Null>=0; <Null> + <Fixed number> = < Fixed number > < Fixed number > + <Null> = < Fixed number >

- Quotation of variable

#1=<Null>
G0X#1Y1000; ……equals toG0X0Y1000
G0X#1+10Y1000;……equals toG0X10Y1000

- Conditional expression

Null variable is equal to 0 to carry out the logical conditional calculation when judging the condition.

Kinds of variable

Common variable

Every address can use the common variable. The common variable has 600 groups, in which #100 ~ #199 represent the non-maintained common variable group in case of power down, and #500~ #999 represent the maintained common variable group in case of power down.

Local variable (#1~ #32)

When calling the sub program, the local variable can be defined by <argument>, and can only be used in program. The local variable of program of each macro is independent, and therefore can be repeated (maximum for 4 times).

G65 Pp1 Ll1 < argument >; p1 : No. of sub program l1 : Repeat time

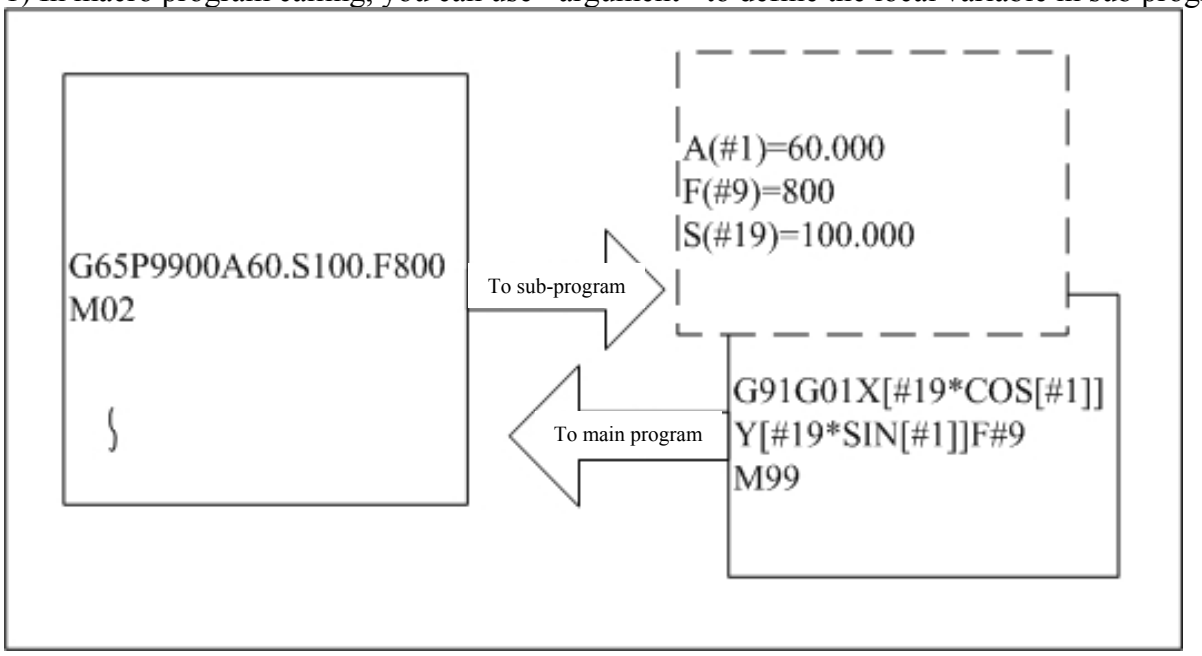
< argument > is Aa1 Bb1 Cc1…Zz1 etc.

Comparison of addresses specified by <argument> and the local variable in program is as follows:

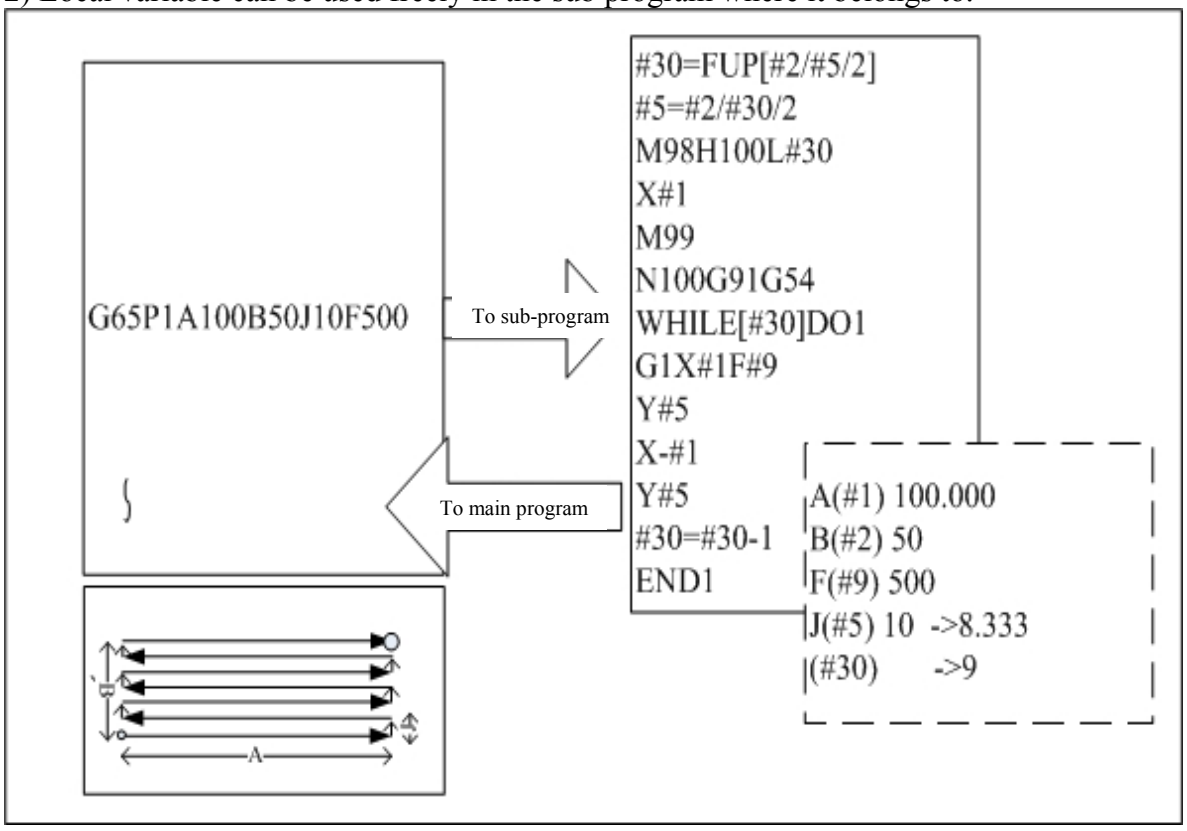
Address	Variable No.	Sub program	Address	Variable No.	Sub program
A	#1	○	N	×	×
B	#2	○	O	×	×
C	#3	○	P	×	×
D	#7	○	Q	#17	○
E	#8	○	R	#18	○
F	#9	○	S	#19	○
G	×	×	T	#20	○
H	#11	○	U	#21	○
I	#4	○	V	#22	○
J	#5	○	W	#23	○
K	#6	○	X	#24	○
L	×	×	Y	#25	○
M	#13	○	Z	#26	○

Argument address represented by "×" cannot be used.
Argument address represented by "○" can be used.

1) In macro program calling, you can use <argument> to define the local variable in sub program.

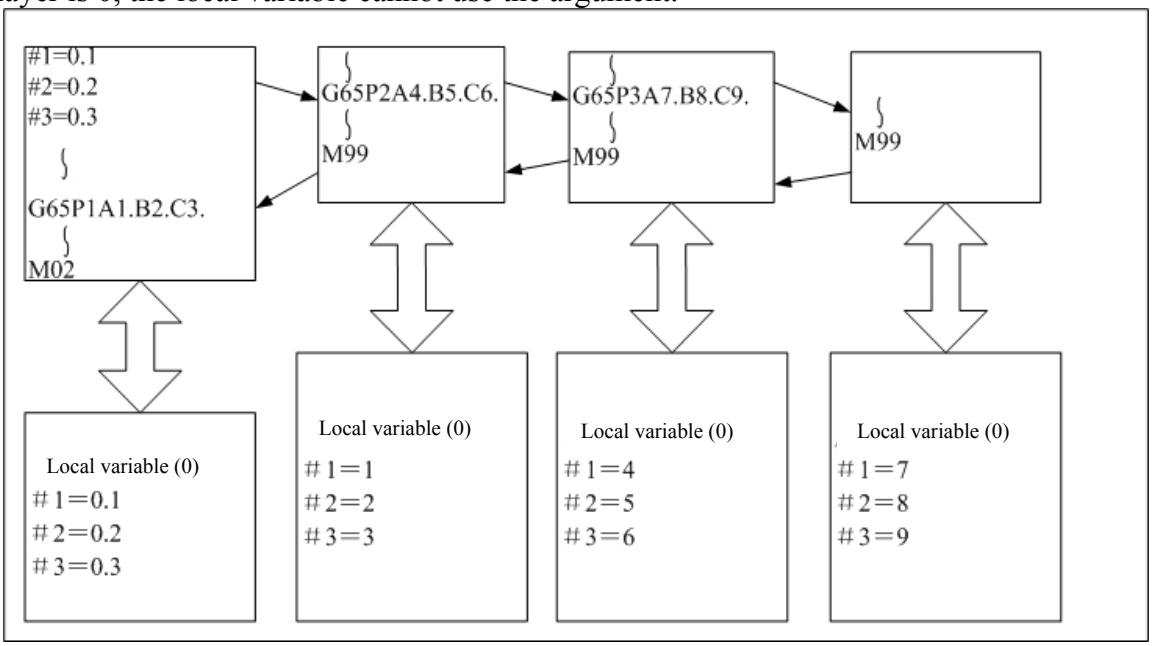


2) Local variable can be used freely in the sub program where it belongs to.



In face-milling processing, argument J means the distance of face-milling is 10mm, but in order to do the equal-distance processing, the distance is changed to 8.333mm.
In addition, local variable #30 is the calculation result of repeated processing.

3) local variable can be used independently in macro calling of each layer for as many as 4 layers. The main program (macro layer is 0) is provided with special local variable. However, in case the layer is 0, the local variable cannot use the argument.



1.4.4 Calculation instruction

Between variables, it is possible to use all kinds of calculation expressions.

Instruction format:

i = [Expression]

Expression can be combined by constant, variable, function or sub expression, etc.

In the following table, #j, #k can be replaced by constant.

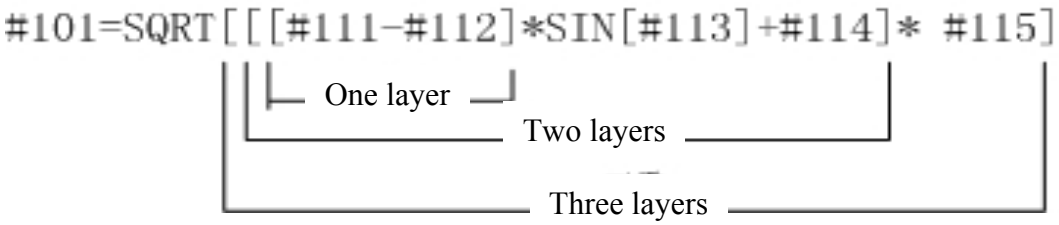
Calculation method	#i=#j	Definition, displacement
Addition & Subtraction	#i=#j+#k #i=#j-#k #i=#j OR #k or #i=#j|#k #i=#j XOR #k or #i=#j^#k	Addition Subtraction 32 bit OR calculation (logical sum) 32 bit XOR calculation
Multiplication & Division	#i=#j*#k #i=#j/#k #i=#j MOD #k #i=#j AND #k or #i=#j & #k	Multiplication Division Remainder 32 bit AND calculation (logical product)
Function	#i=SIN[#k] #i=COS[#k] #i=TAN[#k] #i=ASIN[#k] #i=ATAN[#k] #i=ACOS[#k] #i=SQRT[#k] #i=ABS[#k] #i=ROUND[#k] #i=FIX[#k] #i=FUP[#k] #i=LN[#k] #i=EXP[#k]	sine cos tanθ=sinθ/cosθ arc sine arctan arc cosine square root absolute value round FIX FUP natural logarithm exponent based on e(=2.718…)

- Values without decimal point are eyed as having decimal point (1=1.000)
- The expression behind the function should be enclosed by [ ].

Calculation priority of expression

Priority The smaller the number is, the higher the priority is	Calculation sign
1	#
2	[ ]
3	Function (SIN, COS,EXP…)
4	*,/,MOD
5	+,-
6	GE,GT,LE,LT
7	EQ,NE
8	AND,XOR,OR
9	=

- The expression of the same priority is calculated from left to right.
- The calculation expression may have many priorities. If the expression is long, it is recommended to add [] to force it to be preferential.
- [] can be embedded in calculation, and as many as 5 layers can be embedded. As the following example shows:



Examples of calculation instruction

(1) specification of main program and argument	#i=#j	Definition, displacement
(2) Definition, displacement (=)	#1=1000 #2=1000 #3=#101 #4=#102 #5=#41	#1 1000.000 #2 1000.000 #3 100.000 #4 200.000 #5 -10.000
(3) Addition, Subtraction (+ -)	#11=#1+1000 #12=#2-50 #13-#101+#1 #14=#41-3 #15=#41+#102	#11 2000.000 #12 950.000 #13 1100.000 #14 -13.000 #15 190.000
(4) logical sum (OR)	#3=100 #4=#3 XOR 14	#3=01100100 14=00001110 #4=01101110=110
(5) XOR (XOR)	#3 = 100 #4 = #3 XOR 14	#3=01100100 14=00001110 #4=01101010=106
(6) Multiplication, Division (* /)	#21=100*100 #22=100.*100 #23=100*100. #24=100.*100 #25=100/100 #26=100./100. #27=100/100. #28=100./100. #29=#41*#101 #30=#41/#102	#21 10000.000 #22 10000.000 #23 10000.000 #24 10000.000 #25 1.000 #26 1.000 #27 1.000 #28 1.000 #29 -1000.000 #30 -0.050
(7) remainder (MOD)	#31=#19 MOD #20	#19 48.000 #20 9.000 #31 3.000
(8) logical product (AND)	#9 = 100 #10= #9 AND 15	#9 =01100100 15 =00001111 #10=00000100=4
(9) sine (SIN)	#501=SIN[60] #502=1000*SIN[60]	#501 0.860 #502 866.025
(10) cosine (COS)	#541=COS[45] #542=1000*COS[45.]	#541 0.707 #542 707.107
(11) tangent (TAN)	#551=TAN[60] #552=1000*TAN[60]	#551 1.732 #552 1732.051
(12) arc sine (ASIN)	#531=ASIN[100.500/201.] #532=ASIN[0.500] #533=ASIN[-0.500]	#531 30.000 #532 30.000 #533 -30.000
(13) arctan (ATAN)	#561=ATAN[173205/100000] #562=ATAN[173205/100.] #563=ATAN[173.205/100000] #564=ATAN[173.205/100.]	#561 60.000 #562 60.000 #563 60.000 #564 60.000

	#565=ATAN[1.732]	#565 59.999
(14) arc cosine (ACOS)	#521=ACOS[100./141.421] #522=ACOS[10/14.142] #523=ACOS[0.707]	#521 45.000 #522 44.999 #523 45.009
(15) square root (SQRT)	#571=SQRT[1000] #572=SQRT[10.*10.+20.*20] #573=SQRT[#14*#14+#15*#15]	#571 31.623 #572 22.361 #573 190.444
(16) absolute value (ABS)	#576=-1000 #577=ABS[#576] #3 = 70. #4=-50. #580=ABS[#4-#3]	#576 -1000.000 #577 1000.000 #580 120.000
(17)		
(18) round (ROUND)	#21=ROUND[14/3] #22=ROUND[-14/3]	#21 5.000 #22 -5.000
(19) FIX (FIX)	#21=FIX[14/3] #22=FIX[-14/3]	#21 4.000 #22 -4.000
(20) FUP (FUP)	#21=FUP[14/3] #22=FUP[-14/3.]	#21 5.000 #22 -5.000
(21) natural logarithm (LN)	#101=LN[5] #102=LN[0.5] #103=LN[-5]	#101 1.609 #102 -0.693 error
(22) exponent (EXP)	#104=EXP[2] #105=EXP[1] #106=EXP[-2]	#104 7.389 #105 2.718 #106 0.135

Accuracy of calculation:

Macro variable is valued in 7-digit effective figure, therefore too big or too small the calculated value may cause the loss of accuracy, (9999999.000~0.0000001), and repeated calculation will lead to the accumulation of error. In this way, try to make the value of macro variable in a reasonable range. In addition, when calculating the trigonometric functions and exponential functions, due to the calculation error of function, too big the value is also a reason of causing multiplication of error.

