ADT-CNC4840
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:86-755-26722719 Fax: 86-755-26722718

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

Copyright Statement

Version Upgrading Instruction

Program NO	First update	Version Number	Total page	Edit engineer	Printer engineer
XT20080505B	2012-1-7	A1301	194	Xu Yuwen	

calibration records

Date	version/page	result	confirmation

Notice

Transportation and storage:

1.The product packing case piles the repeat not to be possible to surpass six 2.Cannot climb up in the product packing case, stand or the laying aside heavy

3. cannot use and the product connected electric cable dragging or the transporting product

4. refuses the collision, to scratch the kneading board and the display monitor

5. product packing case should avoid moist, the insolation as well as the rain drenches

Opens a box and check :

1. after opening the packing, please confirm whether is the product

2. inspection product which you purchase in the transportation whether to have on the way damages

3. comparison detailed list to confirm various parts are whether complete, whether there is appendix or transportation damage situations and so on to damage

4. like existence product model symbol, not to lack, please promptly relate with Our company

Connection:

1. participates in the wiring and the inspection personnel must be has the corresponding ability specialists

2. product to earth reliably, the earth resistance should be smaller than 4 ohms, cannot use the neutral axis (zero curve) to replace the grounding

3. wiring to be correct, be reliable, in order to avoid causes the product breakdown or the unexpected consequence

4. with the product connection surge absorber diode must according to the stipulation direction connection, otherwise before will damage product

5. to insert pulls out the plug or turns on the product engine case, must shut off the product power source

Overhaul

1. before the overhaul either replaces the primary device, must the dump

2. have when the short circuit or the overload should the trouble shooting, after the trouble shooting, if only then starts

3. not to be possible passes the power failure frequently to the product, after the power failure, to electrify, time interval at least 1 minute

Others

1. without the permission, please arbitrarily do not turn on the cabinet.

2. the long time does not use, please dump.

3. the special attention do not let the dust, the powdered iron enter the controller.

4. outputs the relay, if uses the non-solid state relay, then must in the relay winding the parallel after flow diode. The inspecting office receives a

telegram the source whether to meet the requirement, ceases burns out the controller.

5. controller's life and the ambient temperature have the very big relations, if processes the scene hyperpyrexia, pays respects installs the radiation ventilator. Controller permission work ambient temperature scope in 0℃-60℃ between. 6. avoids, in the high temperature, moist, the multi-dust or have in the caustic gas environment to use. 7. in the vibration intense place, should add the rubber crash pad to carry on the cushion.

Maintenance

Under general exploitation conditions (environmental condition: The daily average 30℃, the load factor 80%, the service factor daily 12 hours), please press the following project to carry on the daily inspection and the periodic inspection.

Daily inspection	Daily	1.confirmation ambient temperature, temperature, dust foreign matter 2. whether there is exceptionally to vibrate, the sound 3. whether air vent and so on blocked by the yarn
Periodic inspection	one year	1. Firm part whether loose 2. whether terminal table being damage

CONTENT

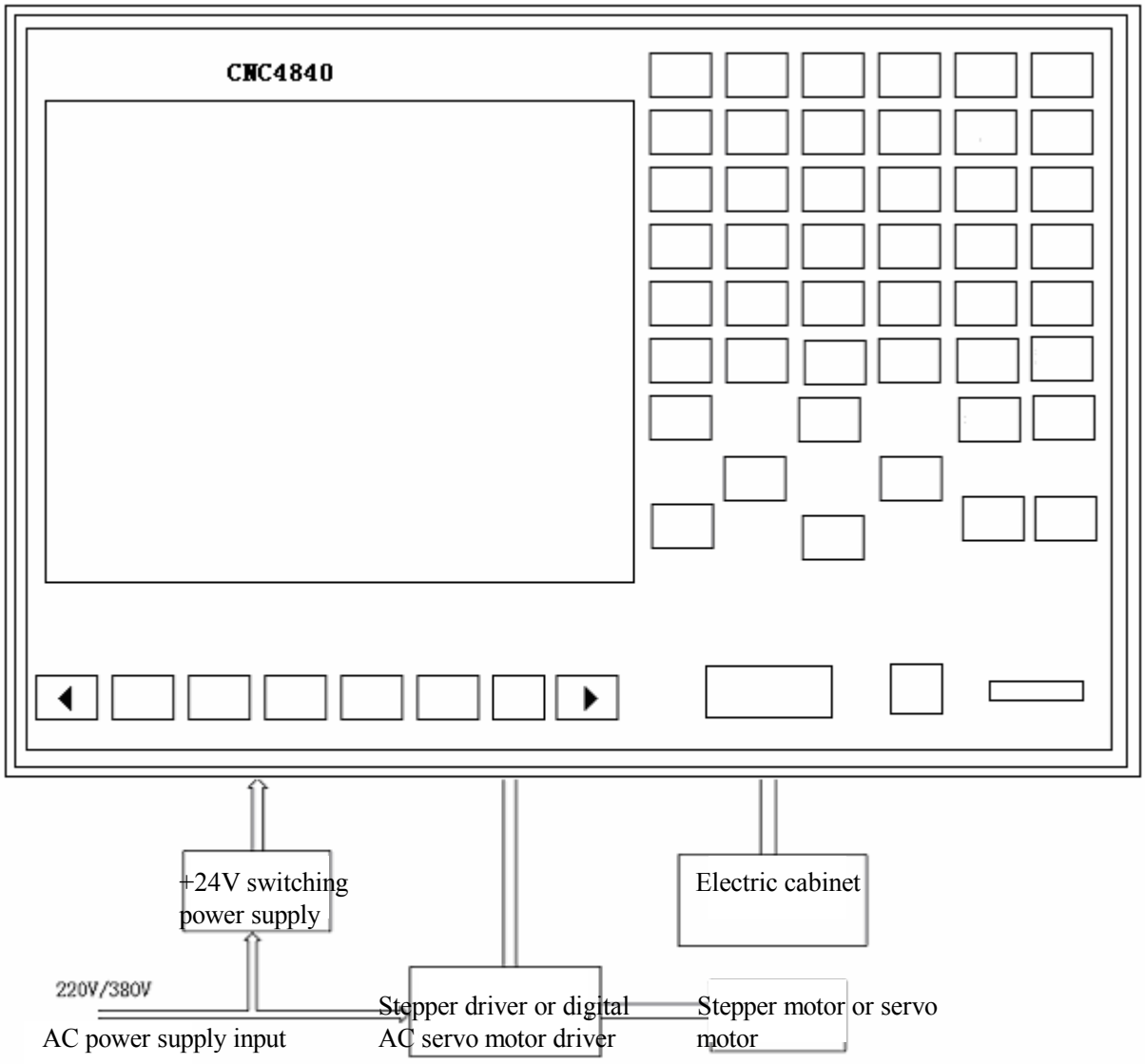
Chapter I Definition of System Interface and Wiring Description

I. System Structure

1. Parts of CNC4840 NCS

CNC4840 NCS is composed by the following main units:

1. CNC control unit (Control device CNC4840)
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.

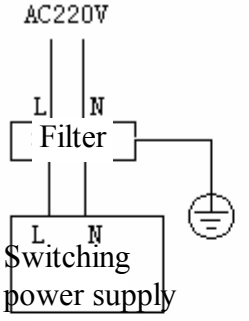
Methods to prevent the interference

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

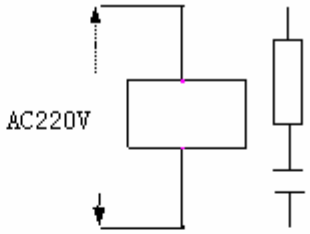
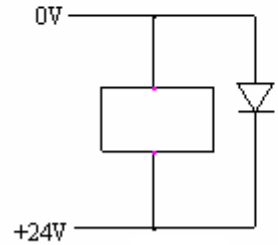
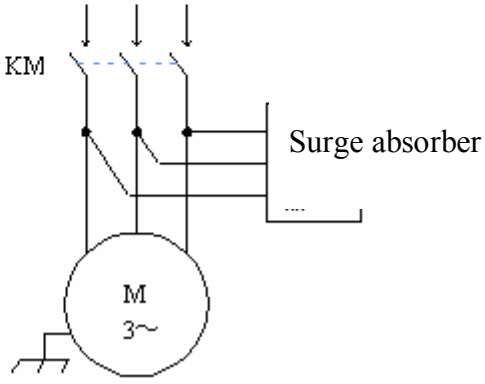


图 1-5



Picture 1-6



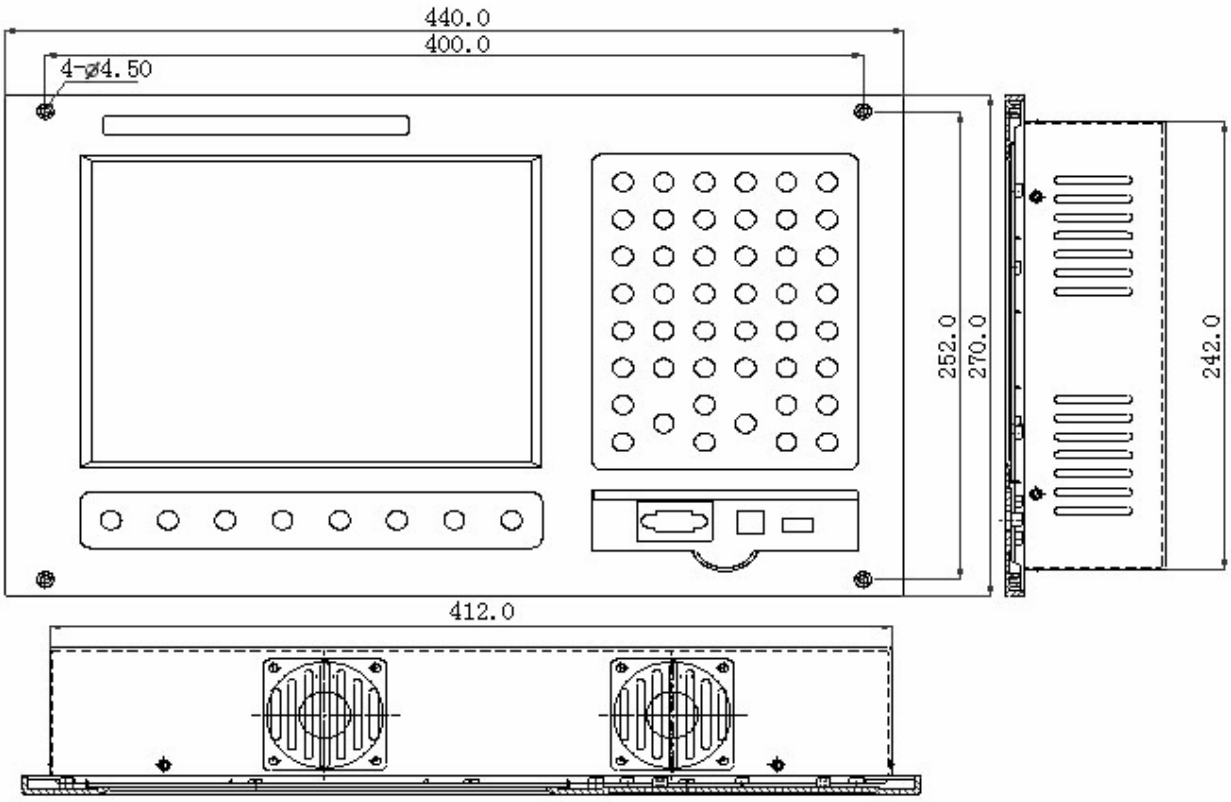
Picture 1-7

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 CNC4840 controller

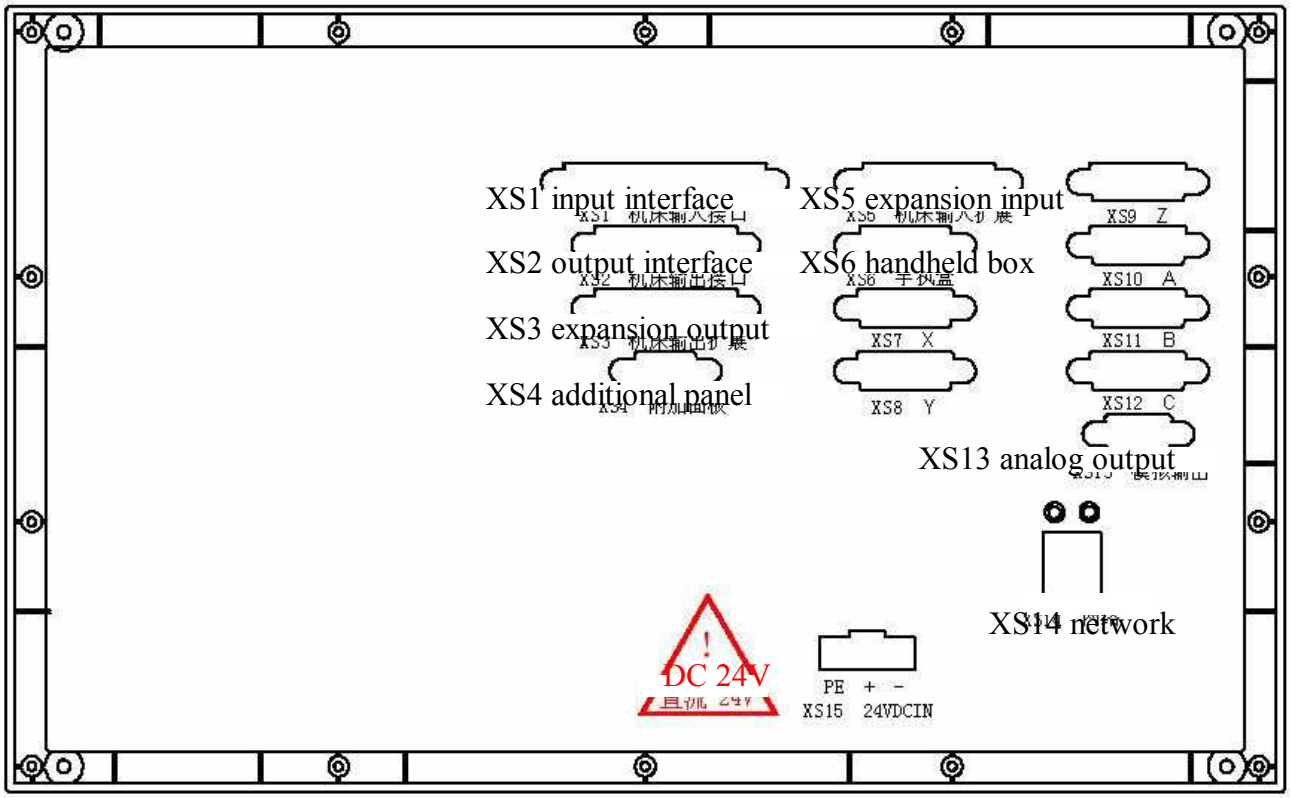


II. External Connection

1. External Interface

CNC4840 control unit is connected to the external devices via the rear and front interfaces.