1.4.5 Control instruction

Using IF-GOTO and WHILE~DO~ can control the flow of program.

Conditions instruction

Instruction format:

IF[conditional expression] GOTO n; (n is the order number in program)

If the conditional expression is established, the program will jump to line n to execute; and if it is not established, the expression will be executed downward as the sequence.

Once [conditional expression] is neglected, the program will execute GOTO sentence unconditionally.

The types of [conditional expression] are as follows:

#i EQ #j	= when #i and #j are equal
#i NE #j	≠ when #i and #j are not equal
#i GT #j	> when #i is larger than #j
#i LT #j	< when #i is smaller than #j
#i GE #j	≥ when #i is larger than or equal to #j
#i LE #j	≤ when #i is smaller than or equal to #j

- The n in GOTO should be existing in program; if not, the system will give a warning of abnormal program.
- #i, #j, n can be replaced by variable. For program segment contains order number n specified by GOTO n, the order number n should be in front of the program segment; otherwise when the program jumps, it may produce error due to the loss of key words. If there is a "/" in front of the specified program segment and Nn behind, the neglect function of this program segment will become ineffective but the program segment will still jump and be executed.
- When performing GOTO instruction to jump, it will search downward first. Once the search fails, it will return to the beginning of program and search downward again. If there is no result when it comes to calling segment, the system will give a warning message.
- EQ and NE can only be used in integer. Values with decimal should be compared with GT, GE, LT, and LE instructions.

Cycle condition judgement instruction

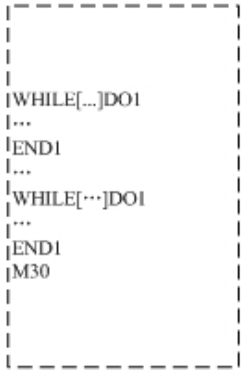
Instruction format:

WHILE [expression]DO m; (m=1,2,3…127)

…

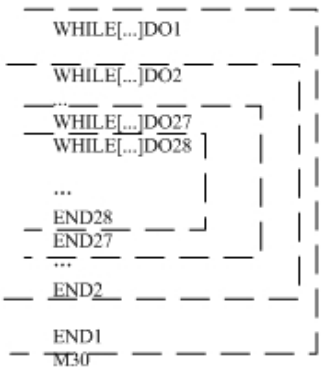
END m;

1) When the conditional expression is established, the program from WHILE to END will be executed repeatedly. If it is not established, it will jump to execute the next program segment of END m.
2) WHILE [expression]DO m and END m should be used in pair. When WHILE [expression] is neglected, DO m and END m are executed in turn repeatedly. The range of M is 1…127.
3) WHILE is allowed to nest, but the maximum nesting layers should not exceed 27.

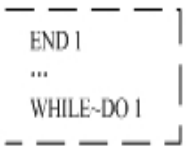
<table>
<tr>
<td>(1) the same identification number can be used repeatedly
 

Correct</td>
<td>(2) the identification number of WHILE~Dom can be specified with any value
 

Correct</td>
</tr>
<tr>
<td>(3) the maximum layers of WHILE~Dom is 27; the range of m is 1~127, and can be specified as you like
 

Correct
Note: In case of nesting, once m is specified, it cannot be used repeatedly.</td>
<td>(4) the layers of WHILE~Dom cannot exceed 28
 

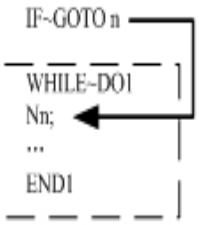
Wrong</td>
</tr>
</table>

<table>
<tr>
<td>(5) WHILE~Dom should be specified before END m
 

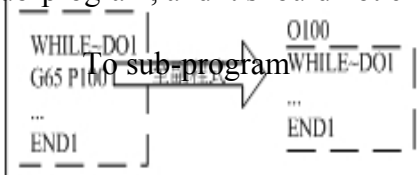
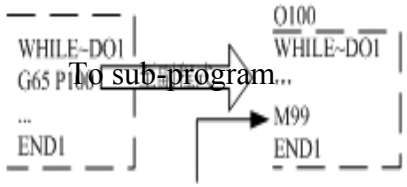
Wrong</td>
<td>(6) WHILE~DO m should be corresponding one by one in the same program
 

Wrong</td>
</tr>
<tr>
<td>(7) WHILE~DO m should not be used crosswise
 

Wrong</td>
<td>(8) it’s capable of calling sub-program, such as M98, G65 and G66 during the WHILE~DO m
 

Allowed</td>
</tr>
<tr>
<td>(9) GOTO cannot go to the cycle of WHILE
 

Wrong</td>
<td>(10) GOTO can skip out the cycle of WHILE
 

Correct</td>
</tr>
<tr>
<td>(11) If you have called sub-program in WHILE~DO cycle, and at the same time executed WHILE~DO inside sub-program, the nesting layers of WHILE is calculated together with main program and sub-program, and it should not exceed 27 layers.
 
</td>
<td>(12) If WHILE and END is not used in pair in macro program, it may produce program error in case of M99.
 

Wrong
M99 will cause the failure match of DO and END.</td>
</tr>
</table>

1.4.6 Notice for using macro

Macro program is a NC program using variable and calculation to group the logical description, which allows the program to be more universal. However, due to its flexible logical calculation mode, it is possible that some of the hidden errors may not be detected. To avoid some logical errors, it is necessary to pay attention to some compiling methods when compiling the macro.

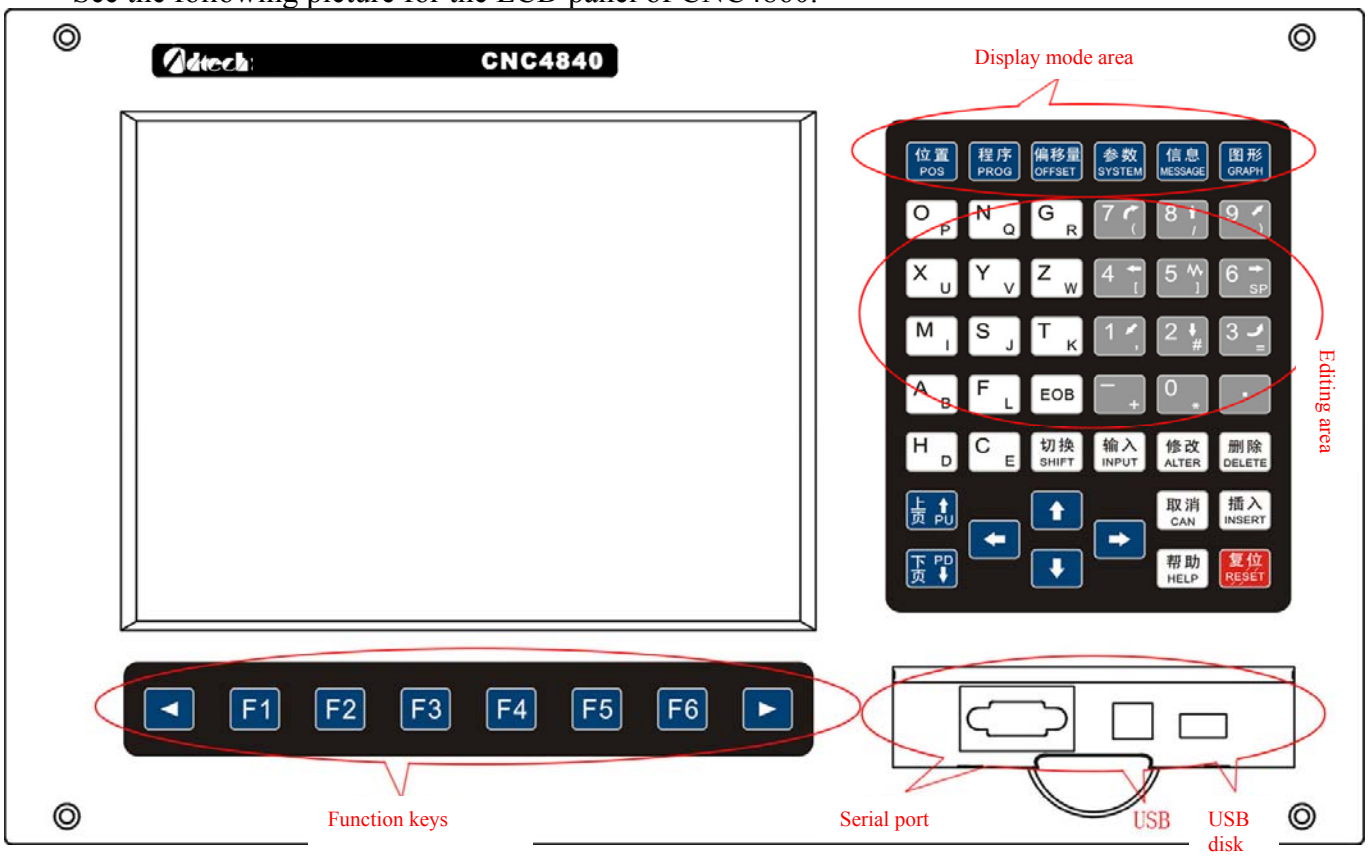
1) Variable Initialization; the variable used in program should be initialized at the beginning of program for transferring the variable value. One intervening variable is also needed for the transfer. this can prevent the program to modify the parameter environment when processing for many times.
2) Use local variable in main program, sub program or macro. The local variable will be cleared when called by program, which produces a clear environment for programming. Even it is wrongly quoted, it is easy to check step by step.
3) Macro is just the same as sub program, can not engage in the tool radius compensation. Therefore, before calling, it is essential to cancel the compensation function first.

1 Chapter III Operation

I. Description of Control Panel

1.2.1 1. LCD panel

See the following picture for the LCD panel of CNC4860.

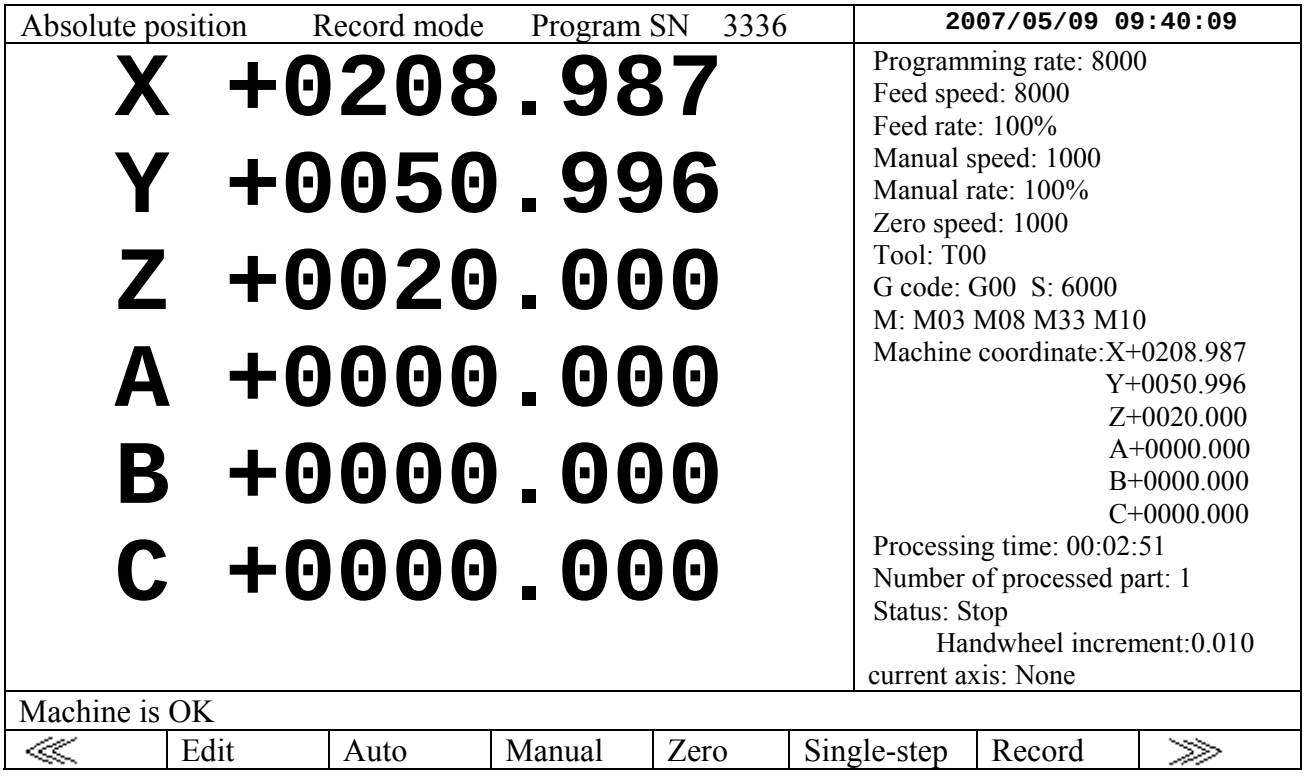


1.2 II. Description of display mode

When the system is started, the LCD will enter the operating state and the top left of screen will show the operating status, the first column is the display mode, the second column is operating mode, while the third column is the number of current program. At the right side, it is the date/time display area.

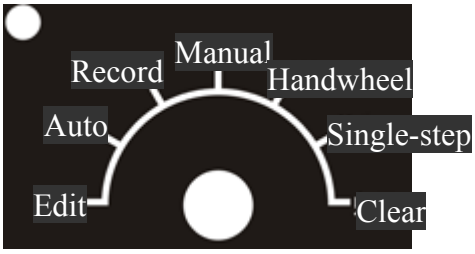
In the middle of the screen, the left is the content of all display interfaces, and the right displays the status of current M code, G code, S code and T code, as well as the processing information.

The bottom displays the menu, and the alarm message when alarming. The top left corner of this line shows the status of machine tool.



There are two pages of menu. You can press the function keys on the right side to switch the main menu or the function key on the left right to exit the sub-menu and return to the main menu. The first page includes [Edit], [Auto], [Manual], [Clear], [Single-step], [Record] (note: the menu of this page will become invalid if additional panel is connected), the second page includes [POS], [PROG], [OFFSET], [SYSTEM], [DIAGNOSIS], and [MESSAGE]. The function keys are displayed corresponding to the display mode of current page. (when the system is started, the menu column is displayed as menu of operating mode, and when you press the function keys on the right side, it is switched to menu of display mode.)

Connect the additional panel, the operating mode is operated by rotary button of mode shift: rotary mode shift is switched to the corresponding operating mode.



S/N	Name	Remark
1	[Edit]	edit mode
2	[Auto	auto mode
3	[Manual]	Manual mode
4	[Clear]	Clear mode

5	[Handwheel]	Handwheel mode
6	[Single-step]	Single-step mode
7	[Record]	Recording mode
8	[POS] 位置 POS	display the interface of coordinate position, the sub-menu has three options, absolute position, relative position, and comprehensive position
9	[PROG] 程序 PROG	display the menu of program edition, MDI, program catalog, serial communication, file management and graph type
10	[OFFSET 偏移量 OFFSET	Display the interface of setting tool compensation, including the display and settings of tool length and radius compensation parameter
11	[SYSTEM] 参数 SYSTEM	display the system parameter, coordinate parameter, network parameter, and setting interface
12	[DIAGNOSIS]	this interface can be entered only in manual mode; it includes input diagnosis and output diagnosis
13	[MESSAGE] 信息 MESSAGE	display the alarm message interface

1.2.1 1. position display

There are three pages for the position interface, you can press F1 F2 F3 to switch among absolute position, relative position, and comprehensive position.

Absolute position: display the position of address X, Y, Z, A,B,C in work piece coordinate system, also called as work piece coordinate, which is corresponding to the programming absolute value. It can be changed by G92 settings.

Relative position: display the displacement of address X, Y, Z, A ,B, and C corresponding to work piece coordinate system. It is can cleared at any time in manual mode, and it is for observing the position or for setting the cutter compensation value of counting way.

Comprehensive position: display the absolute position, relative position, machine tool position, and the allowance. The reference point of machine tool is the origin of coordinate system. Generally, this coordinate system would be change, it's for examining the soft limit.