1. The outer casing of CNC4840 is defined as follows:

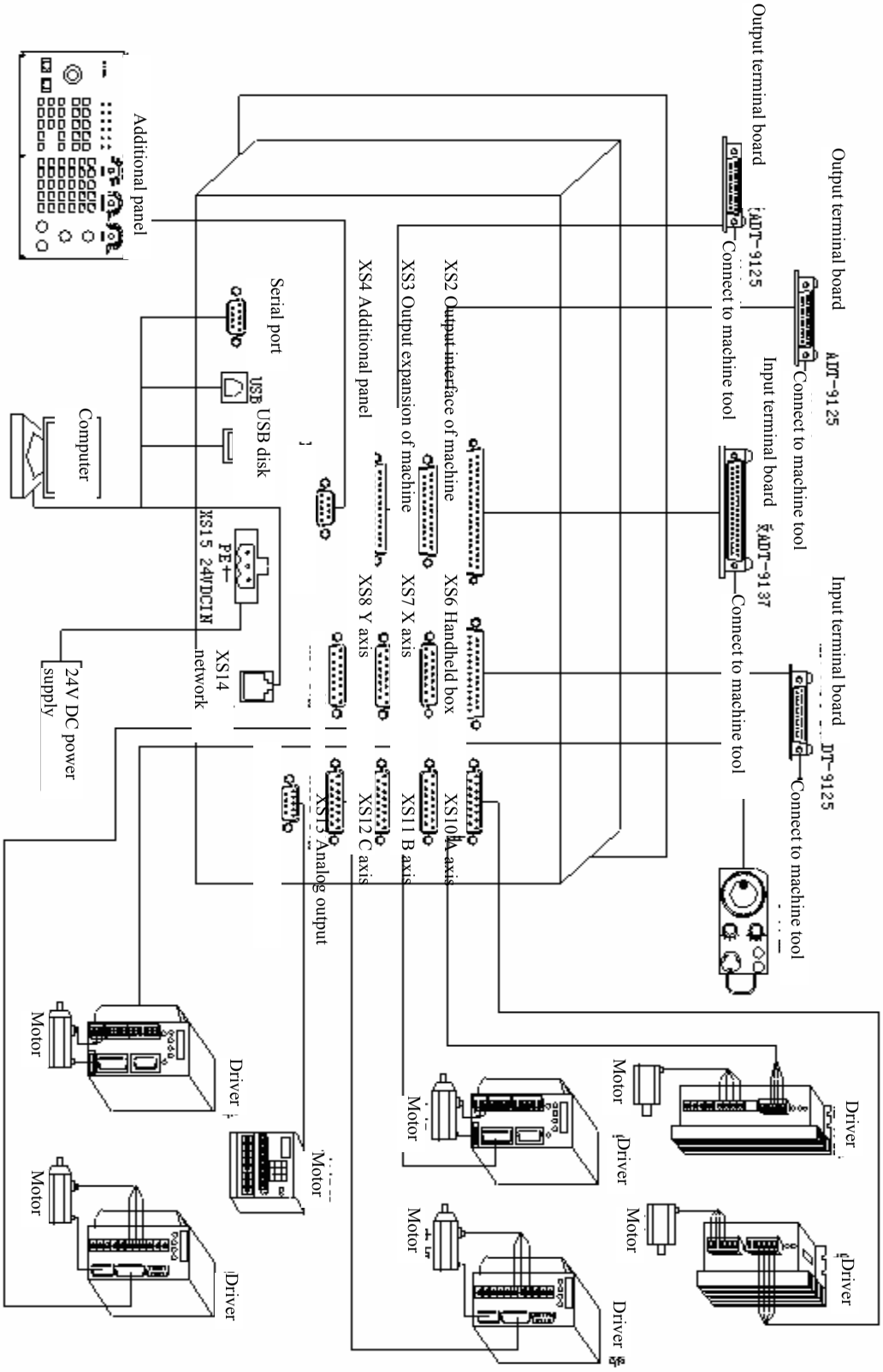


X, Y, Z, A, B, and C refer to the connecting signal of stepper motor driver or digital AC servo driver of each axis. CNC4840 controller uses X, Y, Z, A, B, and C axes at the moment.

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.

CNC4840 controller uses the 24V DC power supply, and the internal power consumption is about 5W.

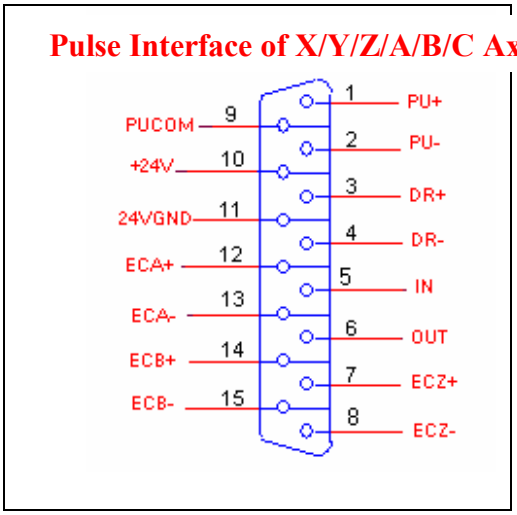
General Wiring Diagram



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:

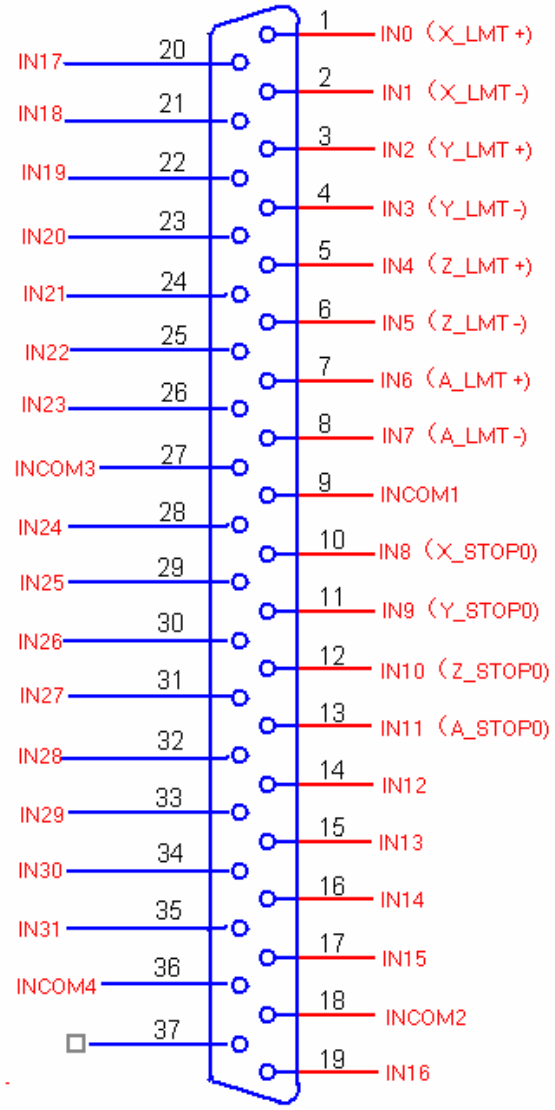
● Pulse interface of axis 1-6



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

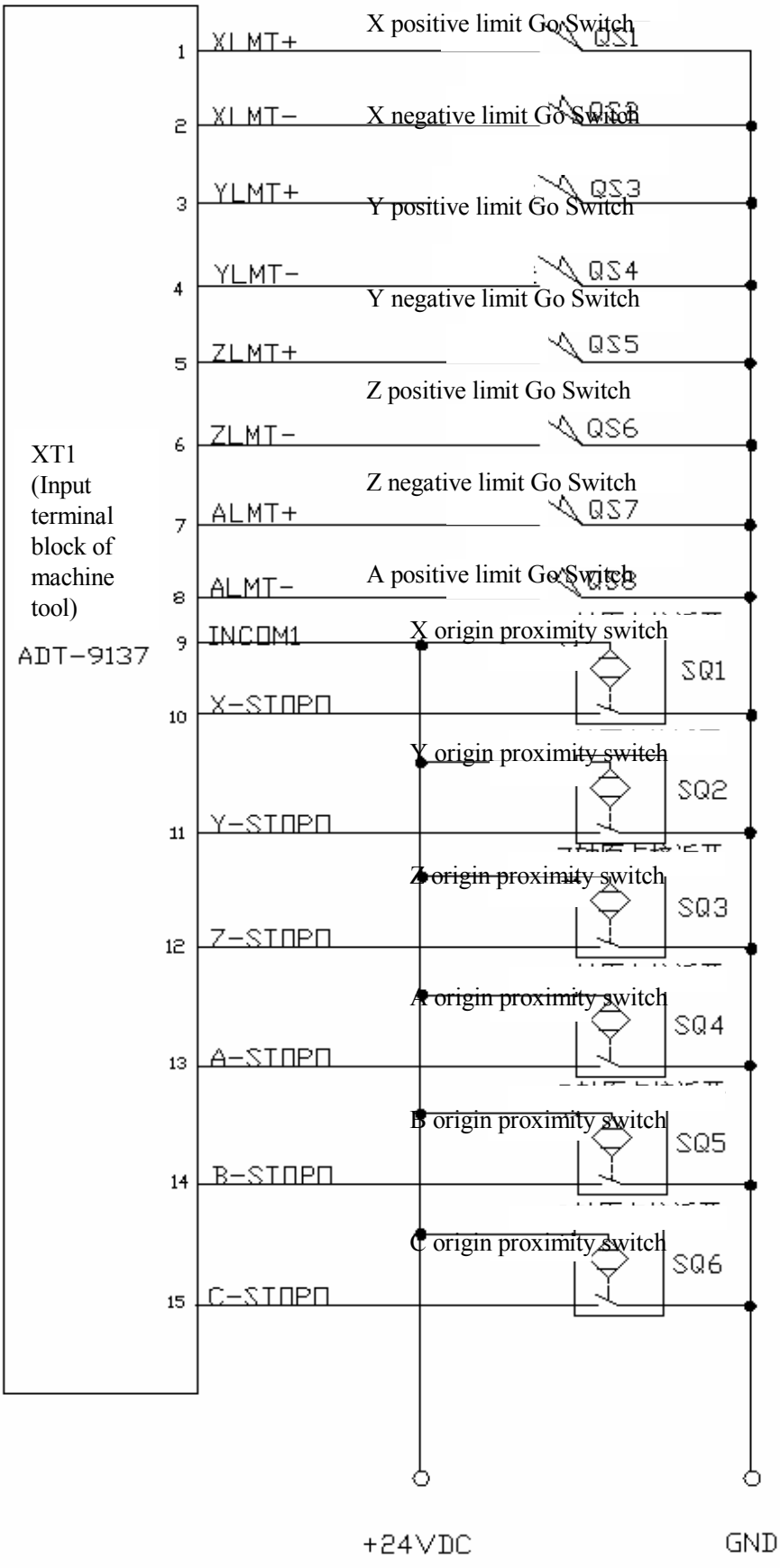
1) The digital input interfaces include the zero points of XYZABC axes, hardware limit signal of XYZA axes, etc. The definition is as follows:

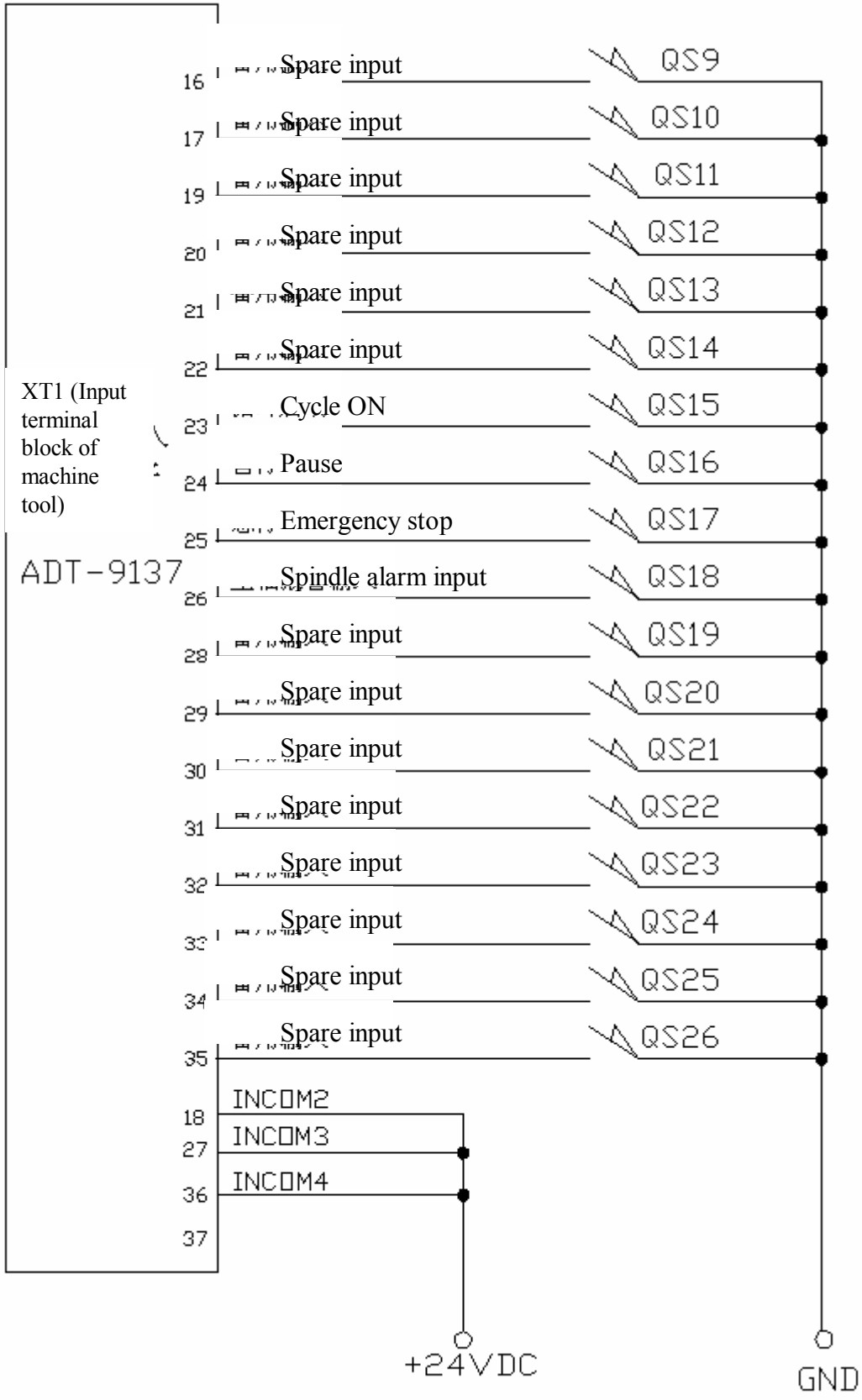


Input Interface of Machine

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

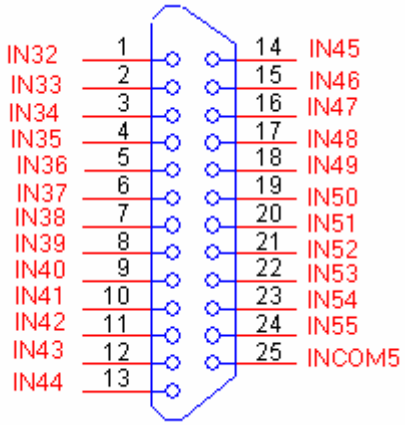
2) Diagram of wiring between input interfaces and photoelectric switch/proximity switch is as follows:




XT1 (Input terminal block of machine tool)
ADT-9137
16
17
19
20
21
22
23
24
25
26
28
29
30
31
32
33
34
35
18
27
36
37
Spare input
Spare input
Spare input
Spare input
Spare input
Spare input
Cycle ON
Pause
Emergency stop
Spindle alarm input
Spare input
Spare input
Spare input
Spare input
Spare input
Spare input
Spare input
Spare input
INCOM2
INCOM3
INCOM4
QS9
QS10
QS11
QS12
QS13
QS14
QS15
QS16
QS17
QS18
QS19
QS20
QS21
QS22
QS23
QS24
QS25
QS26
+24VDC
GND