1. Interface of absolute position

Absolute position Record mode Program SN 3336	2007/05/09 09:40:09
X +0208.987 Y +0050.996 Z +0020.000 A +0000.000 B +0000.000 C +0000.000	Programming rate: 8000 Feed speed: 8000 Feed rate: 100% Manual speed: 1000 Manual rate: 100% Zero speed: 1000 Tool: T00 G code: G00 S: 6000 M: M03 M08 M33 M10 Machine coordinate:X+0208.987 Y+0050.996 Z+0020.000 A+0000.000 B+0000.000 C+0000.000 Processing time: 00:02:51 Number of processed part: 1 Status: Stop Handwheel increment:0.010 current axis: None
Machine is OK	
<<< Absolute Relative Comprehensive	

2. Interface of relative position

Absolute position Record mode Program SN 3336	2007/05/09 09:40:09
X +0208.987 Y +0050.996 Z +0020.000 A +0000.000 B +0000.000 C +0000.000	Programming rate: 8000 Feed speed: 8000 Feed rate: 100% Manual speed: 1000 Manual rate: 100% Zero speed: 1000 Tool: T00 G code: G00 S: 6000 M: M03 M08 M33 M10 Machine coordinate:X+0208.987 Y+0050.996 Z+0020.000 A+0000.000 B+0000.000 C+0000.000 Processing time: 00:02:51 Number of processed part: 1 Status: Stop Handwheel increment:0.010 current axis: None
Machine is OK	
<<< Absolute Relative Comprehensive	

In manual mode, press X U, Y V, Z W or A B key in this interface, and the corresponding address will become green and flash, then press 取消 CAN key, the relative position of flashing address will be reset to 0.

3. Interface of comprehensive position

Absolute position Record mode Program SN 3336	2007/05/09 09:40:09
Absolute position X:+0208.987 Y:+0050.996 Z:+0020.000 A:+0000.000 B:+0000.000 C:+0000.000 Machine position X:+0208.987 Y:+0050.996 Z:+0020.000 A:+0000.000 B:+0000.000 C:+0000.000 relative position X:+0208.987 Y:+0050.996 Z:+0020.000 A:+0000.000 B:+0000.000 C:+0000.000 allowance position X:-0000.000 Y:-0102.241 Z:+0000.000 A:+0000.000 B:+0000.000 C:+0000.000	Programming rate: 8000 Feed speed: 8000 Feed rate: 100% Manual speed: 1000 Manual rate: 100% Zero speed: 1000 Tool: T00 G code: G00 S: 6000 M: M03 M08 M33 M10 Machine coordinate:X+0208.987 Y+0050.996 Z+0020.000 A+0000.000 B+0000.000 C+0000.000 Processing time: 00:02:51 Number of processed part: 1 Status: Stop Handwheel increment:0.010 current axis: None
Machine is OK	
≪ | Absolute | Relative | Comprehensive |	

1.2.2 2. Program display

1. Program edition

In editing mode, you can edit the program, press 上页 PU 下页 PD to page up or page down the content of program.

Absolute position Record mode Program SN 3336	2007/05/09 09:40:09
O3366 (PROGRAM NAME - 1) (DATE=DD-MM-YY - 14-01-07 TIME=HH:MM - 10:41) N100G21 N102G0G17G40G49G80G90 (TOOL - 2 DIA. OFF. - 2 LEN. - 2 DIA . - 22.) M08 M6T2 N106G0G90G54X-144.829Y-30.A0.S10000M3 N108G43H2Z20. N110Z5. N112G1Z0.F4000. N114Y30. N116X-147.829Y-30. N118Y30. N120Z5. G02←————————————Enter data line	Programming rate: 8000 Feed speed: 8000 Feed rate: 100% Manual speed: 1000 Manual rate: 100% Zero speed: 1000 Tool: T00 G code: G00 S: 6000 M: M03 M08 M33 M10 Machine coordinate:X+0208.987 Y+0050.996 Z+0020.000 A+0000.000 B+0000.000 C+0000.000 Processing time: 00:02:51 Number of processed part: 1 Status: Stop Handwheel increment:0.010 current axis: None
Machine is OK	
⋘ Program MDI Directory Communication Graph File	

2. MDI

The main function of MDI operation is to ensure that the instruction can be started and run once inserted in recording mode.

MDI operation:

Select [Record] mode → press [MDI] F2 to switch to MDI interface → enter the words to be executed and press 插入 INSERT → press number key [7] or [Start], the system will run the program segment. It will stop if the operation is over. You can also continue performing the MDI operation.

MDI operation Record mode Program SN 3336	2007/05/09 09:40:09
Program segment mode value G X G90 X+0208.987 Y G17 Y+0050.996 Z G91 Z+0020.000 A G98 A+0000.000 B G80 A+0000.000 C G49 A+0000.000 J G40 K R M T S P Q F	Programming rate: 8000 Feed speed: 8000 Feed rate: 100% Manual speed: 1000 Manual rate: 100% Zero speed: 1000 Tool: T00 G code: G00 S: 6000 M: M03 M08 M33 M10 Machine coordinate:X+0208.987 Y+0050.996 Z+0020.000 A+0000.000 B+0000.000 C+0000.000 Processing time: 00:02:51 Number of processed part: 1 Status: Stop Handwheel increment:0.010 current axis: None
Machine is OK	
⋘ Program MDI Directory Communication Graph File	

3. Program catalog

Press [Catalog] F3 to enter the interface of program catalog as follows. The catalog mainly shows all processing file catalogs in working area and the network parameters.

Operations of reading files:

Press ↑ ↓ to select the number of processing program → press T K key to select the current processing file Oxxxx,

Operations on file management:

You can press <EOB> EOB key to change the file name (note: not available for the current version)

Operations to connect the computer:

Press G R key to enter the USB connection (CNC4860 is connected to the computer in USB communicating mode)

Once CNC4860 is connected to the computer with USB cable, you can press G R key in catalog interface to enter the USB transmission interface where a removable disk will be detected on the computer. Open the removable disk, you will find tow folders (ADT and PRG). Folder ADT is to store CNC4860 system program and system parameters, and the files under this catalog cannot be deleted. Folder PRG is to store the processing files of system (Note: saved as xxxx.CNC). You can press 取消 CAN key to exit the USB communication.

Press 取消 CAN key to disconnect the USB connection with the computer (Note: press it successively twice).

Press 复位 RESET key, and the CNC4860 system will be restarted.

MDI operation Record mode Program SN 3336	2007/05/09 09:40:09
Current working area: \PRG\2222.version number: V2007.04.23 Program saved: 1 spare: 99999 Used capacity: 144217 spare capacity: 25021607 03366 1. IP address: 192.168.0.100 2. Gateway: 192.168.0.1 3. Subnet Mask: 255.255.255.0 4. Computer IP address: 192.168.0.150 5. MAC address: 85.136.218.68.153.238 6. Computer port S/N: 2048 T: Read file EOB: File management G: Enter USB disk CAN: Cancel RESET: reset	Programming rate: 8000 Feed speed: 8000 Feed rate: 100% Manual speed: 1000 Manual rate: 100% Zero speed: 1000 Tool: T00 G code: G00 S: 6000 M: M03 M08 M33 M10 Machine coordinate:X+0208.987 Y+0050.996 Z+0020.000 A+0000.000 B+0000.000 C+0000.000 Processing time: 00:02:51 Number of processed part: 1 Status: Stop Handwheel increment:0.010 current axis: None
Machine is OK	
≪ Program MDI Directory Communication Graph File	

4. Serial communication

You can press [Communication] key to enter the interface of serial communication operation as follows. The main function is to transmit the processing program in mode of serial communication.

<table>
<tr><td>MDI operation Record mode Program SN 3336</td><td>2007/05/09 09:40:09</td></tr>
<tr><td>Controller ID: 1 Baud rate: 115200bps
Data format: 8 data bits, 1 stop bit, odd check
Offline status of MODBus serial port communication
03366

O3366
(PROGRAM NAME - 1)
(DATE=DD-MM-YY - 14-01-07 TIME=HH:MM - 10:41)
N100G21
N102G0G17G40G49G80G90
(TOOL - 2 DIA. OFF. - 2 LEN. - 2 DIA. - 22.)
T: MODBus serial receiving file started
CAN: MODBus serial receiving file closed</td><td>Programming rate: 8000
Feed speed: 8000
Feed rate: 100%
Manual speed: 1000
Manual rate: 100%
Zero speed: 1000
Tool: T00
G code: G00 S: 6000
M: M03 M08 M33 M10
Machine coordinate:X+0208.987
Y+0050.996
Z+0020.000
A+0000.000
B+0000.000
C+0000.000
Processing time: 00:02:51
Number of processed part: 1
Status: Stop
Handwheel increment:0.010 current axis: None</td></tr>
<tr><td colspan="2">Machine is OK</td></tr>
<tr><td colspan="2">≪ | Program | MDI | Directory | Communication | Graph | File</td></tr>
</table>

Controller ID: it's the address number for transmitting processing file CNC4860 via serial port, which can be set by #96 system parameter.

Baud rate: The default baud rate for controller to communicate by serial port with software of upper computer is 115200. If the transmission distance is far, it is recommended to change the baud rate to 38400 to get more stable communication.

Data format: refer to the data format for controller to communicate by serial port with software of upper computer

You can press T K key to start the serial communication and to receive the processing files. MODBus serial communication is at online state and the upper computer can download the processing code file via the serial port.

Press 取消 CAN key to close the serial port and stop receiving the files. MODBus serial communication is at offline state and the upper computer cannot download the processing code file via the serial port.

5. Figure display

Press [Figure] F5 key to enter the figure display interface as follows. It is mainly used to draw the plane figure of tool interpolation path.

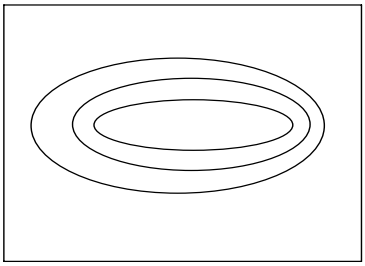
Parameter F1: enable to set coordinate plane, scaling, center coordinate of figure, and display area

Figure F2: Figure display area

Start F3: start drawing figure

Stop F4: stop drawing figure

Clear F5: clear the figure

MDI operation Record mode Program SN 3336	2007/05/09 09:40:09
	Programming rate: 8000 Feed speed: 8000 Feed rate: 100% Manual speed: 1000 Manual rate: 100% Zero speed: 1000 Tool: T00 G code: G00 S: 6000 M: M03 M08 M33 M10 Machine coordinate:X+0208.987 Y+0050.996 Z+0020.000 A+0000.000 B+0000.000 C+0000.000 Processing time: 00:02:51 Number of processed part: 1 Status: Stop Handwheel increment:0.010 current axis: None
Machine is OK	

≪	Parameter	Graph	Start	Stop	Clear		

MDI operation Record mode Program SN 3336	2007/05/09 09:40:09
Coordinate selection: 0 X reduction scale: 300 Y reduction scale: 300 X offset of graph: -200 Y offset of graph: -200 G00 quick move track display(0&1): 0 Polar axis angle (0~360) 45	Programming rate: 8000 Feed speed: 8000 Feed rate: 100% Manual speed: 1000 Manual rate: 100% Zero speed: 1000 Tool: T00 G code: G00 S: 6000 M: M03 M08 M33 M10 Machine coordinate:X+0208.987 Y+0050.996 Z+0020.000 A+0000.000 B+0000.000 C+0000.000 Processing time: 00:02:51 Number of processed part: 1 Status: Stop Handwheel increment:0.010 current axis: None
Machine is OK	

≪	Parameter	Graph	Start	Stop	Clear		

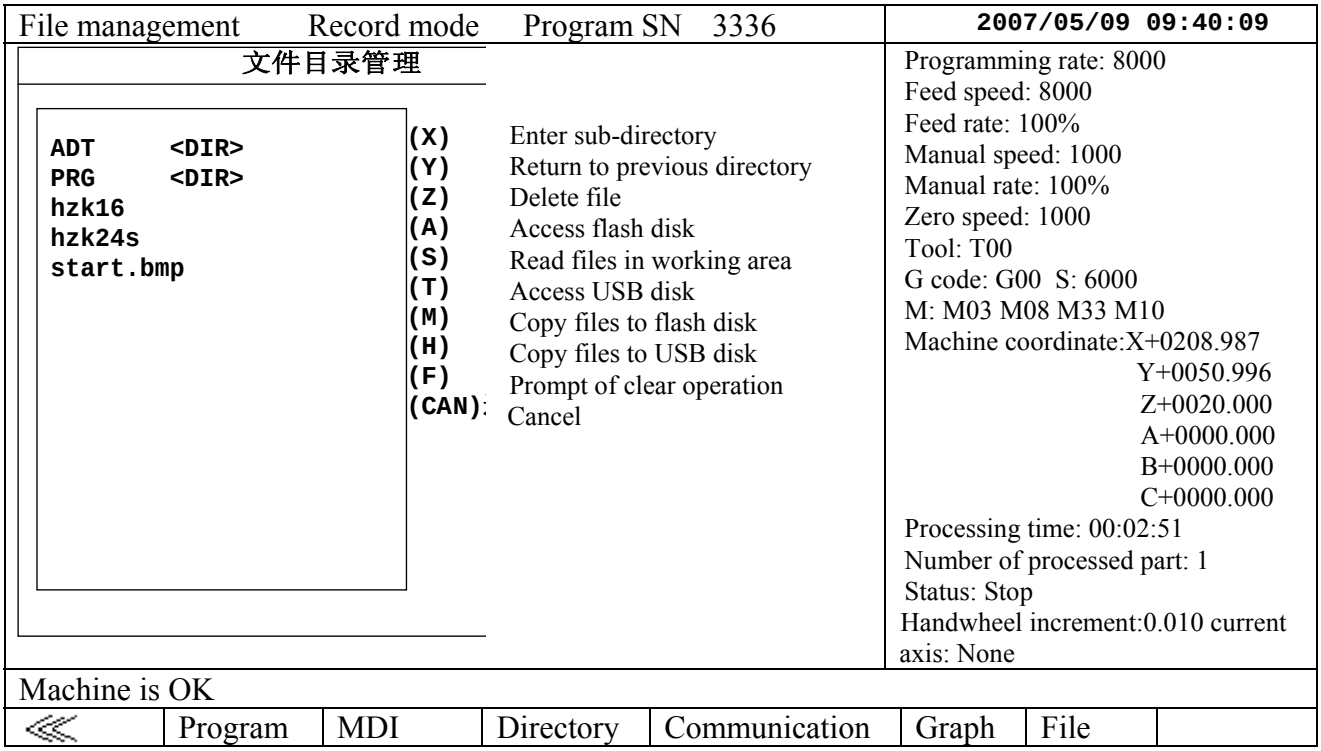
6. File management

Press [File] F6 key to enter the file management interface as follows. The main function is to operate the external USB disk.

Operating interface of file management:

1. Catalog ADT is to store CNC4860 system program and system parameters, and the files under this catalog cannot be deleted.
2. Catalog PRG is to store the processing files of system (Note: saved as xxxx.CNC).
3. Menu operations: You can press X U key to enter the subdirectory, press <Y> Y V key to return to the previous catalog, press Z W key to delete the current selected files (note: files and

catalog under the catalog ADT cannot be deleted), press A B key to access the catalog and files of system flash disk, press S J to read files in working area (note: only xxxx.CNC file under the catalog PRG of flash disk can be read. After the reading, the system will reserve the files of this working area until reading the files of other working area next time), press T K key to access the external USB disk, press M I key to copy xxxx.CNC file in external USB disk to catalog PRG of the flash disk (note: if there are repeated files, the new one will cover the original one).



1.2.3 3. Settings of tool compensation parameters

You can press 偏移量 OFFSET or F3 key to enter the interface of tool compensation settings, and press ↑ ↓ or ← → key to select the tool compensation number that is corresponding to the radius or length. Tool compensation parameters, including the radius compensation and length compensation, are used to set the tool radius and length offset. The radius compensation number is from 01 to 36, and the length compensation number is from 01 to 36. To set or modify the tool compensation value, input the compensation value in editing or recording mode and press 插入 INSERT key to complete.

Tool compensation	Record mode	Program SN	3336	2007/05/09 09:40:09

S/N	Radius	S/N	Length
01	+0000	01	+0000.000
02	+0000	02	+0000.000
03	+0000	03	+0000.000
04	+0000	04	+0000.000
05	+0000	05	+0000.000
06	+0000	06	+0000.000
07	+0000	07	+0000.000
08	+0000	08	+0000.000
09	+0000	09	+0000.000
10	+0000	10	+0000.000
11	+0000	11	+0000.000
12	+0000	12	+0000.000

0

Programming rate: 8000
Feed speed: 8000
Feed rate: 100%
Manual speed: 1000
Manual rate: 100%
Zero speed: 1000
Tool: T00
G code: G00 S: 6000
M: M03 M08 M33 M10
Machine coordinate:X+0208.987
Y+0050.996
Z+0020.000
A+0000.000
B+0000.000
C+0000.000
Processing time: 00:02:51
Number of processed part: 1
Status: Stop
Handwheel increment:0.010 current axis: None

Machine is OK

⋘	Position	Program	Offset	Parameter	Diagnosis	Alarm	

1.2.4 4. Setting and modification of system parameters, coordinate parameters, network parameters, setting parameters and parameter management

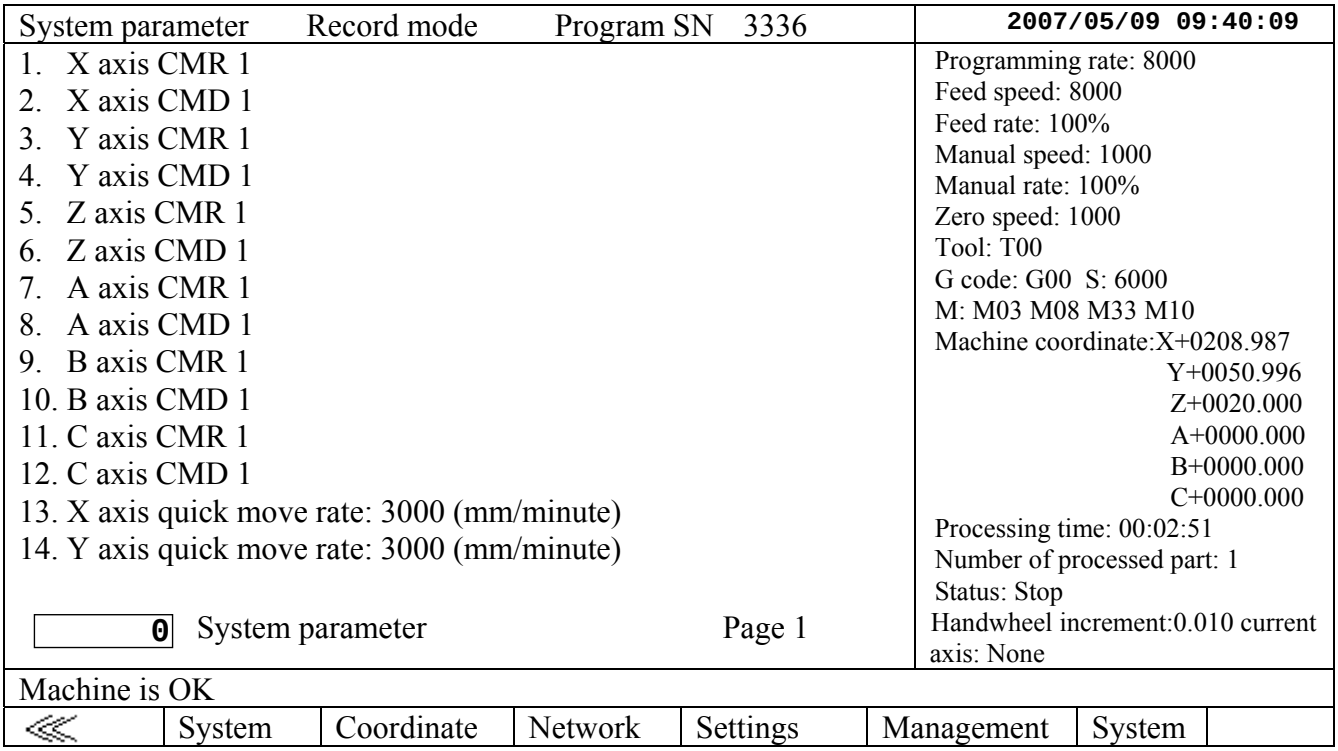
You can press 参数 SYSTEM or F4 key to enter the system parameter interface. To ensure the parameters would not be modified wrongly, the parameters can only be modified in [Record] or [Edit] mode.

Ways to modify the parameters:

In recording mode, press ↑ ↓ key to move the cursor to the parameter need to be modified →enter the data →press 插入 INSERT key, and the data will then be modified. (you can press 取消 CAN key to clear the data that are not inserted. If the input data exceed the allowed range, the system will adjust it to the upper or lower value automatically).

Methods to inquiry the system parameters:

In recording mode, you can input the parameter number and press EOB key to search the corresponding parameter number (If the input parameter number is larger than the numbers of parameters or smaller or equal to 0, it will not get any result).



Settings of workpiece coordinate parameters

In system parameter interface, you can press → cursor key once to switch to coordinate parameter interface and press  key to move the cursor. The main function is to set the mechanical position of workpiece coordinate system. (for methods of setting workpiece coordinate system, see the annex)

1. Settings of workpiece coordinate system

The machine tool has 6 workpiece coordinate systems, from G54~G59respectively, you can select any one of them.

G54.....workpiece coordinate system 1

G55.....workpiece coordinate system 2

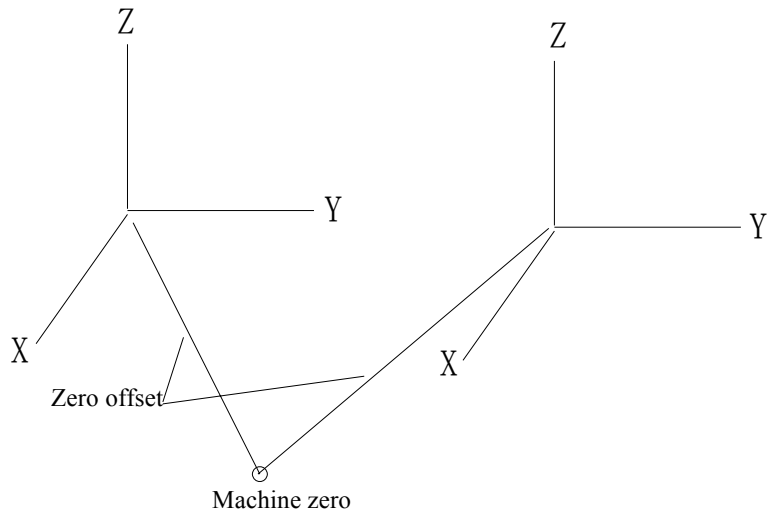
G56.....workpiece coordinate system 3

G57.....workpiece coordinate system 4

G58.....workpiece coordinate system 5

G59.....workpiece coordinate system 6

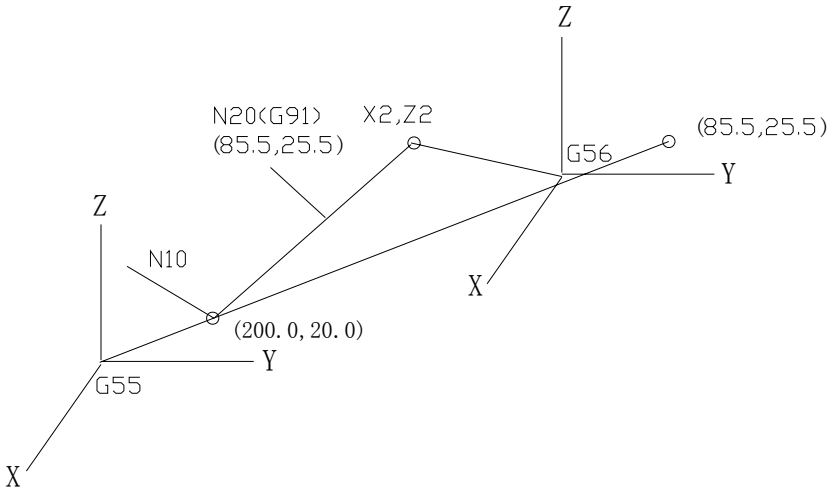
These 6 workpiece coordinate systems are set by distance (Part zero offset) from machine zero to the zero point of each coordinate system .



For example: N10G55G90G00X100.0Z20.0

N20G56X80.5Z25.5

In the above example, the position in workpiece coordinate system 2 (X=100.0, Z=20.0) is quickly located to position in workpiece coordinate system 3 (X=80.5, Z=25.5). If N20 program segment is G91, it's increment movement. When starting to execute the N20 program segment, the absolute coordinate value will be changed to the coordinate value in G57 coordinate system.



After starting the system and returning to machine zero, the workpiece coordinate system 1~6 are established. Select G54 (workpiece coordinate system) when starting the system. The absolute position of the position interface is at the coordinate value of the current coordinate system.

2. Display and input of workpiece zero

In recording or editing mode, you can press ↑ or ↓ to move the cursor to X, Y, Z, or A axis, select the corresponding workpiece coordinate system from G54, G55……G59, input the part zero offset and press 插入 INSERT key to set it as the zero offset of workpiece coordinate system, or press EOB key to set the current machine coordinate as the zero offset of the workpiece coordinate.

Coordinate parameter Record mode Program SN 3336	2007/05/09 09:40:09
1. Workpiece coordinate G54, X (mm): +0.000 G54, Y (mm): +0.000 G54, Z(mm): +0.000 G54, A(mm): +0.000 G54, B(mm): +0.000 G54, C(mm): +0.000 2. 2. Workpiece coordinate G55, X (mm): +0.000 G55, Y (mm): +0.000 G55, Z(mm): +0.000 G55, A(mm): +0.000 G55, B(mm): +0.000 G55, C(mm): +0.000 3. 3. Workpiece coordinate G56, X (mm): +0.000 G56, Y (mm): +0.000 0 Coordinate parameter Page 1	Programming rate: 8000 Feed speed: 8000 Feed rate: 100% Manual speed: 1000 Manual rate: 100% Zero speed: 1000 Tool: T00 G code: G00 S: 6000 M: M03 M08 M33 M10 Machine coordinate:X+0208.987 Y+0050.996 Z+0020.000 A+0000.000 B+0000.000 C+0000.000 Processing time: 00:02:51 Number of processed part: 1 Status: Stop Handwheel increment:0.010 current axis: None
Machine is OK	

	System	Coordinate	Network	Settings	Management	System	

3. Settings of network parameters

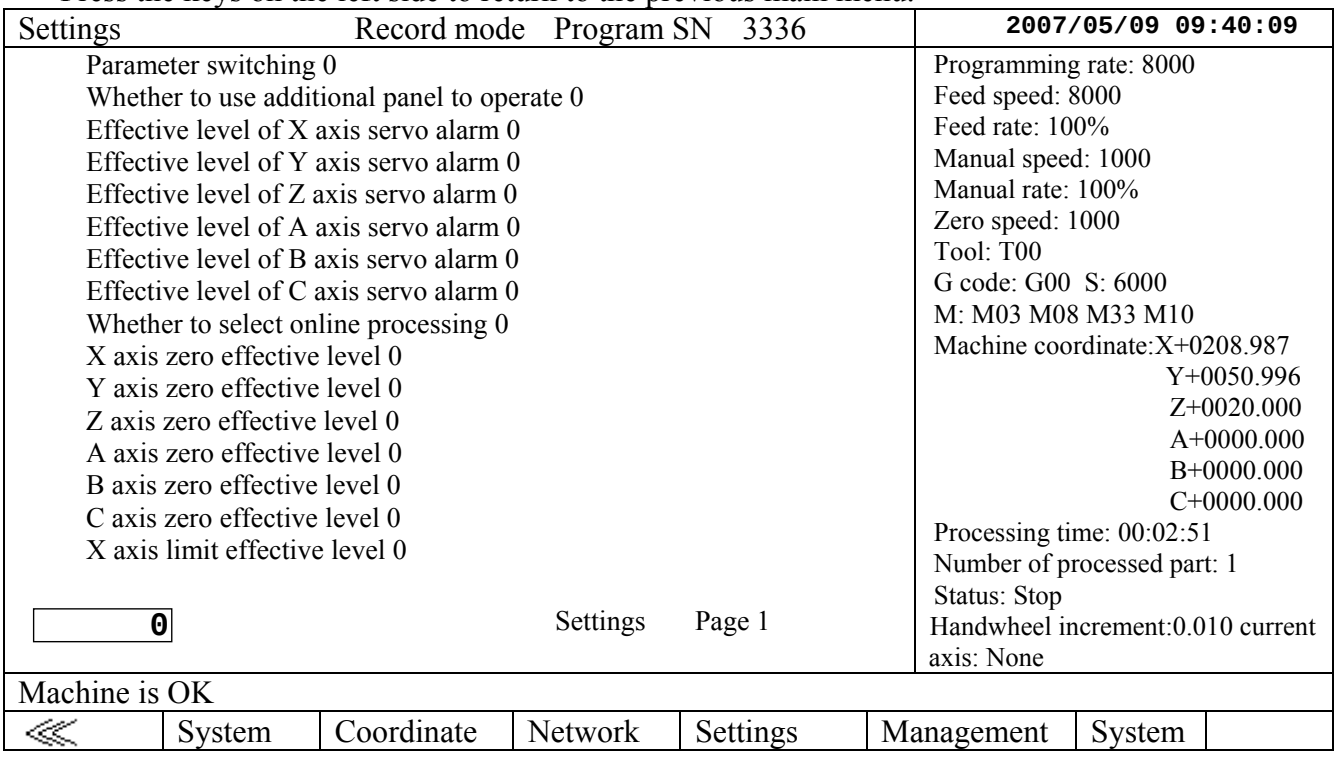
In system parameter interface, you can press the left and right cursor keys to switch to network parameter interface and press  key to move the cursor. (In recording mode, you can set the IP address, gateway, and subnet mask of CNC4860, IP address of upper computer, and the port number of online processing software. The setting methods are the same as those on computer, but the IP address in LAN should not conflict. Other options are set according to specific circumstances)

Network parameter Record mode Program SN 3336	2007/05/09 09:40:09
1. IP address: 192.168.0.100 2. Gateway: 192.168.0.1 3. Subnet Mask: 255.255.255.0 4. Computer IP address: 192.168.0.150 5. MAC address: 85.136.218.68.153.238 6. Computer port S/N: 2048 0 Network parameter Page 1	Programming rate: 8000 Feed speed: 8000 Feed rate: 100% Manual speed: 1000 Manual rate: 100% Zero speed: 1000 Tool: T00 G code: G00 S: 6000 M: M03 M08 M33 M10 Machine coordinate:X+0208.987 Y+0050.996 Z+0020.000 A+0000.000 B+0000.000 C+0000.000 Processing time: 00:02:51 Number of processed part: 1 Status: Stop Handwheel increment:0.010 current axis: None
Machine is OK	

	System	Coordinate	Network	Settings	Management	System	

4. Settings

Setting parameter switching (0: system parameter can be modified; 1: system parameter cannot be modified)
Selection of addition panel (0: do not use additional panel; 1: use additional panel)
Selection of servo alarm level (XYZABC axis 0: low level effective; 1: high level effective)
Whether to choose network online processing (0: No; 1: Yes; for large CNC processing file, you can select network online transmitting and processing)
Selection of zero effective level (XYZABC axis 0: low level effective; 1: high level effective)
Selection of limit effective level (XYZABC) axis 0: low level effective; 1: high level effective)
Axis pulse direction logic (Set the XYZABC axis direction logic, 0: positive logic; 1: negative logic; note: it's required to restart the system after the settings)
Press the keys on the left side to return to the previous main menu.



1.2.5 5. Diagnosis display setting

In manual mode, you can press [Diagnosis] F5 software key to enter the I / O diagnosis interface, and press [Input] and [Output] to switch between the input and output diagnosis interfaces. The main function is to test the input / output status of machine.
The input diagnosis includes basic input, extended input and other inputs.
You can press F4, F5, or F6 key to switch the display interface, and when the input signal of corresponding point is effective, it is displayed in red.
The basic input of machine is corresponding to 37-pin interface of XS1 machine input. For detailed definition and purpose, please refer to section–Connection.