4. Input Expansion Interface

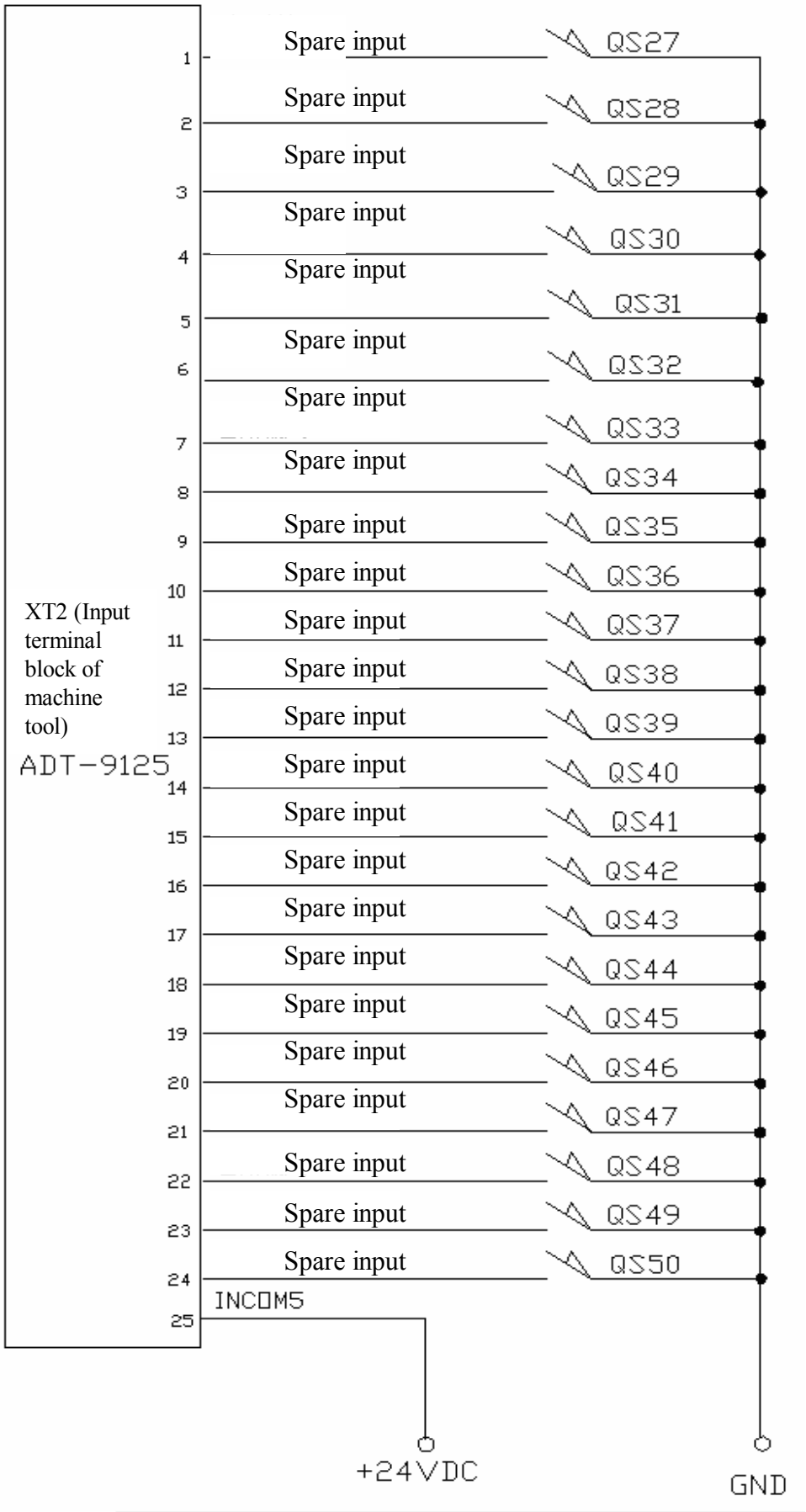
1) Digital input interfaces include BC hardware limit, other spare input signals, etc. the definitions are as follows:



Input Expansion Interface

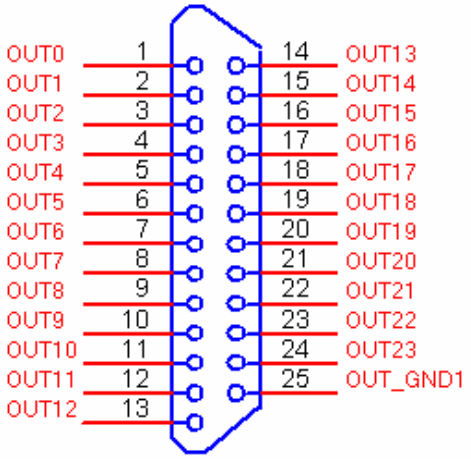
Line S/N	Name	Function
1	IN32	Spare input
2	IN33	Spare input
3	IN34	Spare input
4	IN35	Spare input
5	IN36	Spare input
6	IN37	Spare input
7	IN38	Spare input
8	IN39	Spare input
9	IN40	Spare input
10	IN41	Spare input
11	IN42	Spare input
12	IN43	Spare input
13	IN44	Spare input
14	IN45	Spare input
15	IN46	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

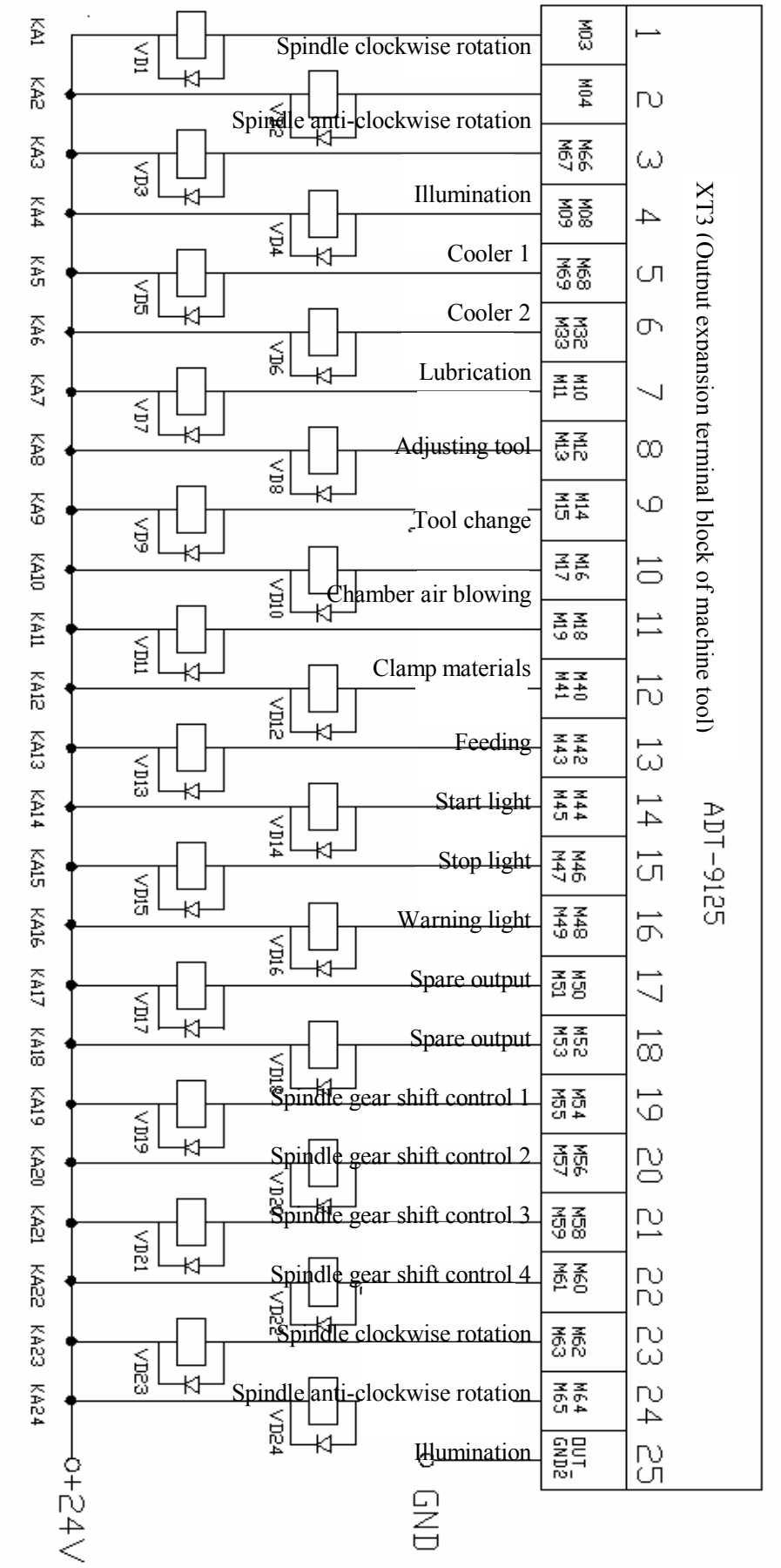
1) The output interface and the wiring are defined as follows:



Output Interfaces

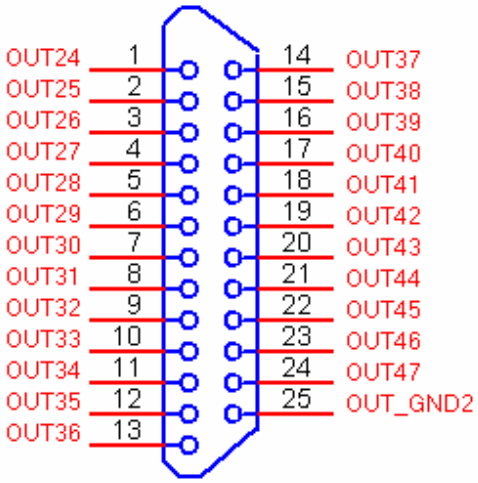
Line S/N	Name	Function
1	OUT0	Spindle clockwise rotation (M03)
2	OUT1	Spindle anti-clockwise rotation
3	OUT2	Illumination (M66, M67)
4	OUT3	Cooler 1 (M08, M09)
5	OUT4	Cooler 2 (M68, M69)
6	OUT5	Lubrication (M32, M33)
7	OUT6	Adjusting tool (M10, M11)
8	OUT7	Tool change (M12, M13)
9	OUT8	Chamber air blowing (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,
22	OUT21	Spindle gear shift control 2 (M60,
23	OUT22	Spindle gear shift control 3 (M62,
24	OUT23	Spindle gear shift control 4 (M64,
25	OUT_GND1	Common power supply of 12v-, 24v-

2) Wiring diagram of output interface is as follows:



6. Output Expansion Interface

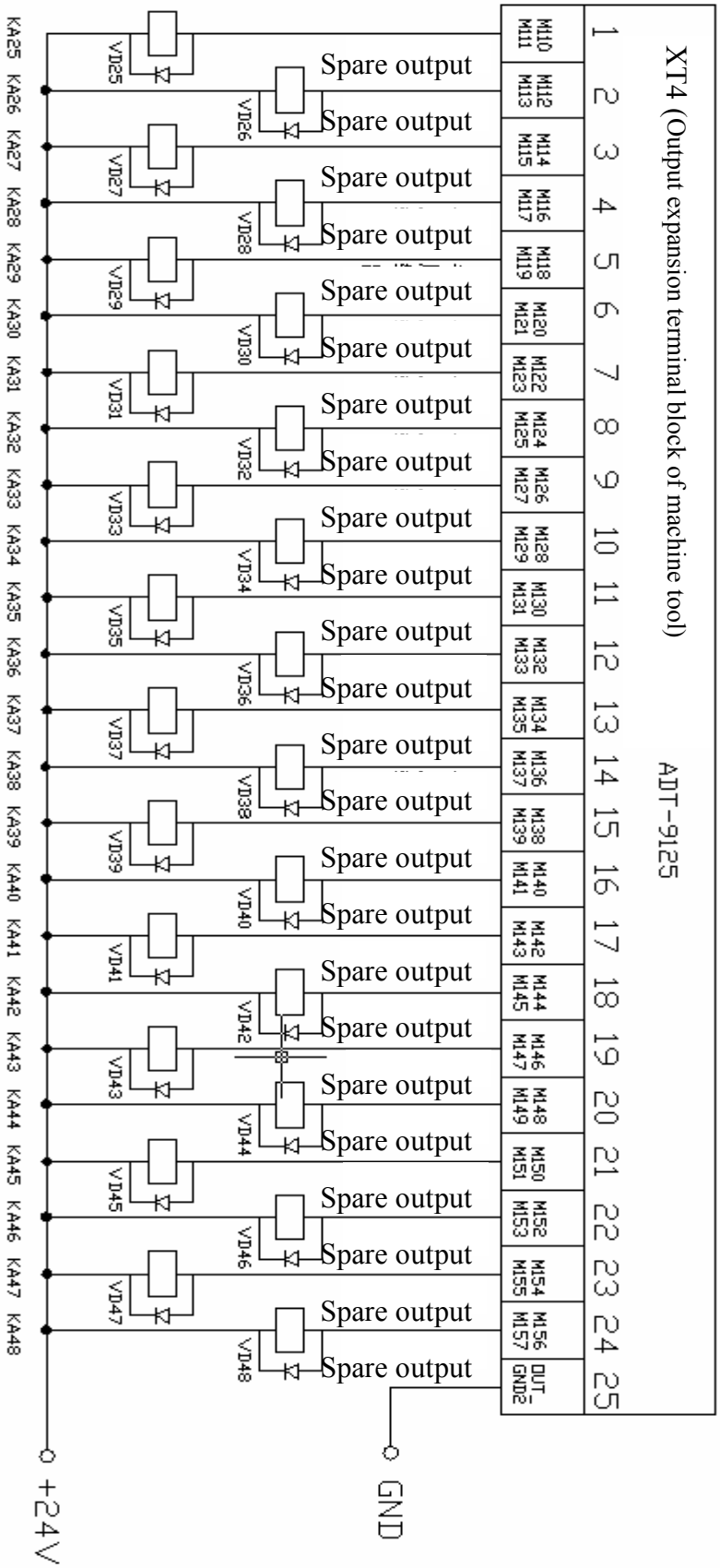
1) The expansion output interface and the wiring are defined as follows:



Output Expansion Interface

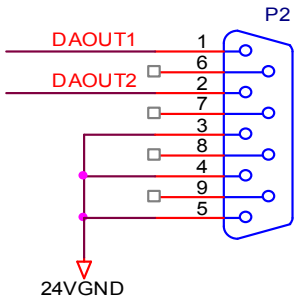
Line S/N	Name	Function
1	OUT24	Spare output (M110, M111)F10
2	OUT25	Spare output (M112, M113)F11
3	OUT26	Spare output (M114, M115)F12
4	OUT27	Spare output (M116, M117)F13
5	OUT28	Spare output (M118, M119)F14
6	OUT29	Spare output (M120, M121)F15
7	OUT30	Spare output (M122, M123)F16
8	OUT31	Spare output (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:

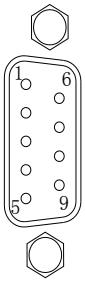


Analog Output

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-

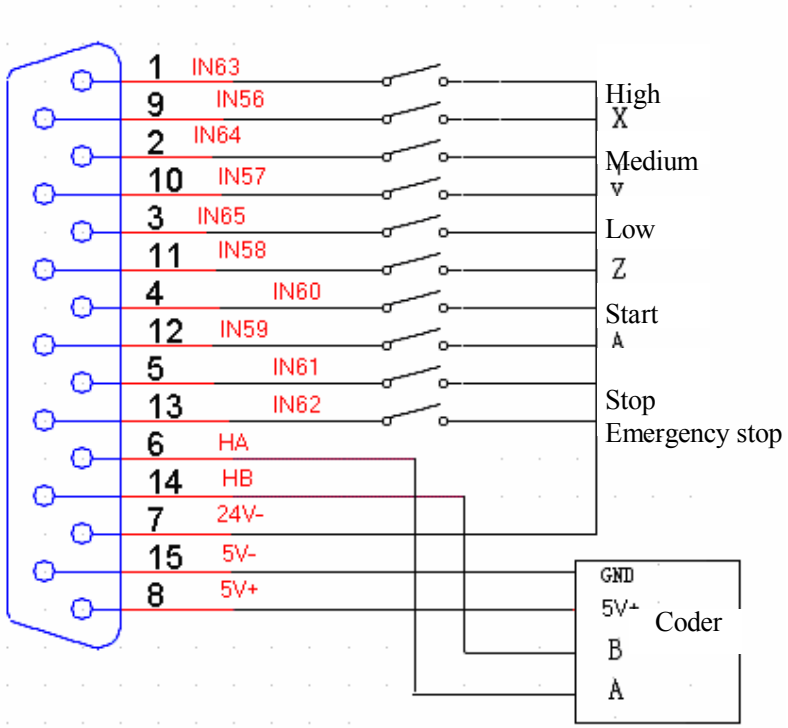
CWF-S1 transducer
signal interface

	VI1
	CM

Note: Choose either DAOUT1
or DAOUT2 to connect the
transducer

8. Handheld box

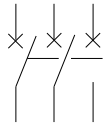
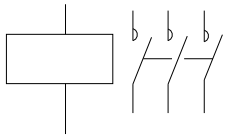
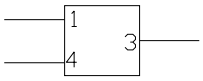
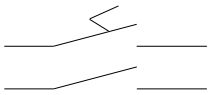
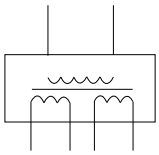
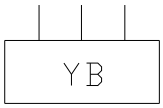
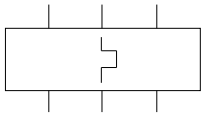
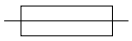
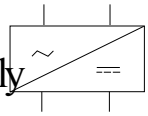
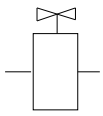
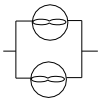
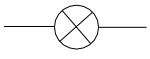
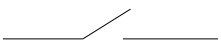
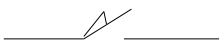
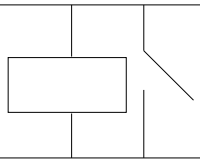
Handheld box: Connected with our standard handheld box ADT-CNC4A, multiplexing with manual pulse generator count and Y-axis coder