Input diagnosis	Manual mode	Program SN	3336	2007/05/09 09:40:09
Basic input of machine				Programming rate: 8000 Feed speed: 8000 Feed rate: 100% Manual speed: 1000 Manual rate: 100% Zero speed: 1000 Tool: T00 G code: G00 S: 6000 M: M03 M08 M33 M10 Machine coordinate:X+0208.987 Y+0050.996 Z+0020.000 A+0000.000 B+0000.000 C+0000.000 Processing time: 00:02:51 Number of processed part: 1 Status: Stop Handwheel increment:0.010 current axis: None
X positive limit I0	X negative limit I1	Y positive limit I2	X positive limit I0	
Z positive limit I4	Z negative limit I5	A positive limit I6	A negative limit I7	
X axis origin I8	Y axis origin I9	Z axis origin I10	A axis origin I11	
B axis origin I12	C axis origin I13	IN14	IN15	
IN16	Adjusting tool I17	IN18	IN19	
Start I20	Pause I21	Emergency stop I22	Main axis alarm I23	
Touch tool setting I24	Tool setting I25	IN26	IN27	
IN28	IN29	IN30	IN30	
Basic input F4	Extended input F5	Other inputs F6		
Machine is OK				
Input	Output			

The extended input of machine is corresponding to extended 25-pin interface of XS5 machine input. For detailed definition and purpose, please refer to section–Connection.

Input diagnosis	Manual mode	Program SN	3336	2007/05/09 09:40:09
Extended input of machine				Programming rate: 8000 Feed speed: 8000 Feed rate: 100% Manual speed: 1000 Manual rate: 100% Zero speed: 1000 Tool: T00 G code: G00 S: 6000 M: M03 M08 M33 M10 Machine coordinate:X+0208.987 Y+0050.996 Z+0020.000 A+0000.000 B+0000.000 C+0000.000 Processing time: 00:02:51 Number of processed part: 1 Status: Stop Handwheel increment:0.010 current axis: None
B positive limit I 32	B negative limit I 33	C positive limit I 34	C negative limit I 35	
IN36	IN37	IN38	IN39	
IN40	IN41	IN42	IN43	
IN44	IN45	IN46	IN47	
IN48	IN49	IN50	IN51	
IN52	IN53	IN54	IN55	
Basic input F4	Extended input F5	Other inputs F6		
Machine is OK				
≪	Input	Output		

Other inputs are corresponding to XS6 handheld box, XS7 X axis, XS8Y axis, XS9Z axis, XS10A axis, XS11B axis, and XS12C axis pulse interfaces. For detailed definition and purpose, please refer to section–Connection.

Input diagnosis	Manual mode	Program SN	3336	2007/05/09 09:40:09
Other inputs				Programming rate: 8000 Feed speed: 8000 Feed rate: 100% Manual speed: 1000 Manual rate: 100% Zero speed: 1000 Tool: T00 G code: G00 S: 6000 M: M03 M08 M33 M10 Machine coordinate:X+0208.987 Y+0050.996 Z+0020.000 A+0000.000 B+0000.000 C+0000.000 Processing time: 00:02:51 Number of processed part: 1 Status: Stop Handwheel increment:0.010 current axis: None
Handwheel X axis I 56	Handwheel Y axis I 57	Handwheel Z axis I 58	Handwheel A axis I 59	
Handwheel start 60	Handwheel stop 61	Handwheel emergency stop 62	Handwheel 0.1 I 63	
Handwheel 0.01 I 64	Handwheel .00165	X axis ALMI 66	Y axis ALMI 67	
Z axis ALMI 68	A axis ALMI 69	B axis ALMI 70	C axis ALMI 71	
X axis ECZI 76	Y axis ECZI 73	Z axis ECZI 74	A axis ECZI 75	
B axis ECZI 76	C axis ECZI 77	X axis ECAI 78	X axis ECBI 79	
Y axis ECAI 80	Y axis ECBI 81	Z axis ECAI 82	Z axis ECBI 83	
A axis ECAI 84	A axis ECBI 85	B axis ECAI 86	B axis ECBI 87	
C axis ECAI 88	C axis ECBI 86	Additional emergency stop 92	C axis ECAI 88	
Basic input F4	Extended input F5	Other inputs F6		
Machine is OK				
≪	Input	Output		

Output diagnosis include basic output, extended input and other input

You can press F4 or F5 key to switch the operating display interface.

The basic output of machine is corresponding to 25-pin interface of XS2 machine output. For detailed definition and purpose, please refer to section–Connection.

Output diagnosis	Manual mode	Program SN	3336	2007/05/09 09:40:09
Basic output of machine				Programming rate: 8000 Feed speed: 8000 Feed rate: 100% Manual speed: 1000 Manual rate: 100% Zero speed: 1000 Tool: T00 G code: G00 S: 6000 M: M03 M08 M33 M10 Machine coordinate:X+0208.987 Y+0050.996 Z+0020.000 A+0000.000 B+0000.000 C+0000.000 Processing time: 00:02:51 Number of processed part: 1 Status: Stop Handwheel increment:0.010 current axis: None
Main axis positive rotation 0	Main axis reverse rotation 1	Illumination 2	7 Cooling A3	
8 Cooling B4	9 Lubricate 05	X adjusting tool 06	Y tool changing 07	
Z blowing 08	4 Clamp 09	5 Feed 010	6 Start the light 011	
M close the light 012	S alarm lamp 013	T discharge materials 014	1 discharge scrap 019	
2 tool magazine +016	3 tool magazine -017	A Backup 018	F Backup 019	
EOB main shift 1	-main shift 2	Main shift 3	Main shift 4	
Basic input F4	Extended input F5			
Machine is OK				
⋘	Input	Output		

The above output points OT00---OT23 are corresponding to [O][N][G][7][8][9][X][Y][Z][4][5][6][M][S][T][1][2][3][F] [H][EOB][-][0] and [.] keys on the panel.

The extended and pulse outputs are corresponding to 25-pin interface of XS3 machine extended output, as well as XS7 X axis, XS8Y axis, XS9Z axis, XS10A axis, XS11B axis, and XS12C axis pulse interfaces. For detailed definition and purpose, please refer to section–Connection.

Output diagnosis	Manual mode	Program SN	3336	2007/05/09 09:40:09
Extended and pulse output				Programming rate: 8000 Feed speed: 8000 Feed rate: 100% Manual speed: 1000 Manual rate: 100% Zero speed: 1000 Tool: T00 G code: G00 S: 6000 M: M03 M08 M33 M10 Machine coordinate:X+0208.987 Y+0050.996 Z+0020.000 A+0000.000 B+0000.000 C+0000.000 Processing time: 00:02:51 Number of processed part: 1 Status: Stop Handwheel increment:0.010 current axis: None
0 Backup 024	N Backup 025	G Backup 026	7 Backup 027	
8 Backup 028	9 Backup 029	X Backup 030	Y Backup 031	
Z Backup 032	4 Backup 033	5 Backup 034	6 Backup 035	
M Backup 036	S Backup 037	T Backup 038	1 Backup 039	
2 Backup 040	3 Backup 041	A Backup 042	F Backup 043	
EOB Backup 044	- Backup 045	0 Backup 046	. Backup 047	
H X axis 048	C Y axis 049	Switch Z axis 050	Input A axis 051	
Modify X axis 048	Delete C axis 053			
Basic input F4	Extended input F5			
Machine is OK				
⋘	Input	Output		

The above output points OT00---OT53 are corresponding to [O][N][G][7][8][9][X][Y][Z][4][5][6][M][S][T][1][2][3][F] [H][EOB][-][0][.][H][C][Shift][Enter][Modify] and [Delete] keys on the panel.

Output diagnosis methods:

Select manual mode → press [Diagnosis] F5 → select output diagnosis interface → press O P main axis positive rotation signal output (main axis positive rotation starts) → press O P main axis positive rotation signal stops (main axis positive rotation stops).

1.2.6 6. Alarm display

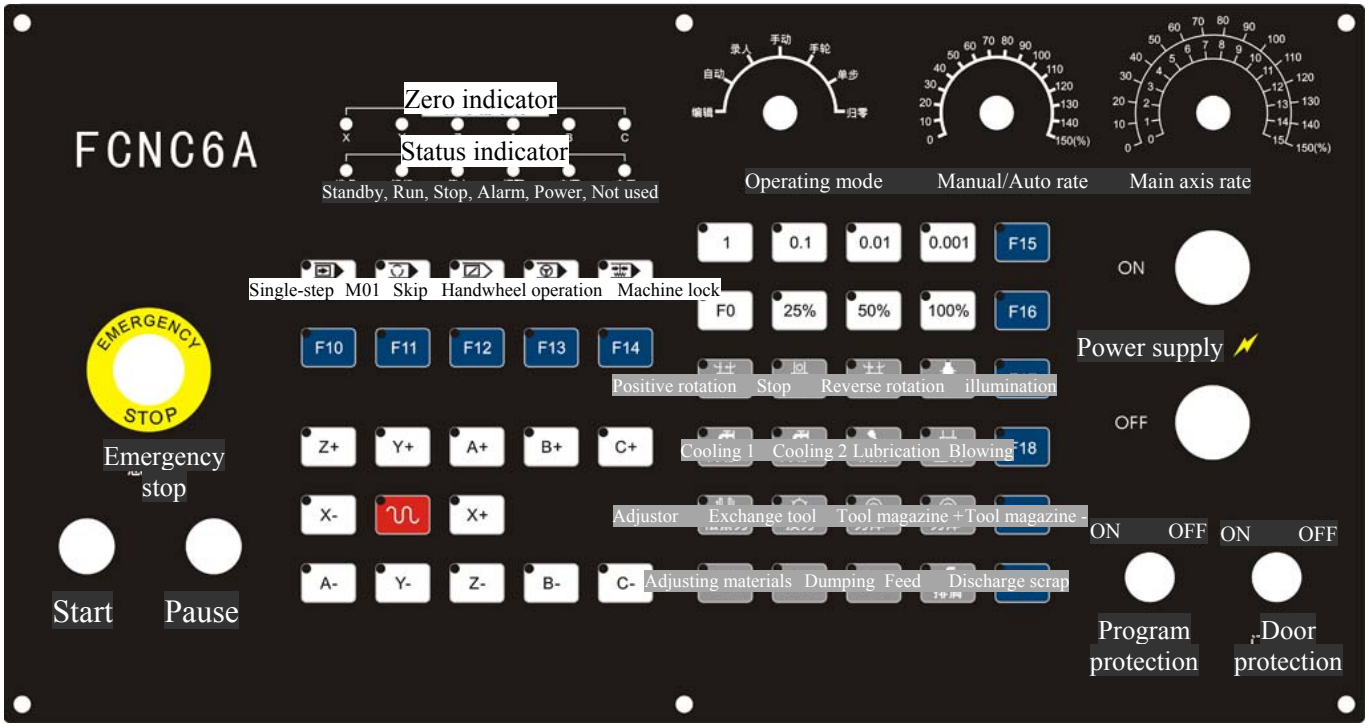
Alarm message	Manual mode	Program SN	3336	2007/05/09 09:40:09
1. Emergency stop 6. Z axis positive limit				Programming rate: 8000 Feed speed: 8000 Feed rate: 100% Manual speed: 1000 Manual rate: 100% Zero speed: 1000 Tool: T00 G code: G00 S: 6000 M: M03 M08 M33 M10 Machine coordinate:X+0208.987 Y+0050.996 Z+0020.000 A+0000.000 B+0000.000 C+0000.000 Processing time: 00:02:51 Number of processed part: 1 Status: Stop Handwheel increment:0.010 current axis: None
Machine is OK				

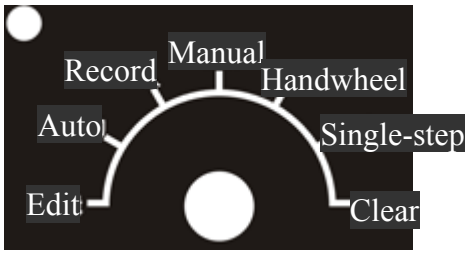
≪	Position	Program	Offset	Parameter	Diagnosis	Alarm	≫

III. Description of operating mode

1.2.7 1. Selection of operating mode

Additional panel



You can rotate the mode shift 

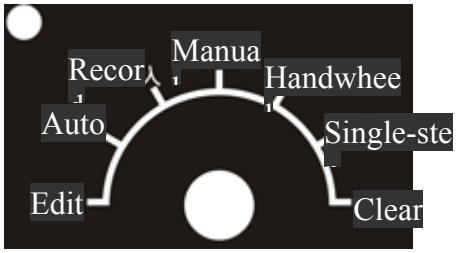
[Edit], [Auto], [Record], [Manual], [Handwheel], [Single-step], and [Clear] on additional panel to enter the corresponding operating mode.

S/N	Name	Purpose
1	[Edit]	edition of program, and the setting of system parameters, coordinate parameters, offset parameters, network parameters, setting parameters and administrative password
2	[Auto]	start/stop the processing code program, adjust the main axis speed and feed rate
3	[Manual]	manual operation of all axes, clear the relative coordinate, operation of auxiliary function and diagnosis function, adjust the manual rate
4	[Clear]	Program reset, machine reset
5	[Single-step]	increase value by single-step, move XYZABC coordinate axes
6	[Handwheel]	operate the handwheel to control XYZABC axes to move
7	[Record]	MDI operation, USB disk operation, serial port operation, and edition setting of parameter

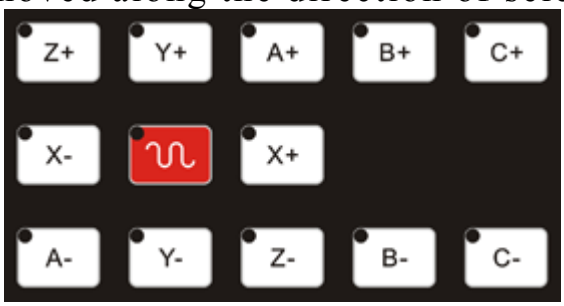
IV. Manual operations

1.2.8 1. Continuous manual feed operations

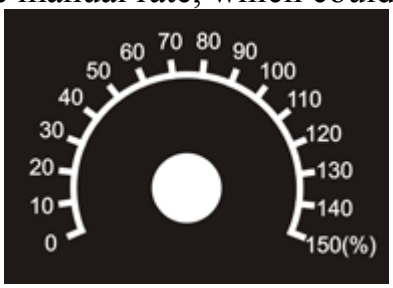
Rotate the mode shift to manual mode to select the manual operation mode.



Select X+, X-, Y+, Y-, Z+, Z-, A+, A-, B+, B-, C+, and C- axes. The machine tool is moved along the direction of selected axis.



In [Manual] mode, in interface displaying the position, rotate the manual rate shift to change the manual rate, which could be set between 0 and 150%.



1.2.9 2. reset relative position

Reset relative position: press X U, Y V, Z W or A B key on CNC4860 panel in the relative position interface in manual mode, the address of pressed key will become green and flash. Then, press 取消 CAN key, the corresponding position of the flashing address will be reset to 0. When you press X U, Y V, Z W or A B key again or change the interface, X, Y, Z, or A will not flash any more.

1.2.10 3. Operations of manual auxiliary functions

1 Tool change

In [Manual] mode, press 换刀 key and the tool rest will be rotated for changing for the next tool.

2 Main axis positive rotation

In [Manual] mode, press 正转 key and the main axis will be rotated in positive direction.

3 main axis stop

In [Manual] mode, press 停止 key and the main axis will be stopped.

4 main axis reverse rotation

In [Manual] or [Single-step/Handwheel] mode, press 反转 key and the main axis will be rotated in reverse direction.

5 Cooling 1, Cooling 2

In [Manual] mode, press any one of 冷却1 and 冷却2 keys and the corresponding cooling motor will be started. Press it again and the cooling motor will then be closed.

6 Lubrication

In [Manual] mode, press 润滑 key and the corresponding lubrication will be on while press it again the lubrication will be off.

Others, such as 照明 illumination, 膛吹 blowing, 松紧刀 adjustor, 刀库+ tool magazine +, 刀库- tool magazine -, 松紧料 adjusting materials, 排料 dumping, 送料 feeding, 排屑 chip cleaning, etc. are as the same as the lubrication operations. For detailed output control and the corresponding M code function, please refer to definition of output pin in <Connection>.

1.2.11

V. Auto operation

1.2.12 1. start of program

5.1.1 operating of storage

Select the program and auto mode, make sure it is in “Continuous” mode and the initial position is correct, then press [Reset] and [Start] keys, the program will start operating immediately. You can press [Stop] key to stop executing the processing program.

5.1.2 MDI operating

Select the recording mode, press [MDI] F2 key to enter MDI operating interface, input the words to be executed and press 插入 INSERT key. After entering the required data, press [Start] key and the program of MDI program segment will be executed. You can press [Stop] key to stop executing the processing program.

1.2.13 2. stop of auto operation

There are two ways to stop the auto operation. One is to input stop order in where it is to stop in advance via the program, and the other way is to use the button on control panel.

5.2.1 Program stops (M00)

After the program segment containing M00 is executed, the auto operation is stopped and the mode data are stored. After the reset, press [Start] key and the program will continue to execute.

5.2.2 Program ends (M30)

(1) represent the end of main program
(2) stop the auto operation and change to reset state
(3) return to the beginning of program

5.2.3 Pause

In auto operation, you can press [Pause] key on control panel to enable the auto operation to pause.

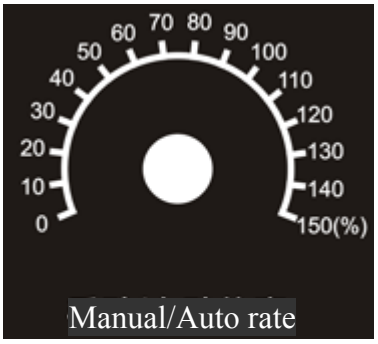
When the [Pause] key is pressed, the machine will have the following states.
1) when the machine is moving, the feed is decelerated and stopped.
2) when executing pause (G04), cease and pause
3)after executing the action of M, S, and T, stop
press [Start] and the program will continue.

5.2.4 Reset

Press 复位 RESET key, and the auto operation will be ended, the auxiliary function will be cancelled, and the cursor will return to the beginning of program while the state is changed to reset state. If reset is done during the operation, the machine will decelerate and stop.

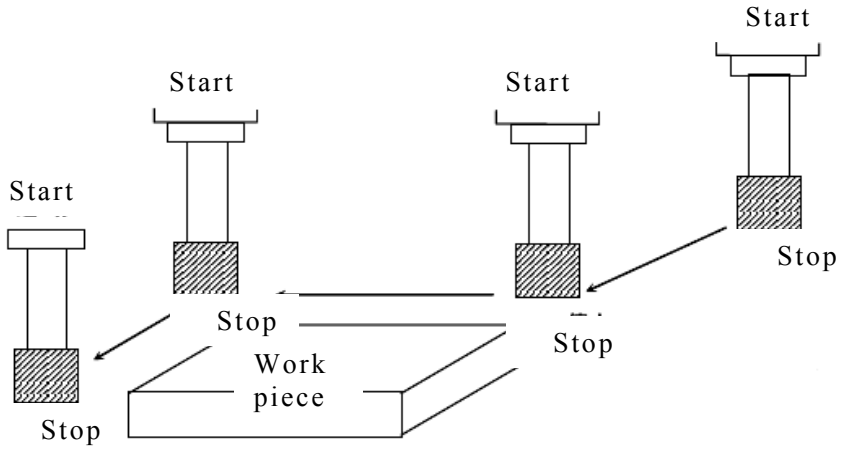
1.2.14 3. feed rate adjustment in auto operation

In [Auto] mode, in the interface of display position, you can rotate the auto rate shift to change the manual rate. The range of the rate is 0~150% (with 10% per shift) . The feed rate is specified by F instruction or parameters.



1.2.15 4. Single program segment

In auto mode, press 单段 key and the system will stop after executing the current program segment. If you press [Start] key again, it will execute the next program segment and stop. This allows you to check the program.



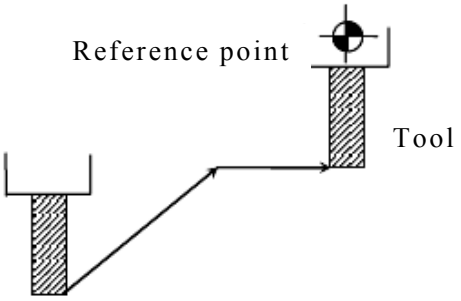
1.2.16 5. Skip the program segment

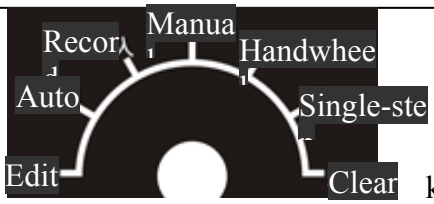
In auto mode, press 选跳 key and the indicator will be on. At this time, the system will skip the program segment automatically when it is executed to program segment with "/" and start executing the next one. This function is to make the program segment with "/" in program to be invalid.

VI. Zero fill

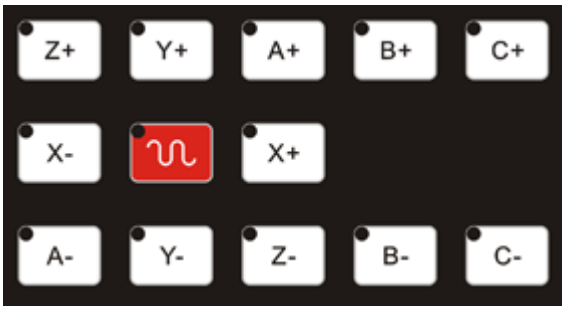
1.2.17 1. return to reference point manually

There is a special mechanical position on CNC machine tool, in where you can set the tool change and the coordinate system. This position is called as the reference point. Generally, when the power supply is connected, the tool is moved to the reference point first. Use the corresponding keys on control panel to move the tool to reference point, this operation is called returning to reference point manually.



Switch the mode shift 

key to reset mode. At this time, press X-, Y-, Z-, A-, B-, and C- keys, the corresponding axis will then return to the mechanical reference point.



Select the way of returning to zero, which will be different due to the difference of the value of parameter “Zero mode”.

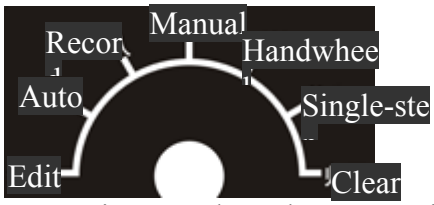
When the value is 0, it means there is no zero switch.

When the value is 1, it means there is one zero switch.

In addition, using the program instruction, it is also possible to make the tool to return to the reference point. This is called returning to reference point automatically.

VII. Single-step/Handwheel operation

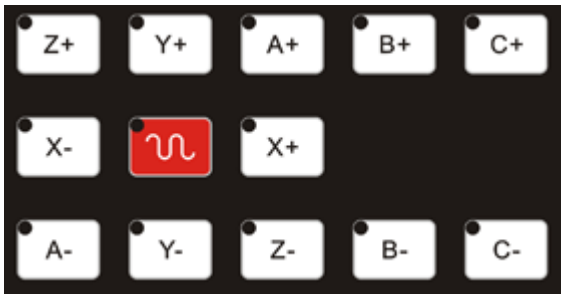
1.2.18 1. Single-step feed

Switch the mode shift 

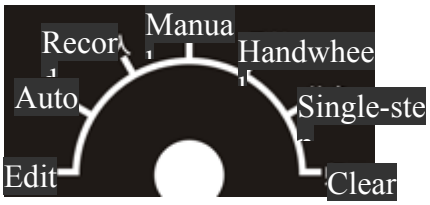
key to “Single-step” mode, and select [Single-step] operating mode. The operating mode column on LCD will show words “Single-step mode”.

Select the moving distance 

key, and press the manual axis-moving key. The axis moves once each time you press this key.

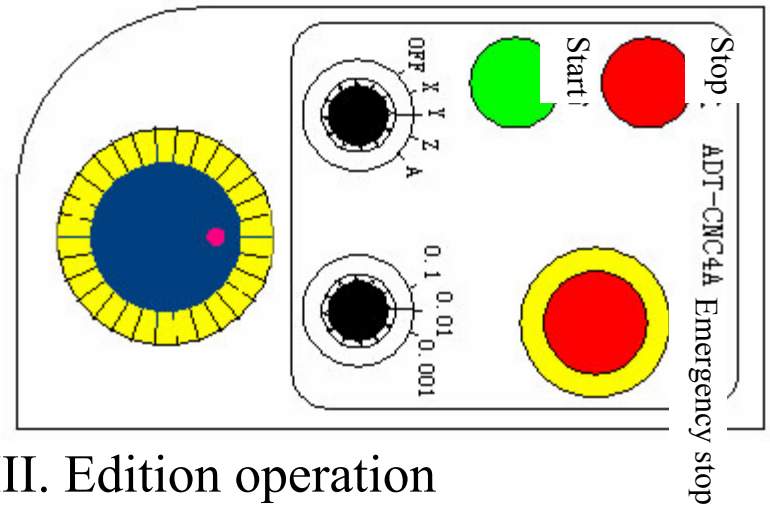


1.2.19 2. Handwheel feed



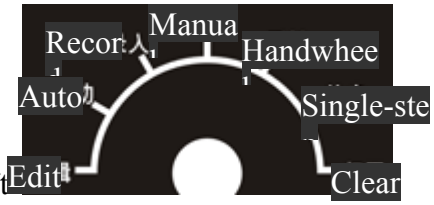
Switch the mode shift key to "Handwheel" mode and select the [Handwheel] operating mode. The operating mode column on LCD will show words "Handwheel mode".

Select the handwheel axis (Rotate the axis selection button on handwheel control box to select [X][Y][Z] or [A] axis shift, the selected handwheel axis will be displayed behind "Current axis". Rotate the moving distance selection button on handwheel control box to select [0.001][0.01] or [0.1] shift) and rotate the handwheel. Rotate it clockwise or anti-clockwise to select the moving direction.



1.3 VIII. Edition operation

1.2.1 1. Preparation before program storage and edition



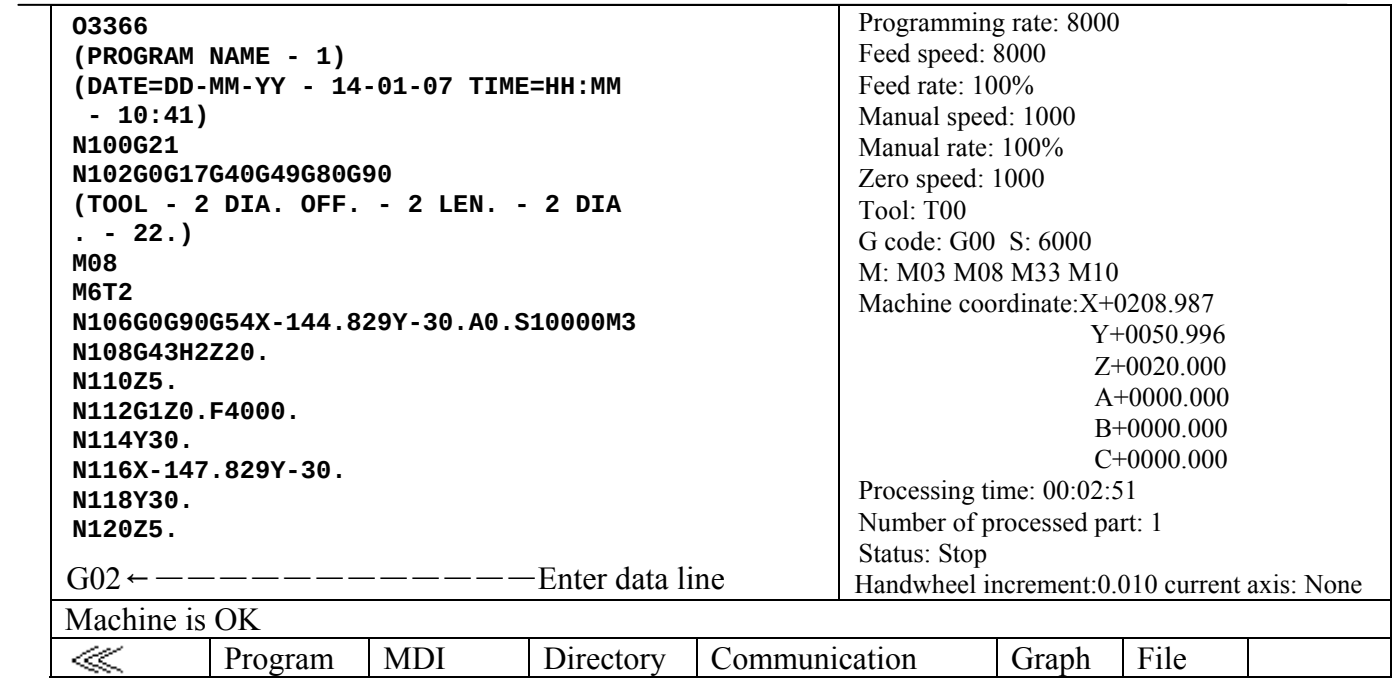
Switch the mode shift key to [Edit] mode, and you can then select the program interface to edit the program.

1.2.2 2. Save the program in storage

In [Edit] mode, press 程序 PROG or F1 key to select the program interface, use the keys to enter the address O P and program number, and then press 插入 INSERT key; by doing this, the program number is saved. After that, input every word of the program with keys, and then press 插入 INSERT key to save the input program.

Note: P, Q, R, U, V, W, B, J, K, L, D, E, and I keys are surrogate keys. You can insert it by pressing the shift key first.

Program editing	Edit mode	Program SN 3336	2007/05/09 09:40:09



1.2.3 3. Program searching

If the storage has stored many programs, when displaying the program, it always displays the program that the current program pointer points to. This program pointer would not loss even in case of power down. You can call the required program by searching and edit or execute the program, which is called program searching.

(1) Searching methods (Edit or auto mode)

Press address O P key, input the number of program that you want to search, and press cursor key ↓. If the input data is wrong, you can press 取消 CAN key to search again.

After the search, the searched program will be displayed on LCD and the program number will be shown on the top right corner of interface. If the required program is not found, an error prompt will be displayed.

(2) Scanning method

Press address O P and the cursor key ↓. When in edit mode, you can press address key O P and then press cursor key ↓ to display the saved program one by one.

1.2.4 4. Adding program

Press address O P and input the program number that you want to add, and press 插入 INSERT key, the corresponding new program segment will then be saved in the storage. If there is a repeated program, it will give an error prompt. The new added program can be edited, modified or deleted as the following method. The new program is as follows:

OXXXX

%

Note: XXXX is the corresponding program number. If there is repeated program, the system will give an error prompt.

1.2.5 5. Deleting program

Press address O P and input the program number that you want to delete, and press 删除 DELETE key, the corresponding program in storage will then be deleted. If the required program number is not found, the system will give an error prompt.

1.2.6 6. Deleting all programs

Delete all programs in the storage.

Press address key O P, input-9999 and press the delete key to delete all the programs.

1.2.7 7. Inserting, modifying, deleting word

The content of program saved in storage can be changed.

In [Edit] mode, select the program that you want to edit and search the word you want to edit. There are two methods as follows:

(A) by SCAN

(B) after searching the word, perform modification, insert and delete operations on word

8.7.1 Searching the word

(1) Scanning method

Scanning word by word

(A) Press cursor key ↓ or →, and the cursor will be moved word by word following the sequence direction, which means the system will show the cursor behind the address of selected word.

(B) Press cursor key ↑ or ←, and the cursor will be moved word by word following the reverse direction, which means the system will show the cursor behind the address of selected word.

(C) If you keep pressing ↓ → or ↑ ←, the system will move the cursor quickly automatically and continuously.

(D) Press 下页 PD key, the interface will be paged down and the cursor will be moved to the beginning of next page.

(E) Press 上页 PU key, the interface will be paged up and the cursor will be moved to the beginning of previous page.

(F) Press PD or PU key continuously, and the system will page down or up quickly and automatically.

(2) Methods to search words

Search the specified words from the position where the cursor is located.

(A) Input the address X U

(B) Input the word 1 , 0 * 0 * . 9) 2 # 5]

(C) Press cursor key ↓ to start searching downward, and press ↑ to start searching upward.

Note 1: If you input S1, you cannot search S12
Note 2: To search S09, you cannot search by just entering S9. You have to input S09.

(3) Methods of searching with address

Search the specified address at the sequence direction from the current position.

(A) by pressing address G R

(B) by pressing cursor key ↓; if you are pressing ↑, the system will search at the reverse direction.

(4) methods to return to the beginning of program

(A) Method 1: Press 复位 RESET key (select program interface in edit mode). When returning to the beginning, the system will display the content of program from the beginning.

(B) Method 2: input the program number by O P 1 0 0 1 and press ↓

(C) Method 3: press address key O P, and press cursor key ↑

8.7.2 Inserting word

Before searching the word that you want to insert, input the address and numbers that you want to insert and press 插入 INSERT key.

8.7.3 Altering word

When the word you want to alter has been searched, input the address and data to be altered and press 修改 ALTER key, the new input word will then replace the word that current cursor points to.