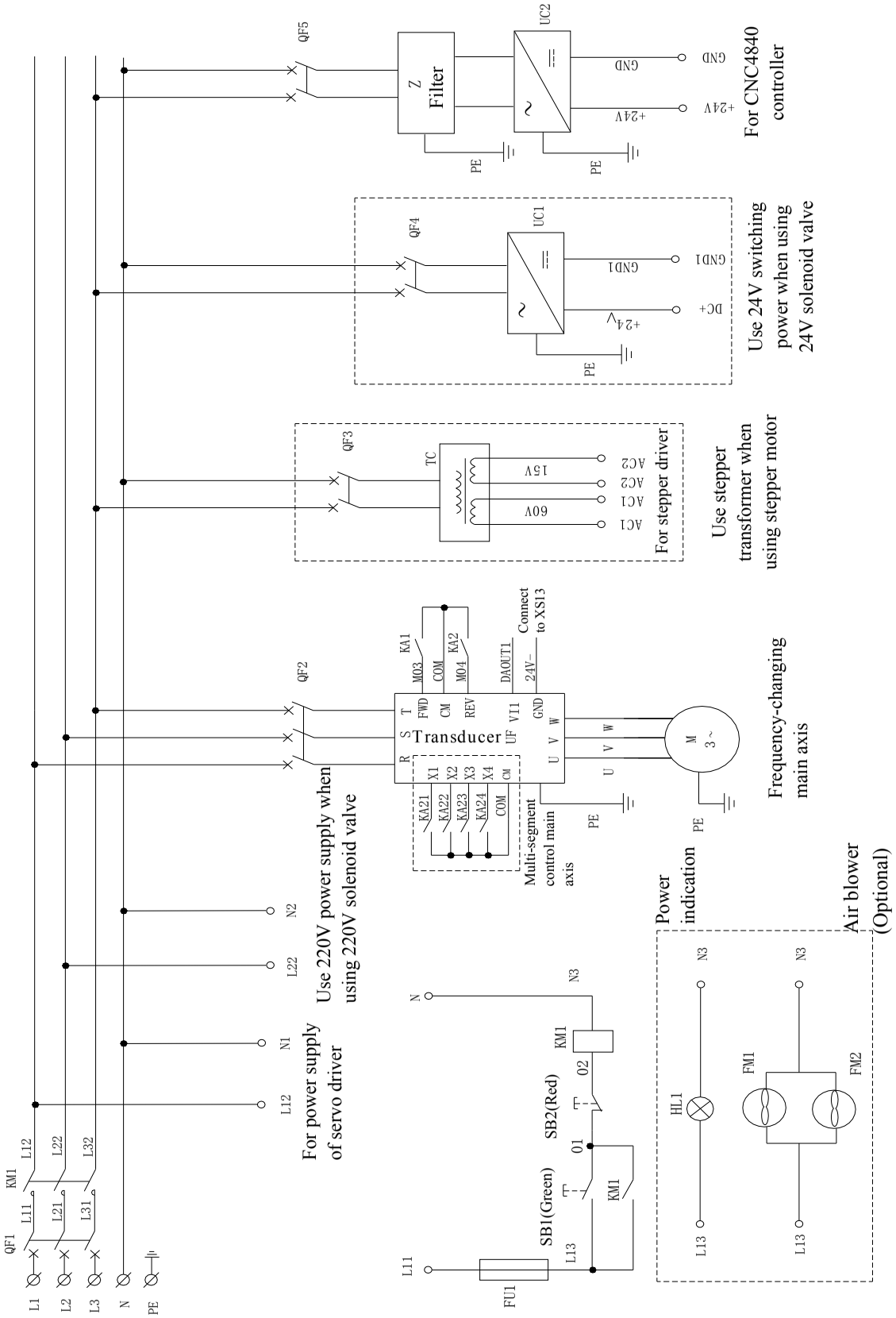
Definition of corresponding casing: Handheld box

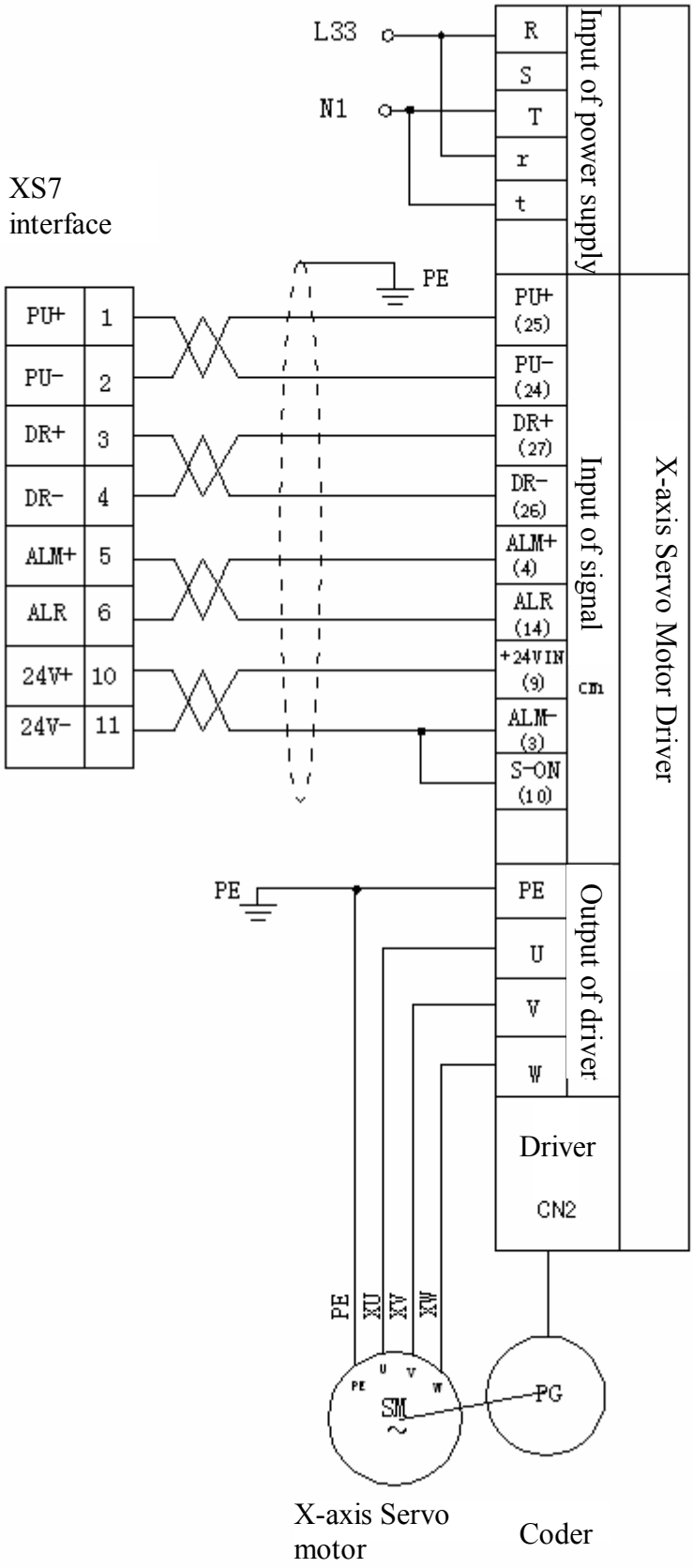
Line S/N	Name	Function
1	IN63	0.1 shift switch-High
2	IN64	0.01 shift switch-Medium
3	IN65	0.001 shift switch-Low
4	IN60	Start-up
5	IN61	Stop
6	HA	Handle coder phase-A input signal
7	24V-	Internal -24V power supply
8	5V+	Internal +5V power supply
9	IN56	Select X axis
10	IN57	Select Y axis
11	IN58	Select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

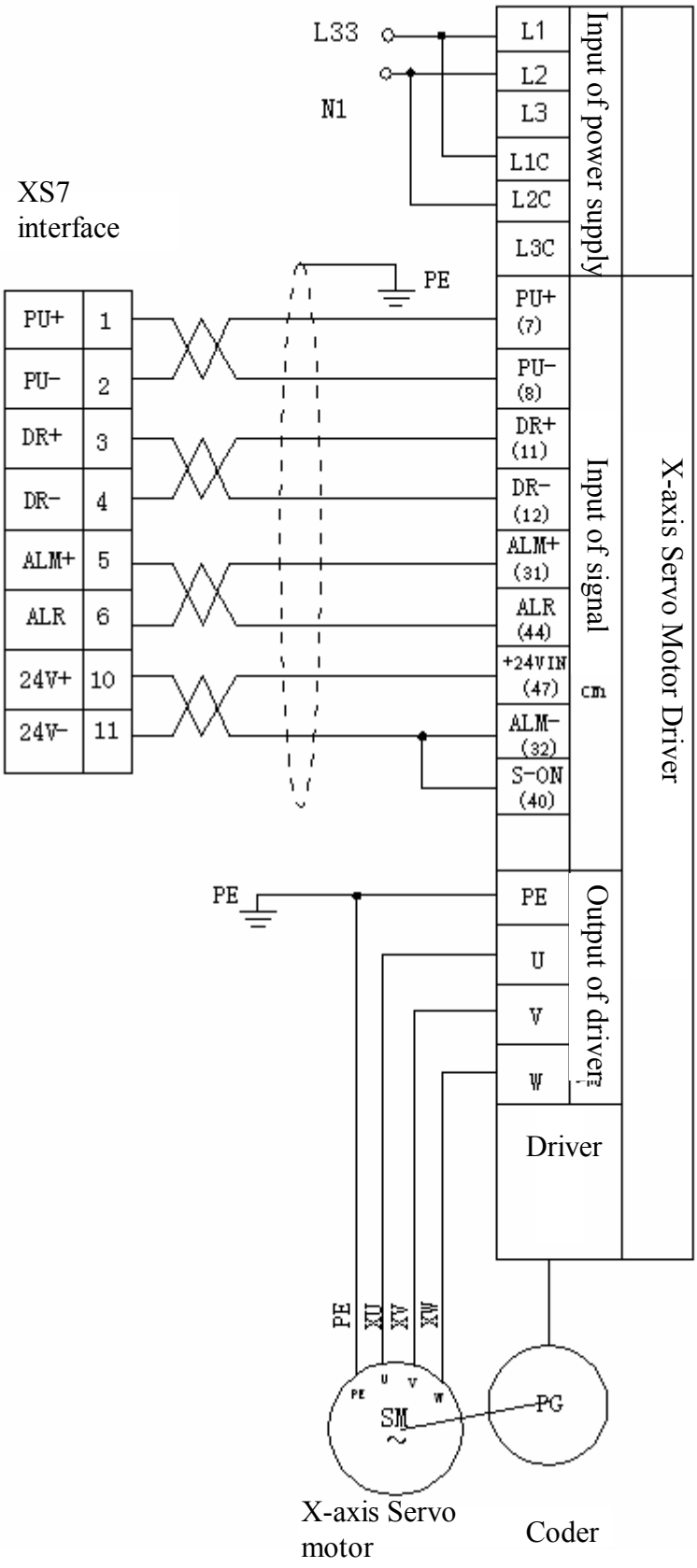
Sign	Name	Chart	Sign	Name	Chart
QF	Breaker		SM	Servo motor	
KM	Contactor		M	Stepper motor	
UF	Transducer		SQ	Proximity switch	
M	Motor		SA	Foot switch	
TC	Transformer		YB	Thermal relay	
Z	filter		FR	thermal relay	
FU	Fuse		UC	Switching power supply	
SB	Button		YV	Solenoid valve	
FM	Air blower		C	compacitor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