8.7.4 Deleting word

When the words to be deleted have been searched, press delete key and the current word will be deleted.

1.2.8 8. Storage capacity

(1) Capacity of memory program: total capacity 56M bytes, each working area of processing program has the maximum 20M bytes

(2) Tool compensation data: 48 groups, in which length compensation has 24 groups and radius compensation has 24 groups.

1.2.9 9. Download of program

(1) Download via USB communication: Use USB cable to connect the computer and CNC4860 controller, rotate the mode shift selection controller to <Record> mode -→press 程序 PROG →press <Catalog> F3 to

show 'File catalog' interface →press letter key G R (connect to computer) to enter the USB communication state. Find the folder "PRG" under the USB disk catalog and add the files to be processed in this folder. After the operation, disconnect the computer and controller, and press 取消 CAN key, the USB transmission program will then be ended. (Note: File name of OXXXX should not be repeated; otherwise, one of the files is taken as the processing file)

Note: The format of processing file should be xxxx.CNC (for example, 0000.CNC). For the content, it should be begun with OXXXX (X is the number from 0 to 9) and ended with %; otherwise, the download of the file cannot be received. For example:

O0001

G90G17G40G80

G0X0Y0Z0

G1X100Y100Z100

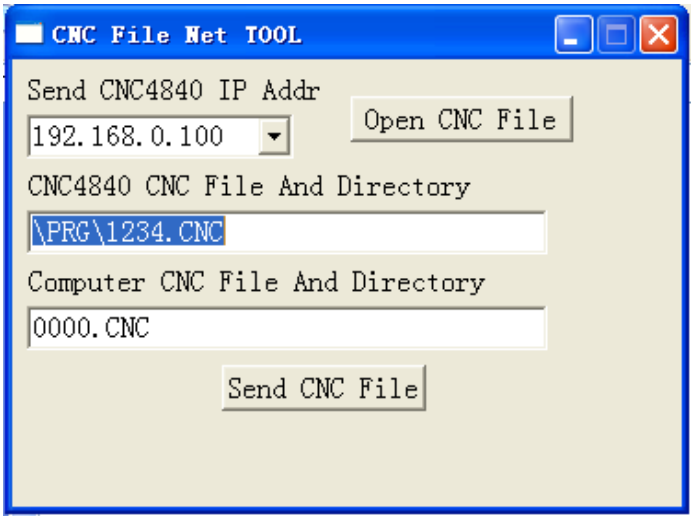
M30

%

(Note: File name of OXXXX should not be repeated; otherwise, one of the files is taken as the processing file）

(2)Download via network:

Install the software downloaded from the Internet on computer and then double click tftpclient.exe to open the software as follows:



Send CNC4860 IP Addr: It's the IP address of CNC4860 controller, which can be edited or selected from the drop-down text box. It is possible to add or reduce the item of IP address by editing IpAddr.ini file in the catalog of tool downloaded from Internet. IpAddr.ini file can be opened and edited by any text editor, and the editing format is as follows:

192.168.0.100

192.168.0.90

192.168.0.106

192.168.0.120

Just edit the IP of every CNC4860 controller in accordance with the above format in IpAddr.ini file.

CNC4860 CNC File And Directory: the directory and file name of CNC files that sent to CNC4860 controller (note: the directory file is \PRG\xxxx.CNC, x is number or character, for example 0000.CNC 0001.CNC).

Computer CNC File And Directory: the file directory and name of CNC processing files in local computer that sent to CNC4860 controller (Note: processing file can be selected from the computer directory by clicking <Open CNC File> button); if the CNC processing file in local computer does not have directory, the system will search for the file from the directory of network download software.

Send CNC File: Click this button to transmit the CNC processing file (Note: this file should exist) (selected in the third step) to lower CNC4860 controller, but the IP address should be corresponding to that of relevant CNC4860 controller.

IX. Recording operation

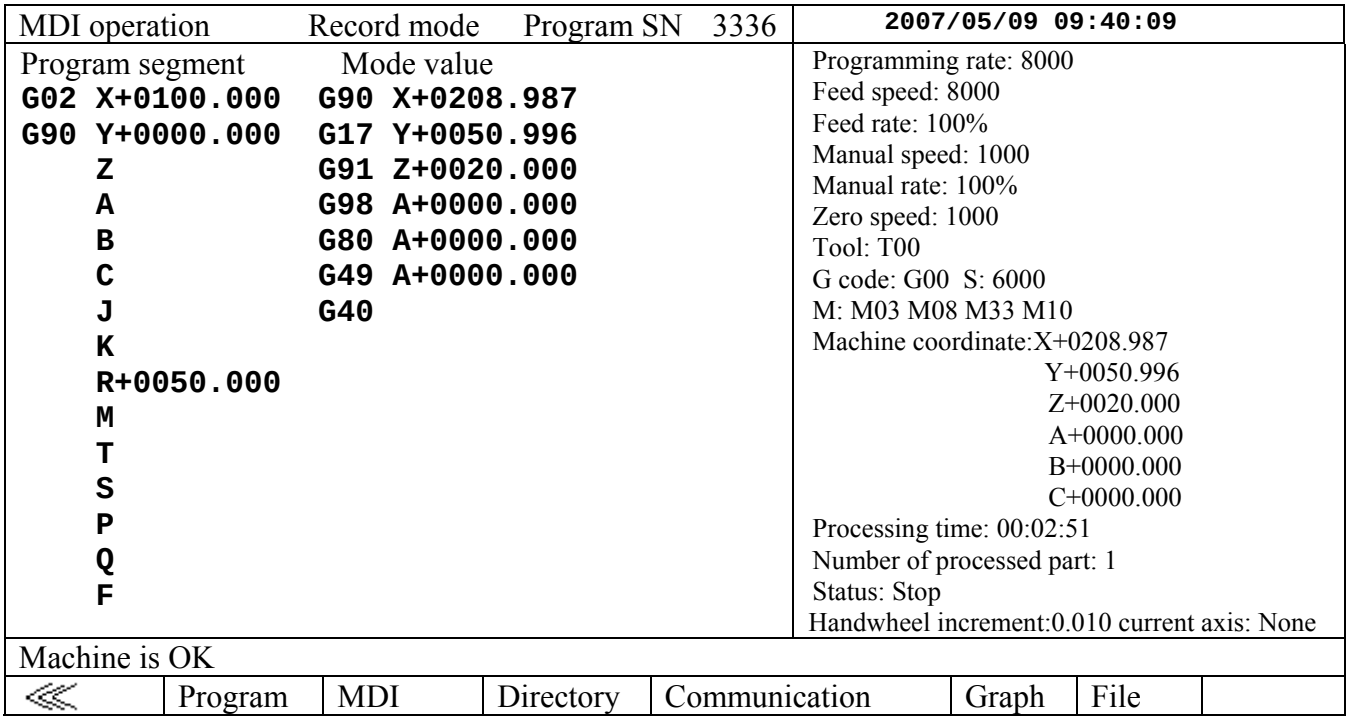
MDI operation

In [Record] mode, press [MDI] 

key to enter the MDI interface, input the words to be executed and press [Insert] 

. After enter all required data, press number key 7 

or [Start], the system will then run the program segment. When the operation is stopped, it can also continue executing the MDI operation.



X. Composite key

When operating without additional panel, the setting parameters allow the key functions of additional panel to be achieved on control panel. These keys have the functions of additional panel only when in special operation mode and display mode. the composite function of key would not affect the original functions. The following composite keys have the same function as the corresponding key on additional panel.

(Note: the composite key can be used only in position display mode and the operations of key are the same as the corresponding key on additional panel.

List of composite key

Operation mode	Display mode	Composite key	Function
[Auto]	[Position]	插入 INSERT	Switching between Single Segment and Continuous
		7	Program started
		8	Program stopped
		↑ ↓	increase and decrease of feed rate

[Manual]	[Position]		Move at X axis + direction
			Move at X axis - direction
			Move at Y axis + direction
			Move at Y axis - direction
			Move at Z axis + direction
			Move at Z axis - direction
			Move at A axis + direction
			Move at A axis - direction
			Move at B axis + direction
			Move at B axis - direction
			Move at C axis + direction
			Move at C axis - direction
			adjust the manual rate
			adjust the main axis rotation speed
			M08/M09 switching
			M32/M33 switching

		EOB	M3 (main axis rotated normally)
		F L	M4 (main axis rotated reversely)
		M I	M5 (main axis stops)
[Clear]	[Position]	6 SP	X axis returned to mechanical zero
		2 #	Y axis returned to mechanical zero
		3 =	Z axis returned to mechanical zero
		9)	A axis returned to mechanical zero
		0 *	B axis returned to mechanical zero
		.	C axis returned to mechanical zero
[Single-step]	[Position]	F L	Single-step increment 1.000
		H D	Single-step increment 0.100
		C E	Single-step increment 0.010
		− +	Single-step increment 0.001

Annex 1 Specification list

Function	Name	Specification
Control axis	Control axis	6 axes (X, Y, Z, A, B, C)
	Number of axis controlled simultaneously	4 axes for linear interpolation, 2 axes for circular interpolation
Input instruction	Minimum setting unit	0.001mm
	Minimum move unit	0.001mm
	Maximum instruction value	±9999.999 mm
Feed	Quick feed speed	X axis, Y axis, Z axis, A axis: 9999mm/minute (maximum value)
	Range of feed speed, feed per minute feed per turn	1~9999mm/minute
		0.0001~500.0000 mm/turn
	auto acceleration/deceleration	Yes
	Feed rate	10~150%
Manual	Manual continuous feed	Yes
	Return to reference point manually	Single axis/several axes return to reference point at the same time
	Single-step/handwheel function	Yes
Interpolation	Positioning, linear interpolation, circular interpolation	G00,G01,G02/G03
Adjusting function	Trial operation, single program segment	Yes
Coordinate system and pause	Pause (s/microsecond)	G04 X/P_
	Settings of coordinate system	G92
	Auto setting of coordinate system	Yes
Operating mode	MDI, auto, manual, single-step, edit	Yes

Safety function	Software limit check	Yes
	Emergency stop	Yes
Program storage and edition	Storage capacity, numbers of stored program	Total capacity 32M bytes, 512 working areas, each of which has 4M bytes, capable of storing 10000 programs
	Program editing	Insert, modify, delete, cancel
	Program number, order number, address, word searching	Yes
	Decimal programming	Yes
Display	320×240 dot matrix 5.7” color LCD display	
	Position, program, tool compensation, parameter, diagnosis, alarm, figure, etc.	Yes
M, S, T function	Auxiliary functions	M2 digit
	Main axis function	S2 digit (shift) S3-5 digit (analog)
	Tool function	T01~18
Compensation function	tool compensation storage	±7 bit, 36 groups
	Backlash Compensation	Yes
Other functions	specification of circular arc radius R	Yes
	Gear ratio	Yes
	starting program in any position	Yes

1.2.10 Annex2. Parameter list

1--12: setting of gear ratio

X axis CMR (X_CMR)

X axis CMD (X_CMD)

Y axis CMR (Y_CMR)

Y axis CMD (Y_CMD)

Z axis CMR (Z_CMR)

Z axis CMD (Z_CMD)

A axis CMR (A_CMR)

A axis CMD (A_CMD)

B axis CMR (B_CMR)

B axis CMD (B_CMD)

C axis CMR (C_CMR)

C axis CMD (C_CMD)

When the leading screws of different pitches are matched with motors of different step angles or servo motors of different pulse counts, or combined using different change gears, it is possible to allow the programming to be consistent with the actual moving distance by setting the gear ratio.

CMR/CMD =P/(L×1000)

CMR: Command multiplication ratio

CMD: Command frequency division factor

P: pulse per turn of motor

L: Moving length of machine tool per turn of motor (mm)

The value of CMD/CMR is actually the pulse equivalent, which is the corresponding moving distance of each pulse, unit in 0.001mm.

Example 1: the motor rotates a circle every 5000 pulses, and the machine tool moves 5mm every time when the motor rotates a circle, that is

CMR/CMD=5000/ (5*1000) =1/1

In this way, it is possible to set CMR=1, CMD=1. The pulse equivalent is 0.001mm.

Example 2: the motor rotates a circle every 5000 pulses and the machine tool moves 10mm every time when the motor rotates a circle, that is

CMR/CMD=5000/ (10*1000) =1/2

In this way, it is possible to set CMR=1, CMD=2. The pulse equivalent is 0.002mm.

Unit: None

Range:1-9999

13--18: Quick-move rate of X, Y, Z, A, B, and C axes

This speed is the speed used by G00 instruction.

Unit: mm/minute

Range:1-9999

19--24: Initial rate of X, Y, Z, A, B, and C axes

It is the initial speed when accelerating and decelerating, used in GOO instruction, and manual and zero motion.

When using stepper motor, it is recommended to adopt a speed of 1-2 revolutions for the motor. The machine tool moves 5mm every time when the motor rotates a circle, 1-2 revolutions per second is 5-10mm/s, converted to mm/minute, the initial speed can be set as 300-600mm/minute.

For servo motor, it is best to have no vibration when it is started and stopped. Too high the speed may produce vibration when operating and cause the stepper motor to loss step.

Unit: mm/minute

Range:1-9999

25--30: Acceleration of X, Y, Z, A, B, and C axes

It is the parameter used in acceleration and deceleration, which refers to the time required from the initial rate to quick-move rate when accelerating. The same as that in deceleration. It is used in GOO instruction, and manual and zero motion.

For stepper motor, the acceleration is recommended to be 500mm/s. If it's faster than 500mm/s, it may cause losing step. Too slow the acceleration will extend the acceleration time, which cannot achieve the maximum speed.

For servo motor, the acceleration can be set higher; however, too high the acceleration would increase the vibration.

Unit: mm/s

Range:1-9999

31--38: Soft limit

X axis positive soft limit, X axis negative soft limit, Y axis positive soft limit, Y axis negative soft limit, Z axis positive soft limit, Z axis negative soft limit, A axis positive soft limit, A axis negative soft limit

Generally, there is hard limit signal on machine tool, it is not necessary to use the soft limit. Set the positive limit as +9999.999, and negative limit as -9999.999.

If hard limit switch is not installed, software limit can be used, which takes machine coordinate system as the base point. The positive limit and negative limit are subject to the actual distance, unit in mm.

For software limit, it is decelerated and stopped when reaching the limit point, it may exceed the set distance a little, which is related to acceleration time and speed. Therefore, it is essential to set the parameter with a certain allowance.

Unit: mm

Range:-9999.999 — +9999.999

39: Feed rate

Commands, such as G01, G02 and G03, are operated following the speed of F instruction. If F instruction is not specified in the program, the above commands are moved at speed set by the parameter. If F instruction is specified, this parameter does not work.

Unit: mm/minute

Range:1-9999

40: Initial speed of feed

During the feeding, acceleration and deceleration operations are also taken. The meaning is almost the same as the initial speed of every axis, except that it is acted on feed instruction.

Unit: mm/minute

Range:1-9999

41: Acceleration of feed

During the feeding, acceleration and deceleration operations are also taken. The meaning is almost the same as the initial speed of every axis, except that it is acted on feed instruction.

Unit: mm/s

Range:1-9999

42--47: Backlash compensation

Backlash compensation of X, Y, Z, A, B, and C axes

Unit: Pulse

Range:0 — 1000

48: Zero returning mode

0- without origin switch

1- one origin switch

2- one speed reducing switch + one origin switch

49: Zero returning speed

It's the speed of axis when returning to zero in zero returning mode.

Unit: mm/minute

Range:1-9999

50: Manual speed

It's the reference speed of each axis in manual mode.

Unit: mm/minute

Range:1-9999

51: Maximum feed rate

To ensure that the wrong F instruction would not damage the machine, and according to the processing capability of machine tool, set the feed speed as the maximum value. When the product of F instruction and feed rate is larger than this value, confine the speed within this range.

Unit: mm/minute

Range: 1-9999

52: Numbers of system tool

Set the numbers of tool used on machine

Unit: Set

Range:1-32

53: M code waiting time

Unit: millisecond

Range:1-9999

54--59: Coordinate settings when X, Y, Z, A, B, and C axes are returning to reference point

The meanings of parameters are explained in details in tool setting description.

Unit: mm

Range:-9999.999 — +9999.999

60: Line number increment

It is used in programming the program, 0 represents prohibiting generate the line number automatically.

Unit: None

Range:0-100

61: Maximum speed of main axis

It's the revolutions of motor when PWM exports 10V voltage.

Unit: revolutions per minute

Range:1-20000

62: Line number of main axis coder

When using main axis position coder, this value refers to the line number of coder, which means how many pulse each revolution produces.

Unit: line

Range:100-99999

63--68: Zero direction of X, Y, Z, A, B, and C axes

It's used to set the zero direction of X, Y, Z, and A axes.

Unit: None

0---- returning to zero at positive direction

1---- returning to zero at negative direction

69: Feed of linear and circular interpolation

Subdivision for setting linear or circular interpolation is default as 0.2.

Unit: mm

Range:0.0--1.0

Note: This value should not be set too big. It's generally set between 0.1 and 0.5. It can also be set as 0.

70: G73 Retract amount in fixed cycle

Unit: mm

Retract amount in fixed cycle G73 instruction

71: G83 Retract amount in fixed cycle

Unit: mm

Retract amount in fixed cycle G83 instruction

72: Acceleration value of linear and circular interpolation

Unit: None

Acceleration value of linear and circular interpolation

73: Interpolation speed mode

Unit: None

0: Interpolation acceleration/deceleration

1: Interpolation constant speed

74: Code processing mode

Unit: None

Default as 0

75--80: Handwheel reference speed of X, Y, Z, A, B, and C axes

It's the reference speed of axis in handwheel mode.

Unit: mm/minute

Range:1-9999

81--83: A1--A3 axis zero offset

Unit: mm

It's the calibration value after ABC axes returning to zero, in order to ensure the level of worktable

84: Tool-changing speed

It's the operating speed of machine tool when executing the tool-changing instruction, which is determined according to the actual condition by the manufacturer.

Unit: mm/minute

Range:1-9999

85: Space between tools

Center distance of any two tools

Unit: mm

Range: 0 - 9999.999

86: Up-move value of tool changing

Z axis coordinate in horizontal tool magazine when main axis is moved horizontally, which is also called safety height of tool changing

Unit: mm

Range: -9999.999 - +9999.999

87: X axis reference point for tool changing

X absolute coordinate of the first tool in tool magazine; regardless of horizontal tool magazine or vertical tool magazine

Unit: mm

Range: -9999.999 - +9999.999

88: Y axis reference point for tool changing

Y absolute coordinate of the first tool in tool magazine; regardless of horizontal tool magazine or vertical tool magazine

Unit: mm

Range: -9999.999 - +9999.999

89: Z axis reference point for tool changing

Z axis coordinate in horizontal tool magazine when main axis is moved horizontally, which is also called safety height of tool changing, or the Z axis coordinate when the main axis is ready to put the tool back the tool magazine

Unit: mm

Range: -9999.999 - +9999.999

90: Tool placing coordinates of Z axis

It's the Z axis coordinate when the main axis releases the tool during the tool changing, used in vertical tool magazine

Unit: mm

Range: -9999.999 - +9999.999

91: Tool fetching coordinates of Z axis

It's the Z axis coordinate when the main axis is ready to fetch the tool during the tool changing, used in vertical tool magazine

Unit: mm

Range: -9999.999 - +9999.999

92: Oil pumping mode of lubricating oil pump

Oil pumping mode of lubricating oil pump

Unit: None

0: oil pumping mode of non-inching lubricating oil pump

1: oil pumping mode of inching lubricating oil pump

93: Switching frequency of inching lubricating oil pump

Unit: millisecond

Range:10000 – 0

94: Dry run processing speed of handwheel

Used to set the speed of single axis motion and interpolation motion of each axis during the dry run code processing of handwheel (default value: 500)

Unit: mm/minute

Range:0 – 10000

95: Baud rate of serial port communication

Used to set the rate (default value: 115200b/s) of serial port communication between controller and the upper software; it should be consistent with the baud rate of serial port set by upper computer software; reference value (9600, 38400, 115200) etc.

Unit:b/s

Range:0---200000

96: Baud rate of serial port communication

Used to set the address number for controlling the serial port communication with the upper computer

Unit: None

Range:1---255

Note: Parameters that are not described in parameter list should not be modified, just use the default value.

1.2.11 Annex 3 List of alarm message

Message	Meaning	Solutions
Emergency stop	Emergency stop button is pressed	resume the emergency stop switch
X axis limit	X axis positive limit or negative limit effective	Move the X motor reversely in manual mode
Y axis limit	Y axis positive limit or negative limit effective	Move the Y motor reversely in manual mode
Z axis limit	Z axis positive limit or negative limit effective	Move the Z motor reversely in manual mode
A axis limit	A axis positive limit or negative limit effective	Move the A motor reversely in manual mode
X axis over travel	X axis exceeds the range of software limit	Move the X motor reversely in manual mode
Y axis over travel	Y axis exceeds the range of software limit	Move the Y motor reversely in manual mode
Z axis over travel	Z axis exceeds the range of software limit	Move the Z motor reversely in manual mode
A axis over travel	A axis exceeds the range of software limit	Move the A motor reversely in manual mode
X axis alarm	X axis driver alarm	Check the X driver
Y axis alarm	Y axis driver alarm	Check the Y driver
Z axis alarm	Z axis driver alarm	Check the Z driver
A axis alarm	A axis driver alarm	Check the A driver

1.2.12 Annex 4 List of error code

Error code	Meaning
Please reset	It's the exit when it is not finished during the processing, just press the [Reset] key. To ensure the processing, the machine tool should be at the correct initial position.
1- no enter sign in the end of data	There should be an Enter in the end of data
2-the number of characters in one line exceeds 255	the number of characters in one line should not exceed 255
3- started with illegal character	Non-standard code character is not allowed
4-G code or M code format error	G code or M code should have 1-2 digits
5- non-supported G code	Refer to list of G code in user manual
6- non-supported M code	Refer to list of G code in user manual
7- no figure behind the characters	No data behind the address code
8- code repeated	X and U, or Z and W are used at the same time
9-T code format error	Refer to description of T code
10-S code format error	Refer to description of S code
11-	
12-sub-program number error	the format of P code behind M98 is wrong
13- sub-program number not found	The specified sub-program number is not found
14-sub-program call exceeds 9 layers	May be the sub-program calls itself, modify the program
15-G7X-G8X code format error	Refer to description of G7X-G8X code
16-G7X8X code format error	
17-H code format error	
18-radius is not specified in the circular interpolation	
19-radius error in the circular interpolation	

20- G28, G29 are specified in fixed cycle mode	
21- axis outside the appointed plane is instructed in circular interpolation	
22-the compensation offset number selected by H code is too large	
23-Feeding amount Q of G73 and G83 instruction is smaller than 0	
24-delay time P is not specified in G7X and G8X instructions	
25- R plane value in hole processing cycle G7XG8X is larger than the initial plane	
26- In radius compensation, instruct compensation start or logout in G2G3	
27- Compensation value of radius compensation is larger than R value of G0203 programming	
28- M98/M99 is specified in tool radius compensation	

1.2.13 Annex 5 List of G functions

G code	Group	Function
G00	01	Positioning (quick move)
G01		Linear interpolation (Cutting feed)
G02		Circular interpolation CW (Clockwise)
G03		Circular interpolation CCW (Anti-clockwise)
G04	00	Pause, stop
G17	02	XY plane selection
G18		ZX plane selection

G19		YZ plane selection
G20	06	Imperial data input
G21		Metric data input
G28	00	Return to reference point
G29		return from reference point
*G40	07	Tool radius compensation cancelled
G41		Left tool radius compensation
G42		Right tool radius compensation
G43	08	Tool length offset at positive direction
G44		Tool length offset at negative direction
*G49		Tool length offset cancelled
*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
G65	00	Macro command
G73	09	Fixed cycle of deep-hole drilling
G74		Fixed cycle of left-hand thread tapping
G76		Fixed cycle of fine boring
*G80		Cancel fixed cycle
G81		Fixed cycle of drilling
G82		Fixed cycle of drilling
G83		Fixed cycle of deep-hole drilling
G84		Fixed cycle of tapping
G85		Fixed cycle of boring
G86		Fixed cycle of boring
G87		Fixed cycle of reverse boring
G88		Fixed cycle of boring

G89		Fixed cycle of boring
*G90	03	Absolute value programming
G91		Increment programming
G98	10	return to initial plane in fixed cycle
G99		return to R point plane in fixed cycle

1.2.14 Annex 6 setting of workpiece coordinate and tool setting

When using CNC4860 controller, it is recommended to install the machine zero point switch.

After returning to the zero point, the workpiece coordinate system 1~6 are established. Select G54 coordinate system (workpiece coordinate system 1) when starting the machine. The absolute value of position interface is the coordinate value in the current coordinate system.

(Note: it's not necessary to use G92 to set coordinate system when the functions of workpiece coordinate are selected. If G92 is used to set the coordinate system, it may move the coordinate system 1~6. Do not use G92 together with G54~G59, unless you are going to move the workpiece coordinate system G54~G59)

1. Methods for tool setting with machine zero point

First, make sure "Zero mode" is set as 1. Use [Zero Mode] to make XYZA axes to return to the reference point, at this time the machine tool coordinate of position (comprehensive position) is 0. After returning to the reference point, select a tool from tool holder and take the tool tip as the starting point of program. In [Manual mode], press X+ or X-, Y+ or Y-, Z+ or Z-, A+ or A-, B+ or B-, C+ or C- to move to the starting point of workpiece (zero point of workpiece coordinate system). Press 参数 SYSTEM or F1 -→and then press F2 key to switch to coordinate parameter interface (the value of machine position X, Y, Z, A, B, and C under the interface is the mechanical coordinate value of current point of tool). press ↑ and ↓ key to allow the cursor to move to G54, G55……G59, select the relevant X, Y, Z, A, B, C workpiece coordinate, select [Record] and press EOB key, you can then set the mechanical coordinate value of current point of tool as the origin of workpiece coordinate system.

If the offset of origin of workpiece coordinate system to the origin of machine coordinate system is known, you can input the value directly in the setting interface of workpiece coordinate system and then press 插入 INSERT key to set the origin of workpiece coordinate system.

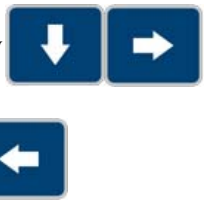
1.2.15 Annex 7 Table of operating environment

Operating temperature	0°C— 45°C
Optimum operating temperature	5°C— 40°C

Operating humidity	10%——90% No condensation
Optimum operating humidity	20%——85%
Storage temperature	0℃—50℃
Storage humidity	10%——90%

1.2.16 Annex 8 **Description of keyboard**

The keyboard of CNC4860 is divided into two areas, edition area and operation area. The following table is the description of all keys:

S/N	Name	Purpose
1		having different functions in different display modes
2		To insert a EOB when the input buffer storage has no data, and confirm the modification of parameter in parameter interface
3	Address  /Figure 	Enter letter and number
4	Cursor key 	There are two ways of cursor movement: ↓→: move down the cursor a subdivision; ↑←: move up the cursor a subdivision. Keep pressing the cursor key allows the cursor to move continuously. At the same time, ←→ can be used as search key.
5	[Position] 	Select [Position] interface
6	[Program] 	Select [Program] and [File management] interfaces
7	[Offset] 	Select [Too compensation] interface
8	[System] 	Select [System] interface

9	[Message] 信息 MESSAGE	Select [Message] interface
10	[Graph] 图形 GRAPH	Select [Graph] interface
11	[Insert] 插入 INSERT	insert the character or sign in buffer storage into the program
12	[Cancel] 取消 CAN	Clear the character or sign entered into the input buffer storage. For example, when the buffer storage is displayed as N001, press [Cancel] key and the N001 will be cancelled.
13	[Alter] 修改 ALTER	to modify the program or field during the edition
14	[Shift] 切换 SHIFT	to switch between upper and lower shift
15	[Input] 输入 INPUT	Not available
16	[Delete] 删除 DELETE	delete a current subdivision
17	[Help] 帮助 HELP	show the operating function descriptions of composite function keys
18	[Reset] 复位 RESET	Clear the alarm, CNC reset

1.2.17 Annex 9 Download of program

1. Download via USB communication: Use USB cable to connect the computer and CNC4860 controller, select <Record> mode on controller-→press 程序 PROG →press F3 to switch to the 'File catalog' interface →press <G> G R (connect to computer) to enter the USB communication state. Find the xxxx.CNC file such as 0000.CNC in folder "PRG" under the USB disk catalog, open it with text editor and add the processing files started with OXXXX (X is the numbers from 0 to 9) and ended with % to this folder. After the operation, disconnect the computer and

controller, and press 取消 CAN key twice and the <Position> 位置 POS to return to position interface.

Note: The content of file should be begun with OXXXX (X is the number from 0 to 9) and ended with %; otherwise, the download of the file cannot be received. For example:

O0001

G90G17G40G80

G0X0Y0Z0

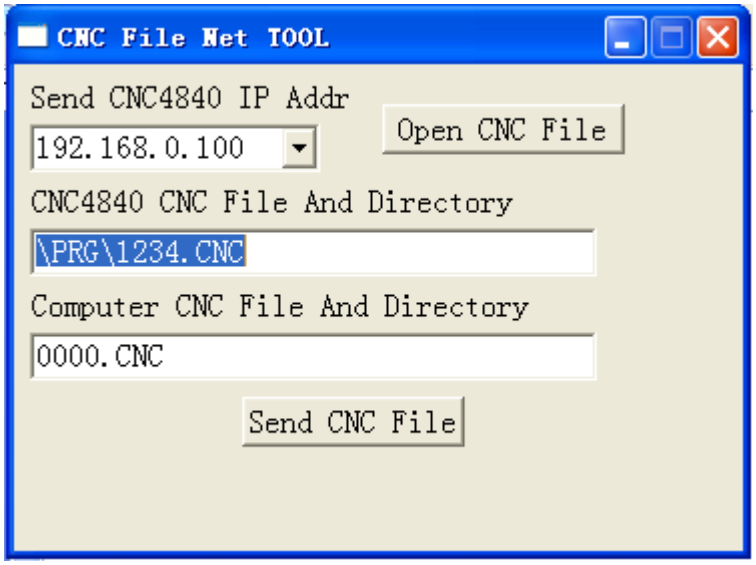
G1X100Y100Z100

M30

%

2. Download via network:

Install the software downloaded from the Internet on computer and then double click tftpclient.exe to open the software as follows:



Send CNC4860 IP Addr: It's the IP address of CNC4860 controller, which can be edited or selected from the drop-down text box. It is possible to add or reduce the item of IP address by editing IpAddr.ini file in the catalog of tool downloaded from Internet. IpAddr.ini file can be opened and edited by any text editor, and the editing format is as follows:

192.168.0.100

192.168.0.90

192.168.0.106

192.168.0.120

Just edit the IP of every CNC4860 controller in accordance with the above format in IpAddr.ini file.

CNC4860 CNC File And Directory: the directory and file name of CNC files that sent to CNC4860 controller (note: the directory file is \PRG\xxxx.CNC, x is number or character, for example 0000.CNC 0001.CNC).

Computer CNC File And Directory: the file directory and name of CNC processing files in local computer that sent to CNC4860 controller (Note: processing file can be selected from the computer directory by clicking <Open CNC File> button); if the CNC processing file in local computer does not have directory, the system will search for the file from the directory of network download software.

Send CNC File: Click this button to transmit the CNC processing file (Note: this file should exist) (selected in the third step) to lower CNC4860 controller, but the IP address should be corresponding to that of relevant CNC4860 controller.

3. Download via serial port (Instructions for File Manager with Comm):

1. Operating environment of software: For the hardware, Pentium+ or compatible processing chip, 800*600/1024*768 compatible display, with standard UART serial port (three-wire system)

2. Installation instructions of supported file *.OCX before using the software: Run install.bat batch file in directory \Support\, and install the registration for executing the file *.OCX.

3. User manual for software

1. After the registration of *OCX file, run File Manager with Comm.exe in directory \Support\, and the following interface will be shown. (Note: You can also create a shortcut on desktop, and you can then run the software by clicking the shortcut.)

2. Pop up system initiation interface

3. Pop up main interface of program (as follows)

A. Select the COM1 or COM2 port according to the wiring, and select the baud rate in accordance with the device , and then open the port.

B. Double click the device number in device list, and the corresponding catalog list will be shown. Double click the catalog list to pop up the corresponding file list.

C. In file list, you can right click to add, delete, upload or download the file.

D. The buttons at the left side allow you to open the local file, save the local file or save file as the local file.

E. You can save the file uploaded in step C as local file following the step D, or download the file opened in step C.

F. This software supports multi-machine online file management.