10. Legend of connection between CNC4840 and servo/stepper driver

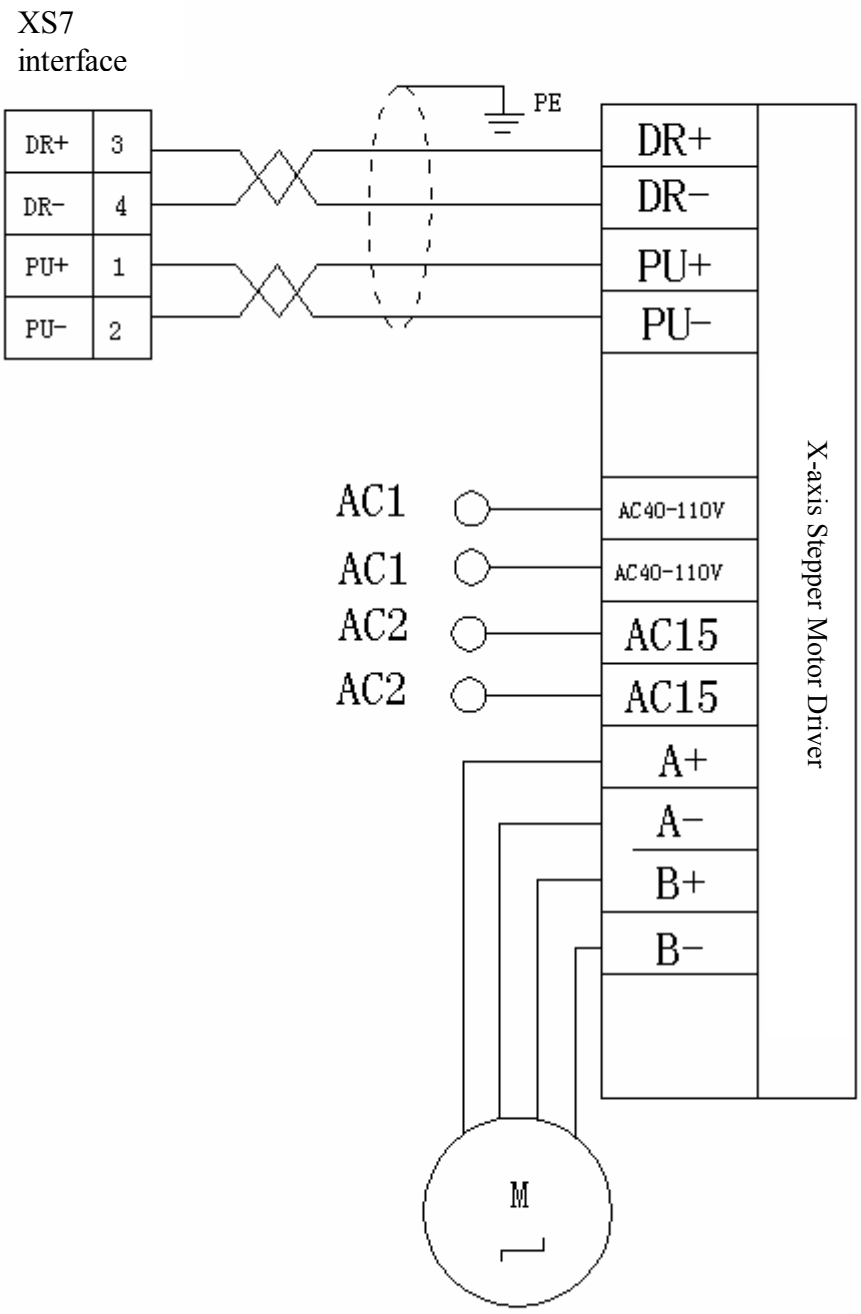




Example 1: Connection with JaBao QS5 driver

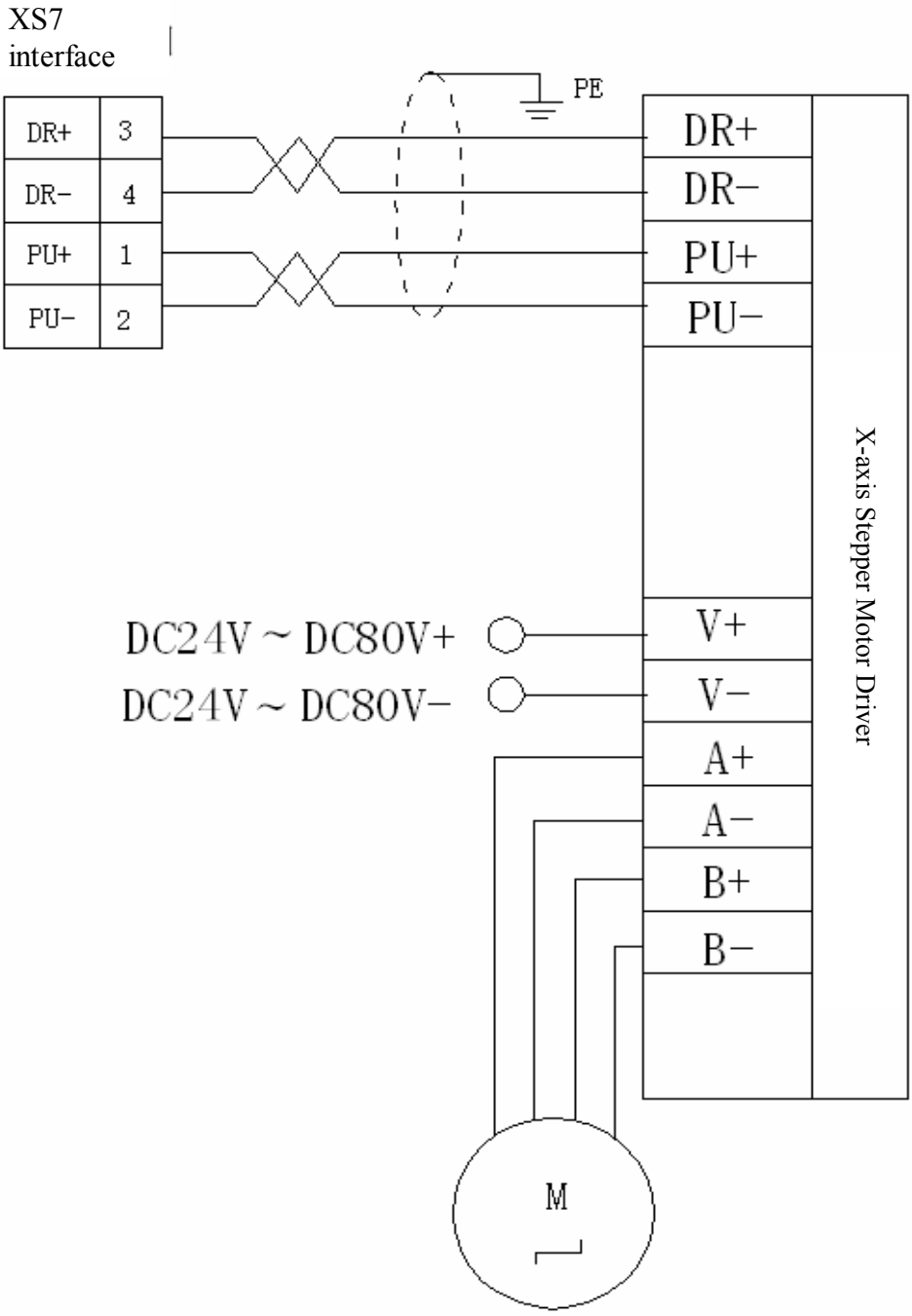


Example 2: Connection with SGDM driver



X-axis Stepper Motor

Example 3: Connection with Q2BYG1106M stepper driver



X-axis Stepper Motor

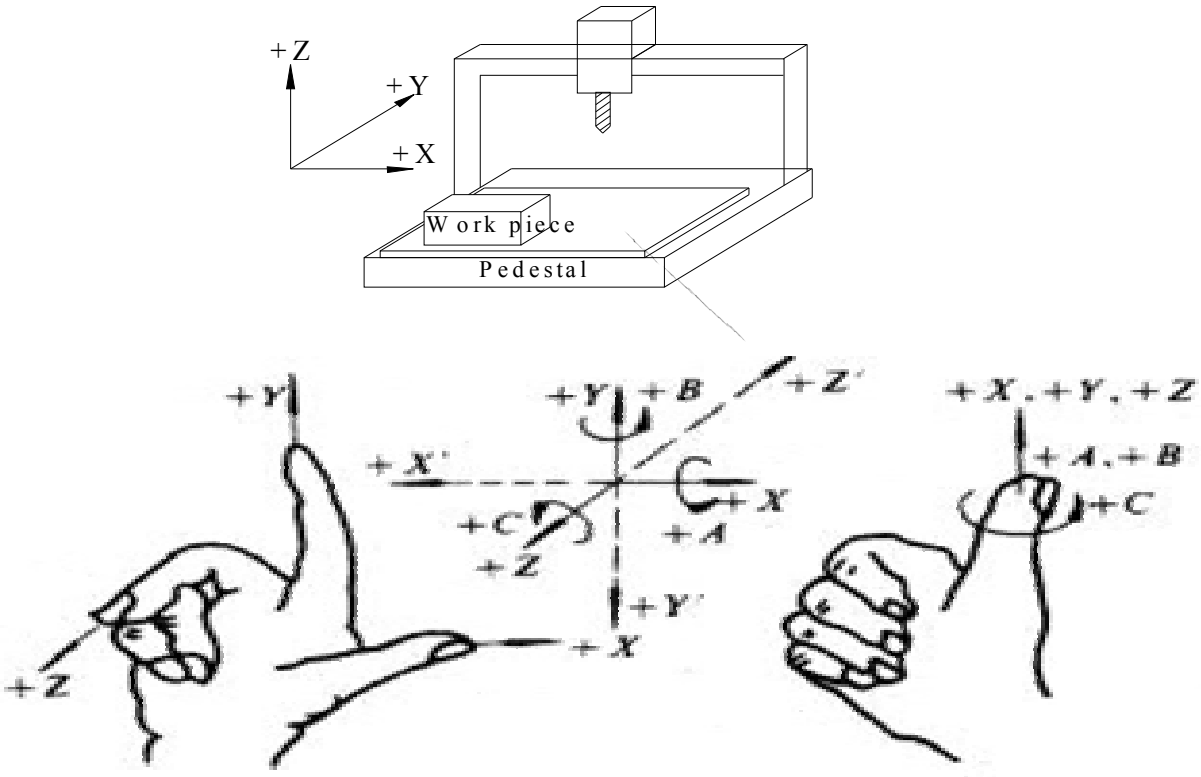
Example 4: Connection with Q2BYG808M stepper driver

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)

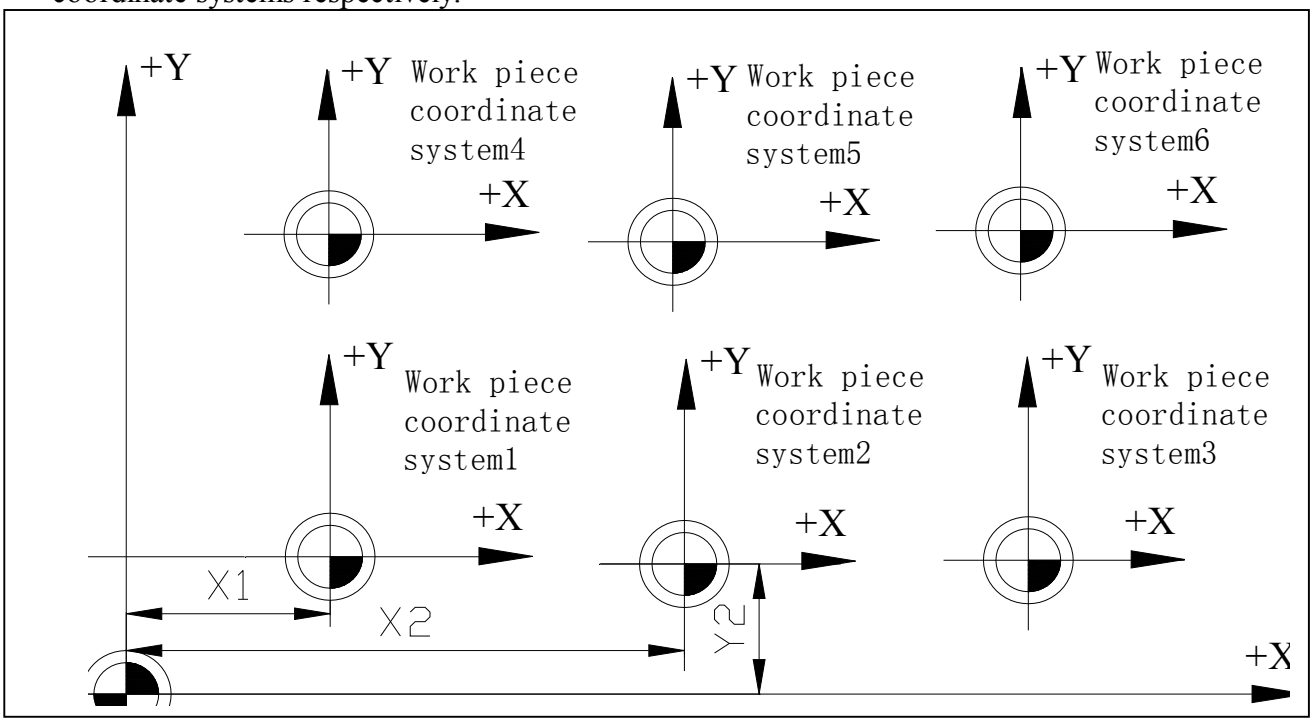
1) Coordinate system of machine tool

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.



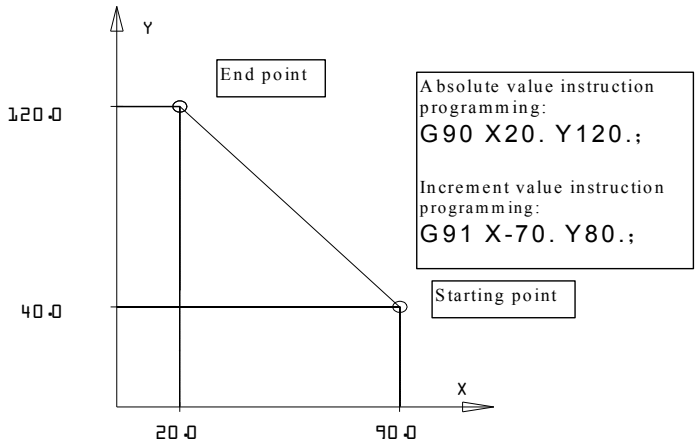
3) Programming of absolute coordinate and relative coordinate (G90, G91)

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.

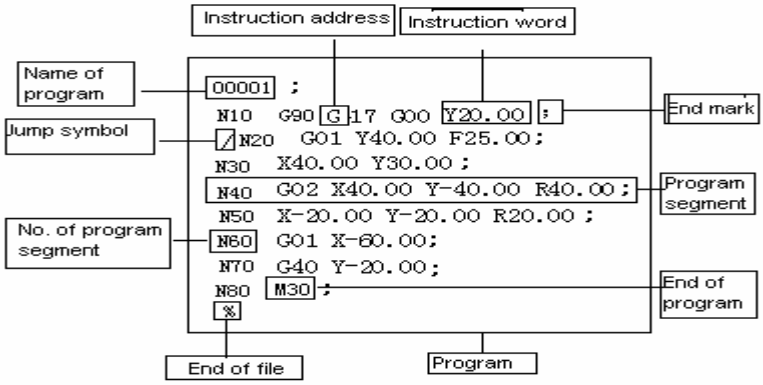
Quick locating feed occurs in the motion between quick feed and location in mode of instruction G00, manual quick move and fixed cycle, and the speed is determined by machine tool parameters. In quick locating feed, the motion of each involved axis is not related, which is moved at the set quick speed.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

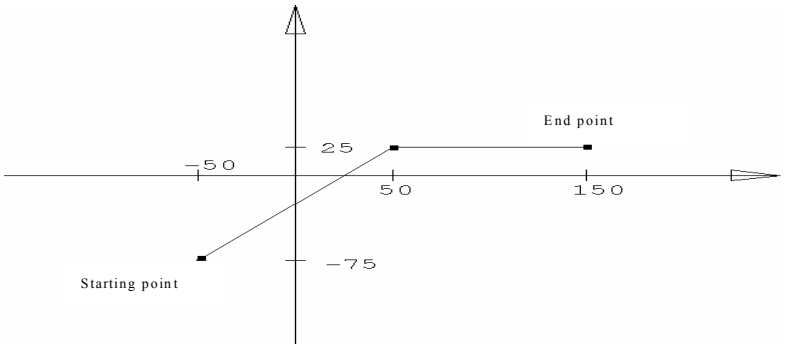
G00 X_Y_Z_ ;

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

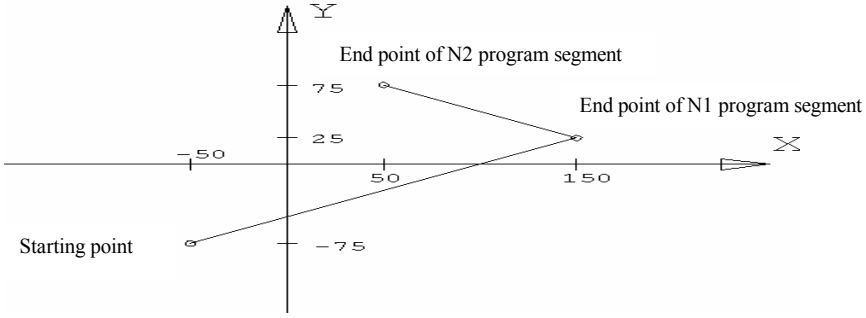
G01 instruction enables the current interpolation mode to be linear interpolation. The tool moves from the current position to position appointed by IP, and the track is a line. F specifies the speed of tool moving along the line, unit in mm/min.

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

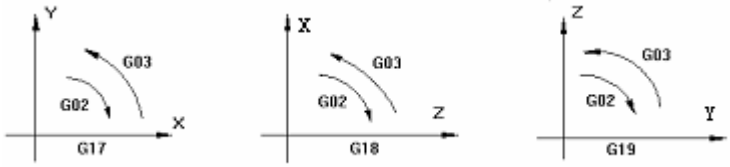
G18 { G02 / G03 } X__ Z__ { (I__ K__) / R__ } F__ ;

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

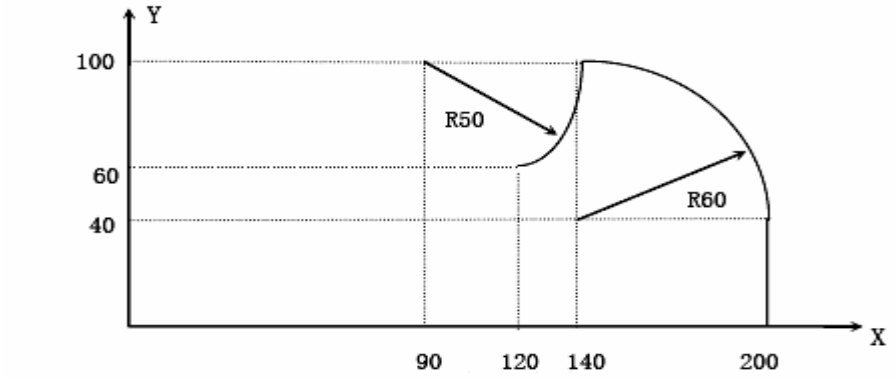
Here, the direction of circular arc, for X--Y plane, is that when viewing the plane from positive to negative of Z axis. Likewise, for X-Z or Y-Z plane, the viewing direction should be from the positive of Y or X to the negative of Y or X (applicable to right handed coordinate system, as follows).



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.

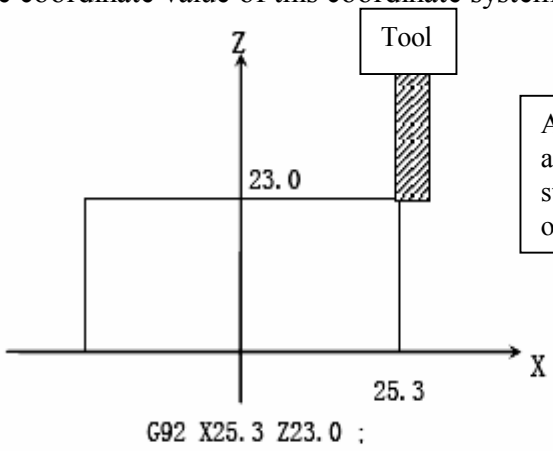
Switch on the machine and return to the origin of machine tool, the workpiece coordinate systems 1~6 are then created. G54 is the initial mode when electrified. The absolute position is the coordinate value of current coordinate system.

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.

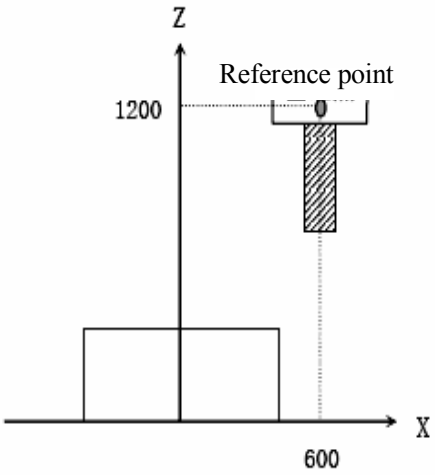
3) Programmable work piece coordinate system (G92)
Format: (G90) G92 X_Y_Z_ ;

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.

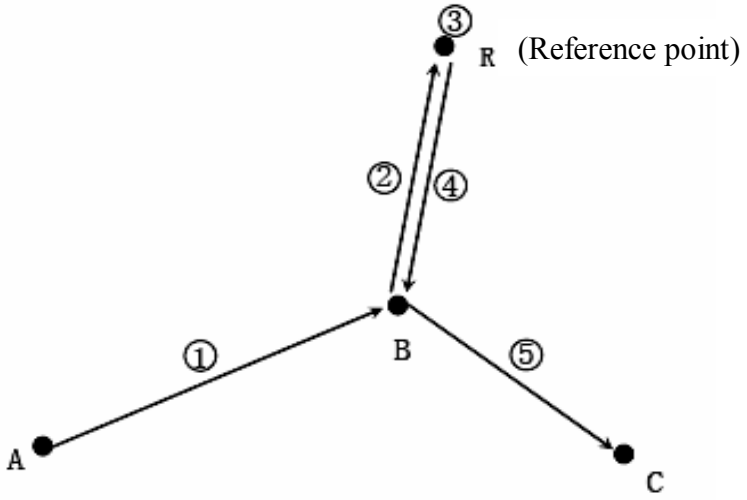
When executing G28 instruction before returning to reference point manually, the motion from intermediate point for each axis is the same as that of returning to reference point manually, and the direction of motion from the intermediate point is positive.

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.

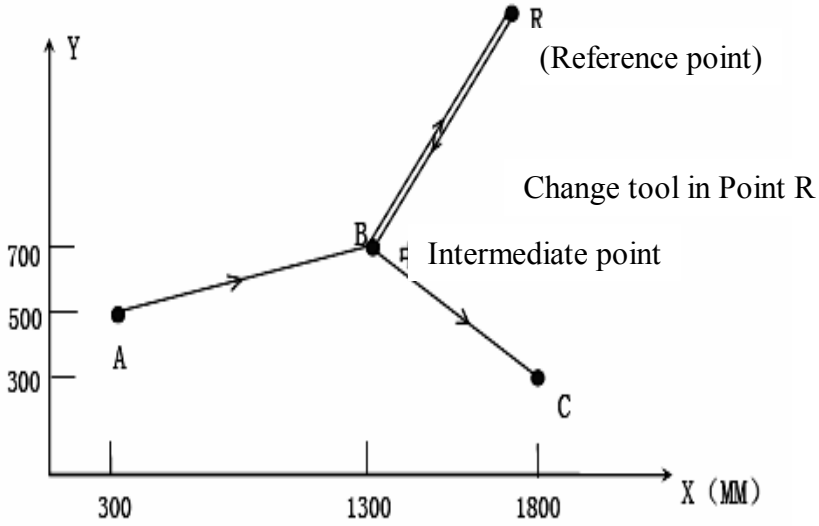
Return from reference point automatically (G29)

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)

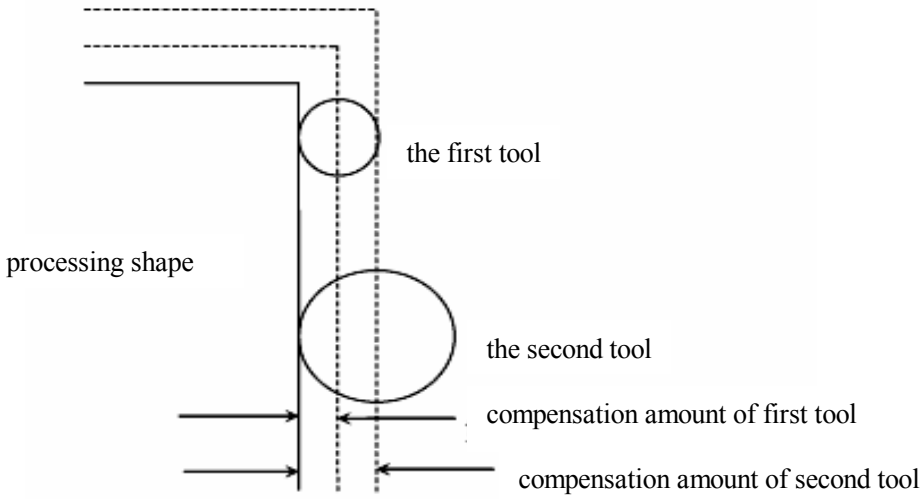
Format: G27 IP_;

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.

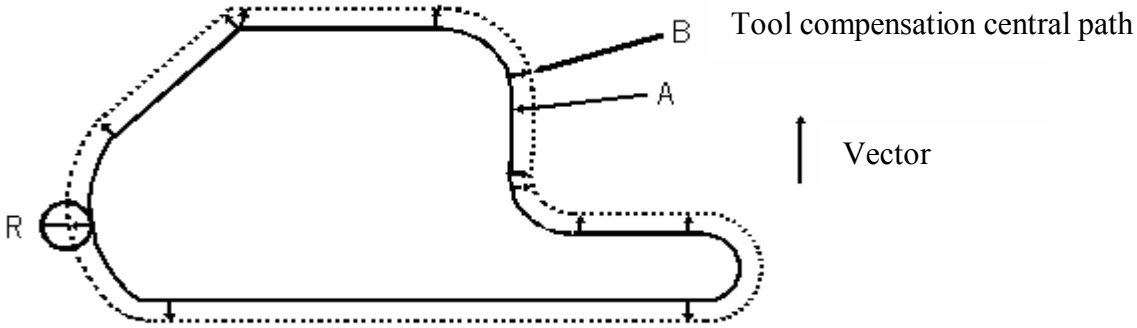
1.2.7 Tool compensation (G40, G41, G42, G43, G44, G49)

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

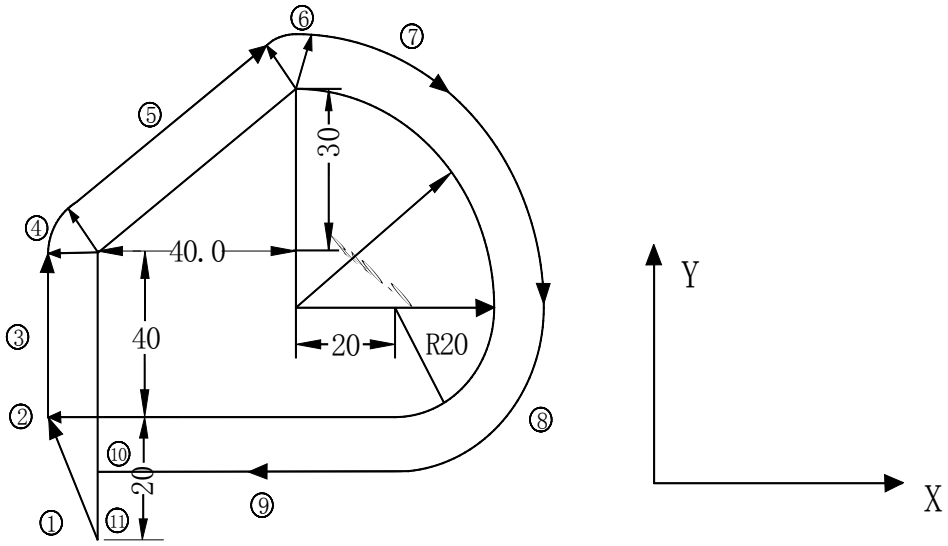
5) G40, G41 and G42

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.

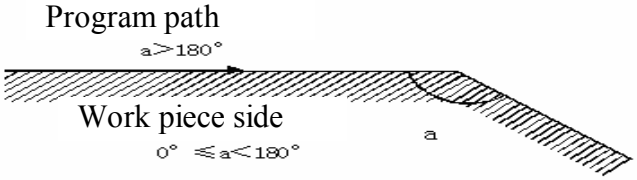
Read into two program segments in single program segment mode, execute the first one, and then stop.

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):

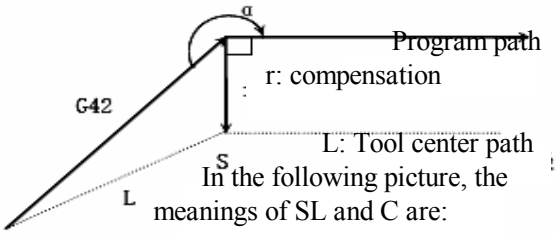
Inside

Outside



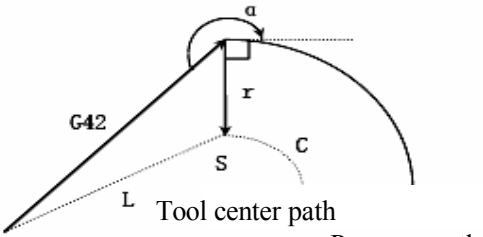
(a) Move along the inside of corner (a≥180°)

(i): Line →Line



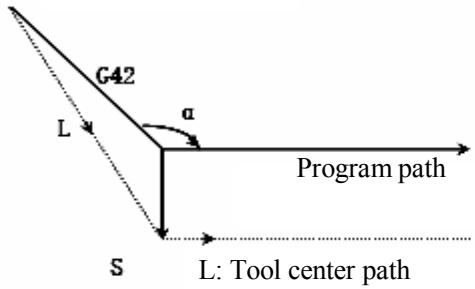
In the following picture, the meanings of SL and C are:
S: Single block stop point
L: Line
C: Circular arc

(ii): Line → Circular arc

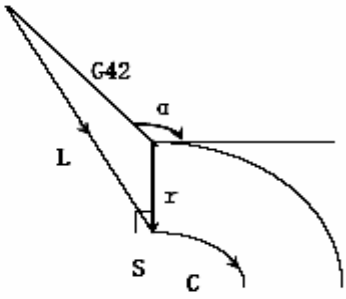


(b) Move along the outside of corner at obtuse angle (180°>a≥90°)

(i): Line →Line



(ii): Line → Circular arc

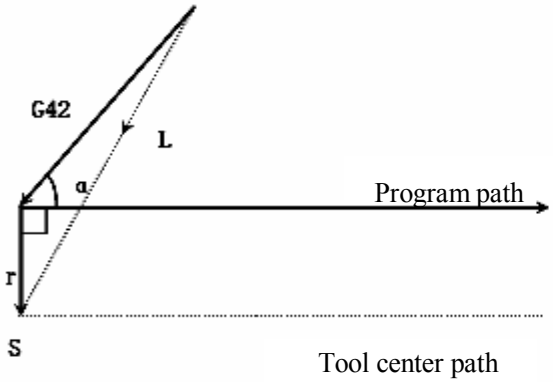


Tool center path

Program path

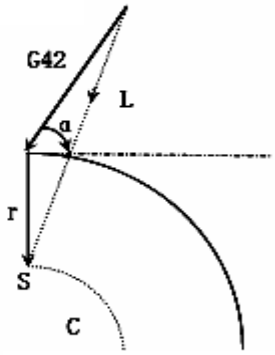
(c) Move along the outside of corner at acute angle (a<180°)

(i): Line →Line



Tool center path

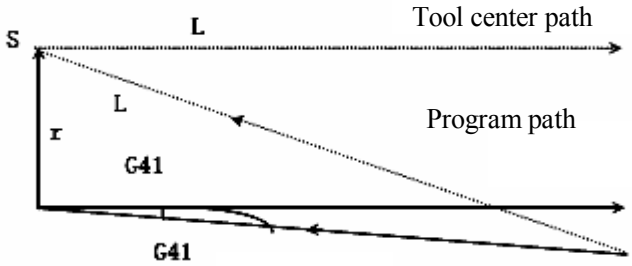
(ii): Line → Circular arc



Tool center path

Program path

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



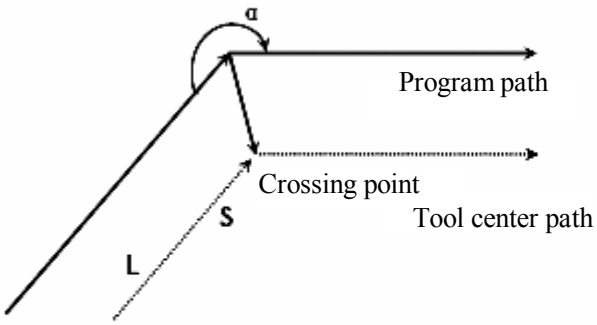
a<1°

c. Compensation mode

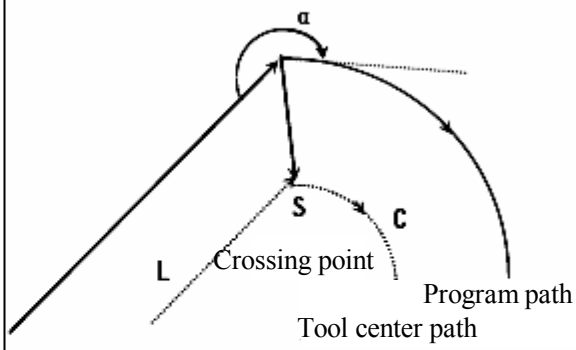
In compensation mode, if you do not appoint two or more non-moving instructions (auxiliary function or pause, etc.) continuously, the compensation will be executed properly; otherwise, there will be over cutting or short of cutting. Compensation plane cannot be modified when in compensation mode; otherwise,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

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

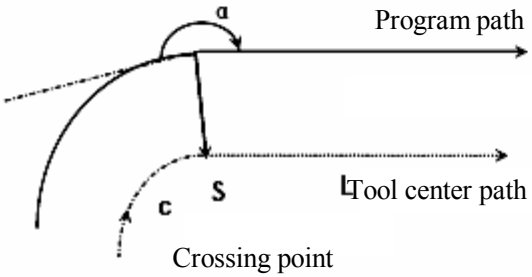
(i) Line →Line



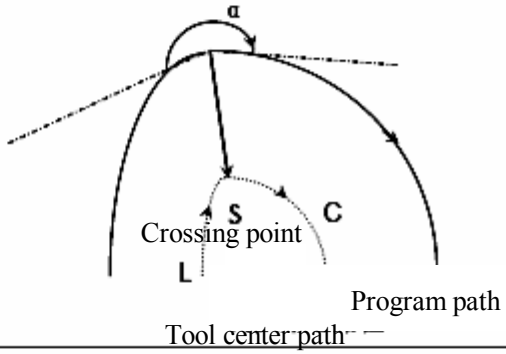
(ii) Line →Circular arc



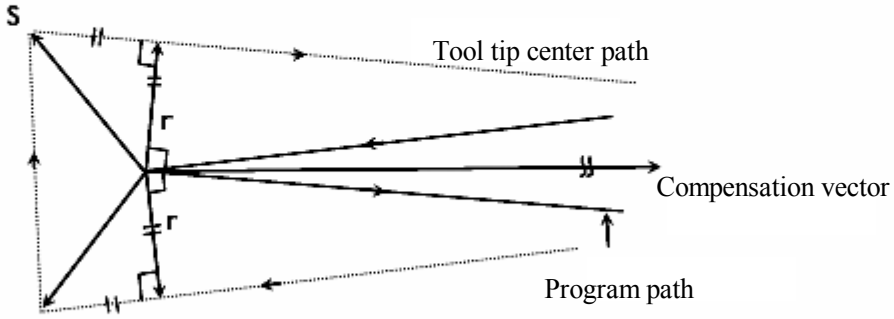
(iii) Circular arc →Line



(iv) Circular arc →Circular arc



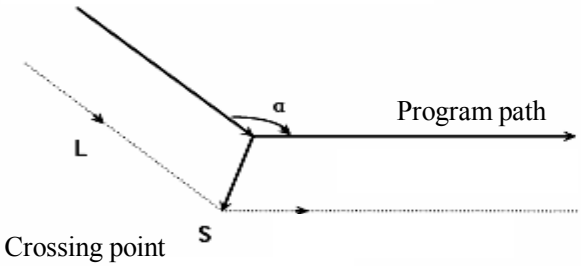
(v) processing <1° inside and enlarging compensation vector
(I) Line →Line



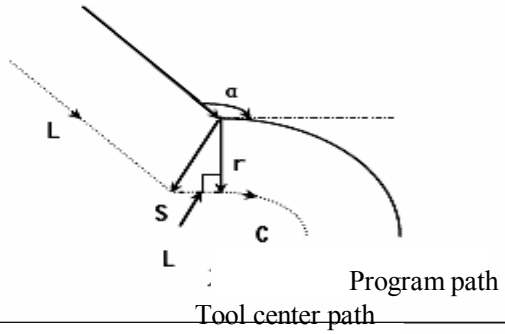
Considering the following conditions with the same method
(II) Circular arc →Line
(III) Line →Circular arc
(Iv) Circular arc →Circular arc

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

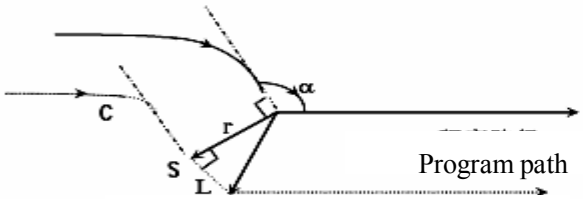
(i) Line →Line



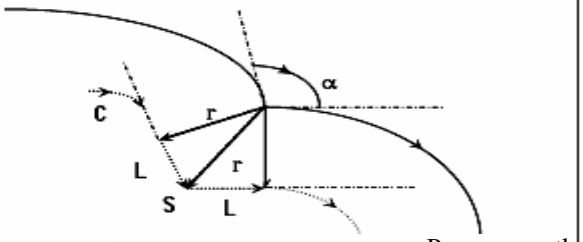
(ii) Line →Circular arc



(ii) Circular arc →Line

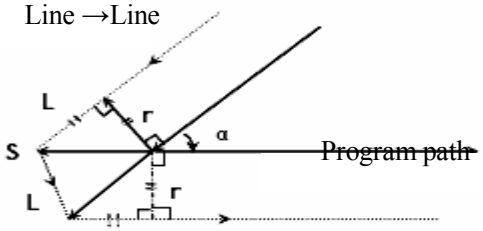


(iv) Circular arc →Circular arc

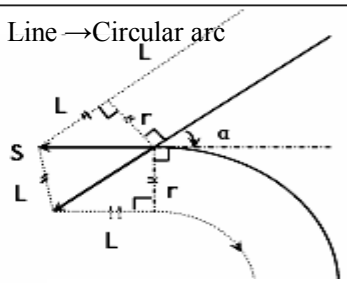


(c) move along the outside of corner at acute angle (a<90°)

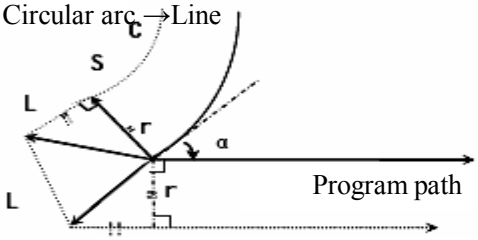
(iii) Line →Line



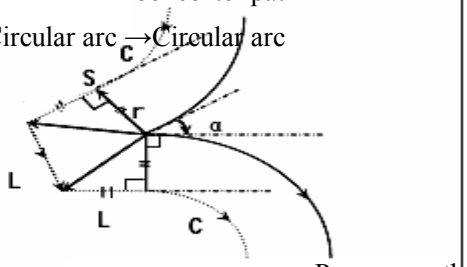
(ii) Line →Circular arc



(iv) Circular arc →Line

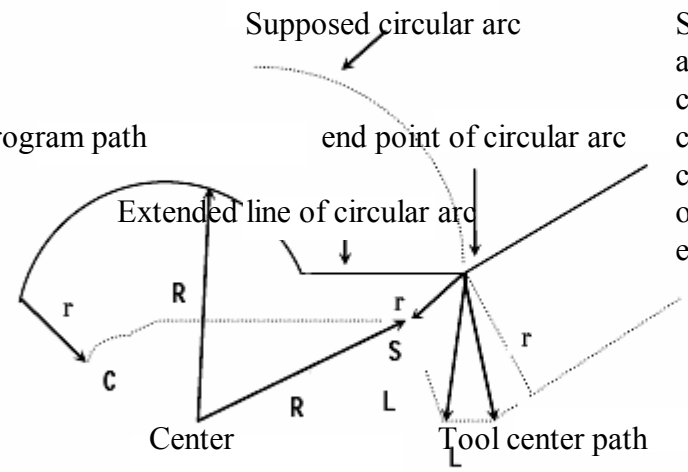


(iv) Circular arc →Circular arc



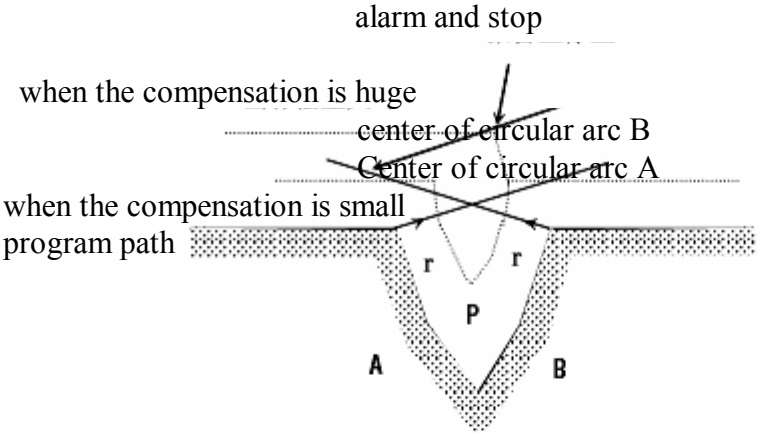
(d) special conditions

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



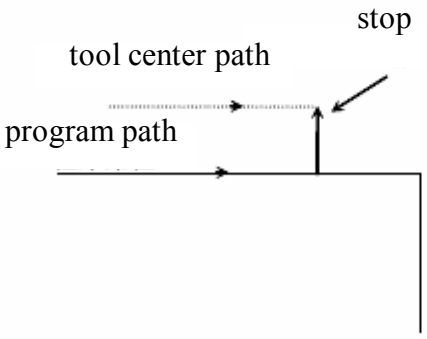
When the circular arc is not on end point, the extended line is as shown in the left picture. Suppose a circular arc comes across the end point, and the compensation takes the supposed circular arc as vector, the tool center path is different from the offset path while considering the extended line of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

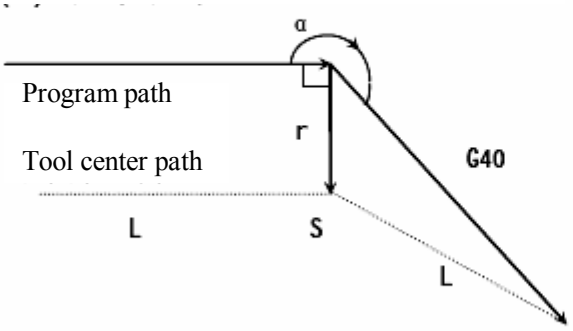
In compensation mode, the system enters the compensation cancellation mode when the program satisfying any following conditions is executed, and the action of this program segment is called as compensation cancellation.

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

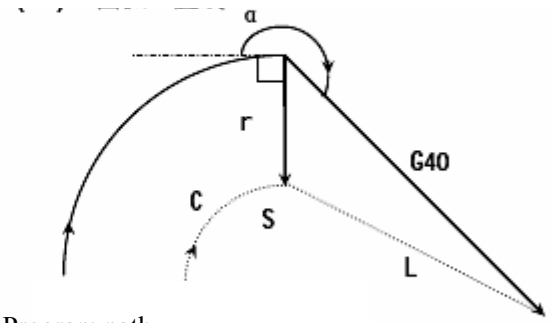
When executing the compensation cancellation, circular arc instruction (G03 and G02) cannot be used; otherwise, it will give out an alarm (P/S34) and the tool will stop.

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

(i) Line →Line



(II) Circular arc →Line

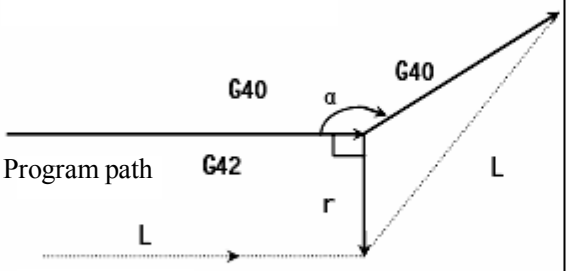


Program path

Tool center path

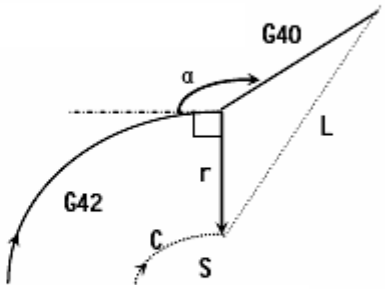
(b) Move along the inside of corne (90°≤a≤180°)

(I) Line →Line



Tool center path

(II) Circular arc →Line

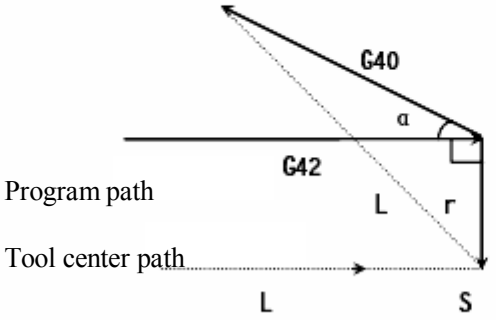


Program path

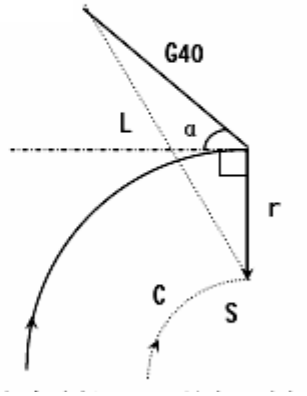
Tool center path

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

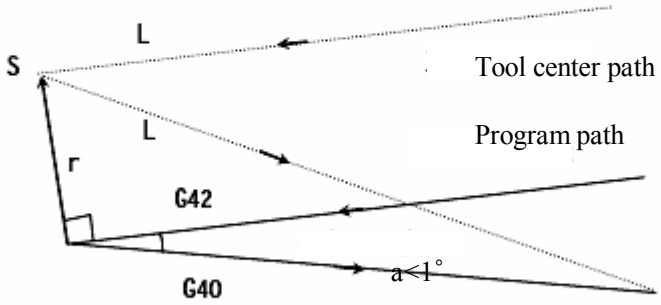
(i) Line →Line



(II) Circular arc →Line



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



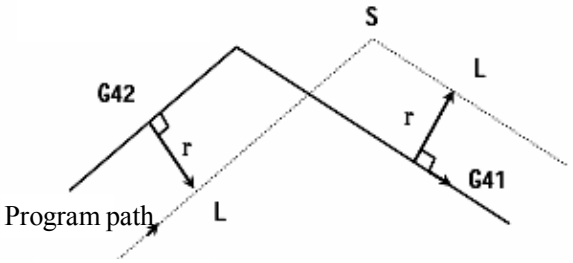
e. Change compensation direction in compensation mode

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

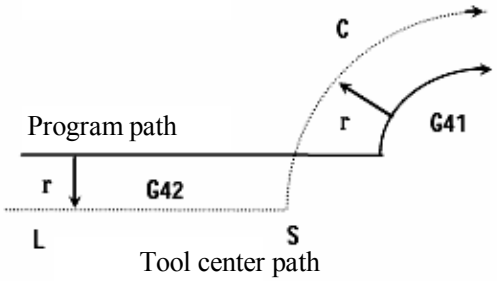
Sign of compensation / G 码		
G41	Left compensation	Right compensation
G42	Right compensation	Left compensation

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.

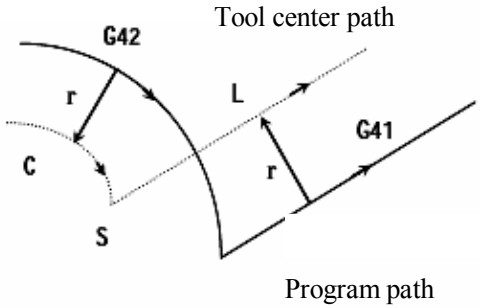
(i) Line →Line



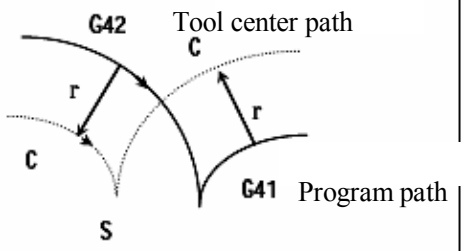
(II) Circular arc →Line



(iii) Circular arc →Line



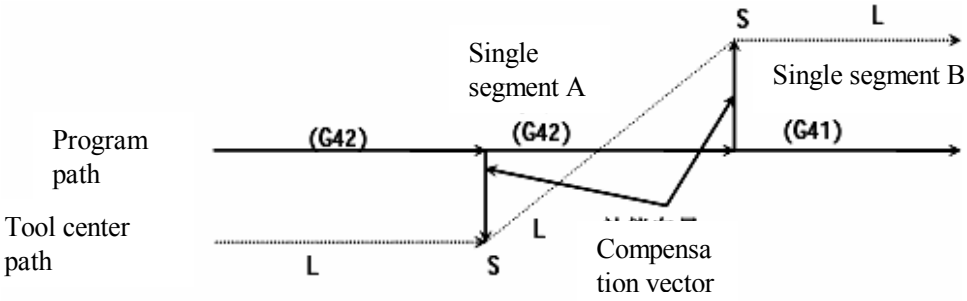
(iv) Circular arc →Circular arc

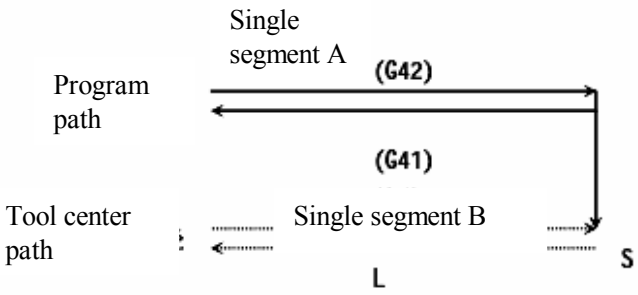


➢ If the compensation is executed normally, but there is no crossing point

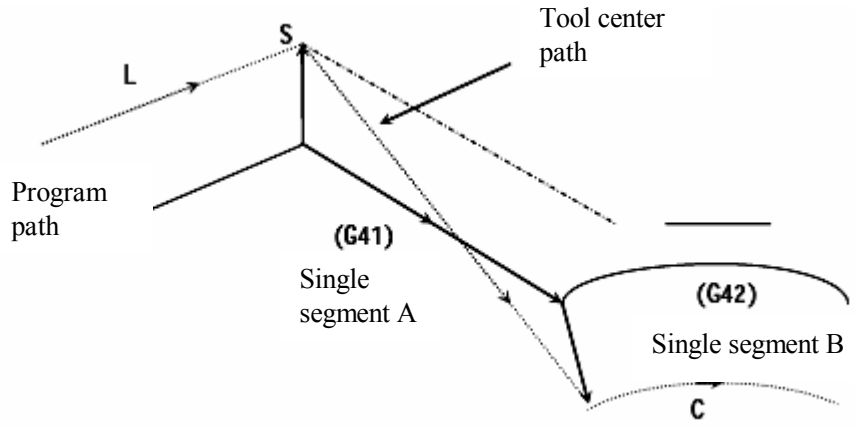
When using G41 and G42 to change the offset direction from program segment A to B, if it is not required to compensate the crossing point of path, make a vector at the starting point of program segment B that is vertical to program segment B.

✧ Line----Line

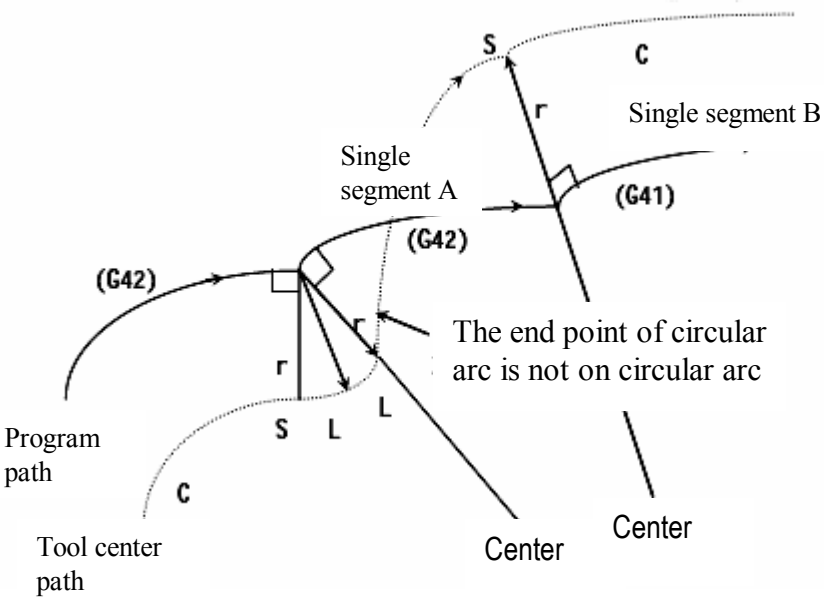




✧ Line----Circular arc

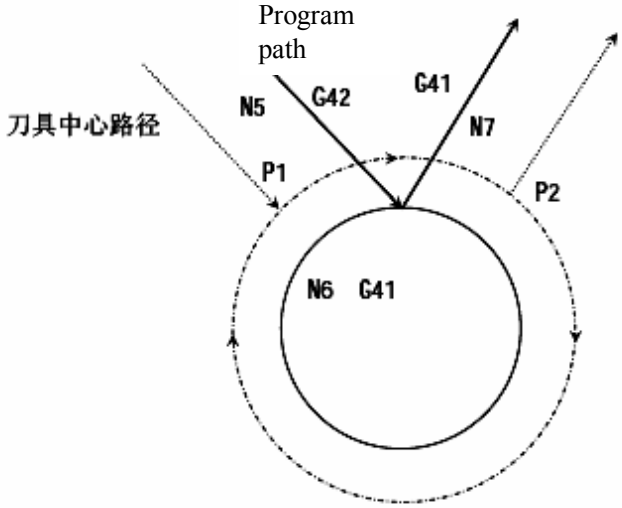


✧ Circular arc ---- Circular arc



➢ 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;

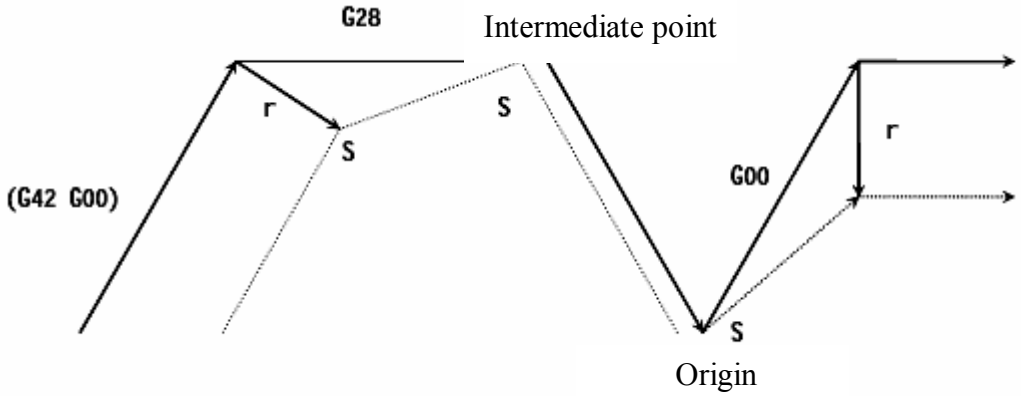
At this time, the tool center path is not a circular arc but a section of arc between P1 and P2
In some conditions, it may give an alarm may because of affecting the check.
If you want the tool to move along the circle, it should be instructed by segment.

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

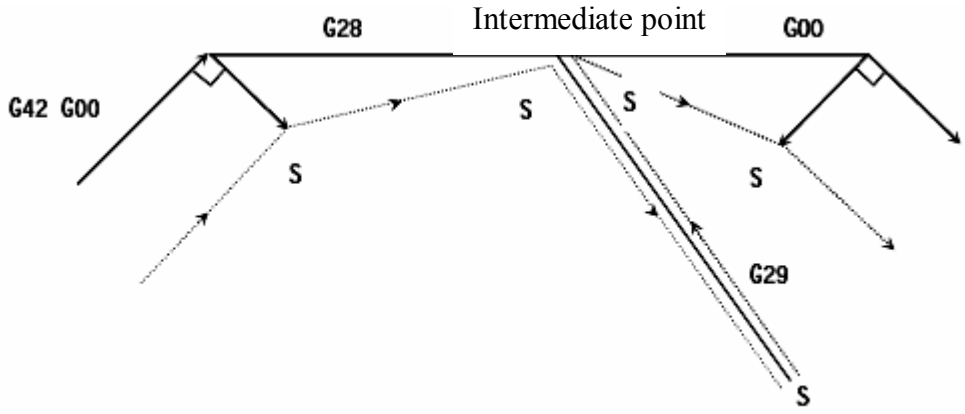
In compensation mode, if it is instruction G28, the compensation will be cancelled at intermediate point and the compensation mode is resumed automatically after the reference point is returned.



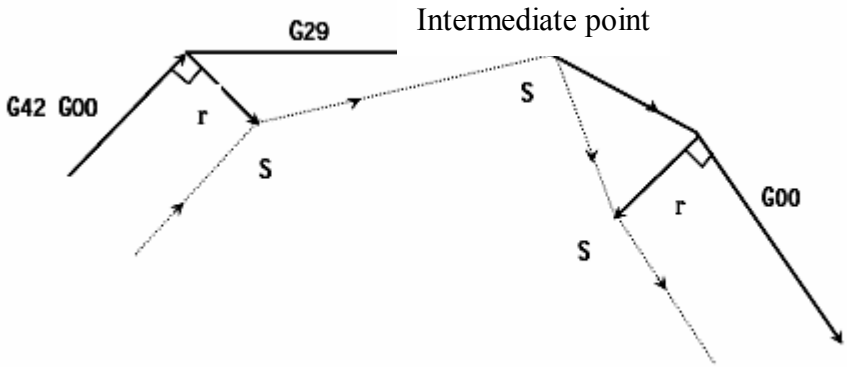
➢ G29 returns from reference origin automatically

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

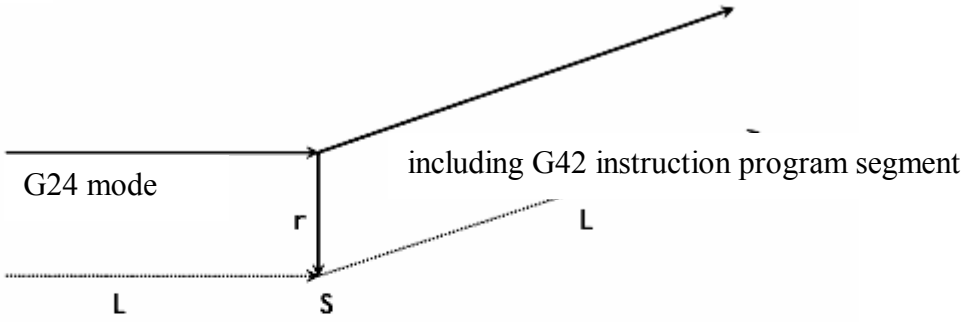


g. Tool radius compensation G code in compensation mode

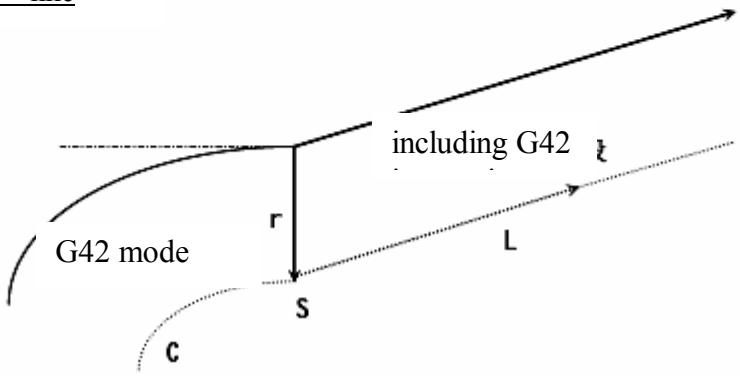
In compensation mode, when appointing tool radius compensation G code (G41, G42), it will form a vector that may form right angle with previous program segment, and it is nothing to do with the processing inside and outside. However, if you appoint this G code in circular arc instruction, you will not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

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

Line---line



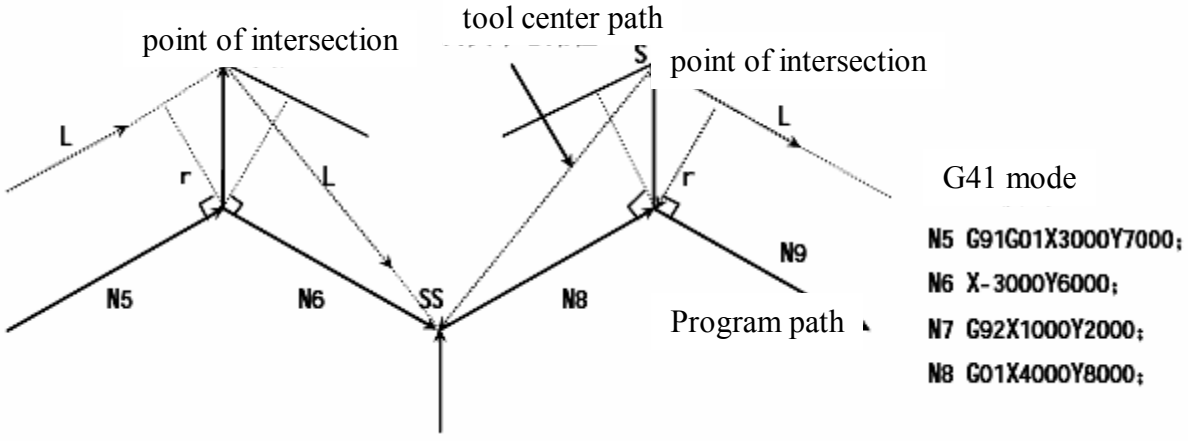
Circular arc---line



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

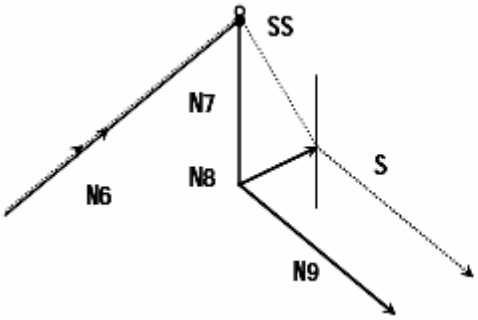
There is no tool movement in the following program segments. In these segments, the tool would not move even there is crossing point in tool radius compensation mode.

(1) M05:…… M code output
(2) S21:…… S code output
(3) G04 X10000:…… Pause
(4) （G17）Z100:…… No moving instruction in compensation plane
(5) G90:…… Only G code
(6) G01 G91 X0:…… Movement is 0

Not move

➢ Instructions at the beginning of compensation

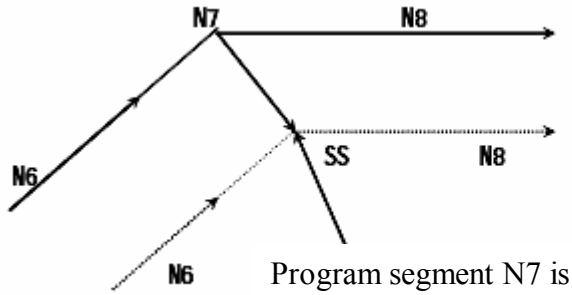
If there is no tool movement for the instructions at the beginning of compensation, it will not generate the compensation vector.




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

➢ Instructions in compensation mode

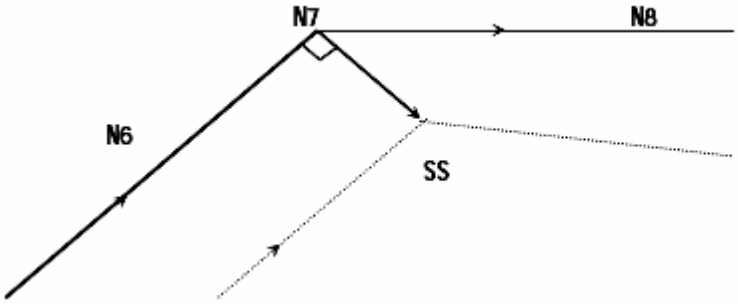
When only one program segment without tool movement is instructed in compensation mode, the vector and tool center path are the same as those when without this program segment.(refer to item (3) Compensation mode) At this time, the tool movement program segment is executed at the stop point of single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




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

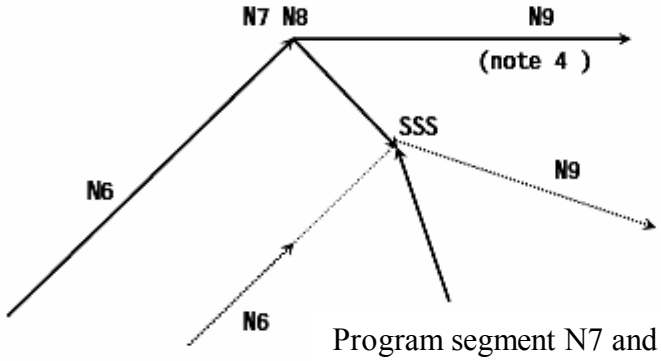
Program segment N7 is executed here

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.



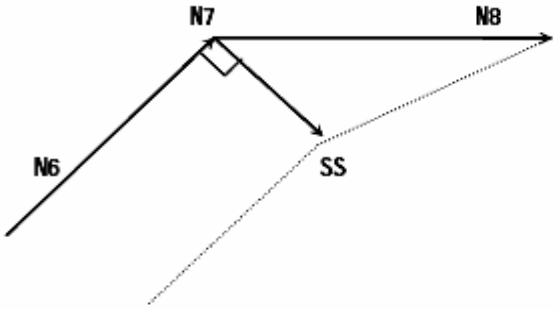
N6 G91X100.0 Y 200.0 ;
N7 S21 ;
N8 G04 X1.0 ;
N9 X100.0 ;

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.

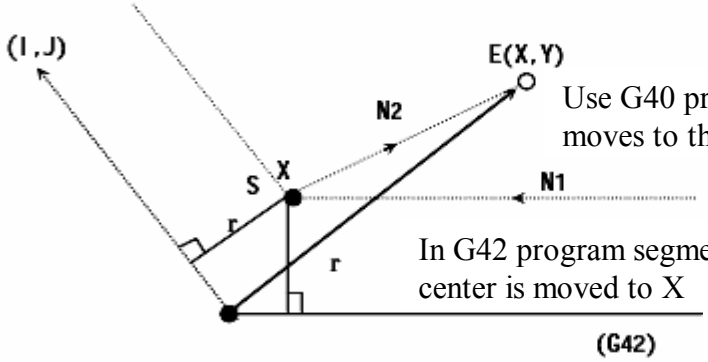


N6 G91X100.0 Y100.0 ;
N7 G40 ;
N8 X100.0 Y0 ;

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

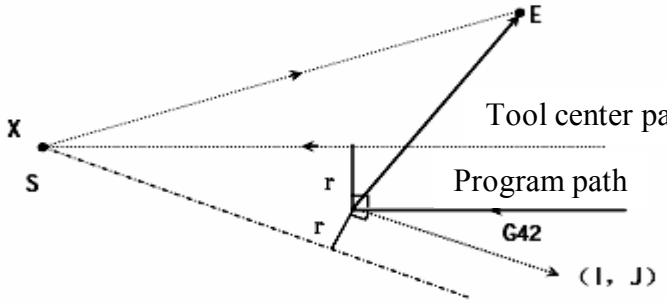
N1 (G42 mode)

N2 G40X□Y□I－J－;

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

Program path

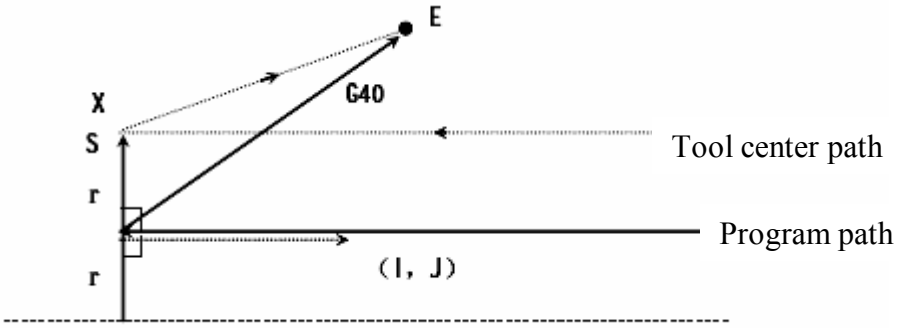
Note: the crossing point of tool path calculated by CNC has nothing to do with the appointed processing inside or outside.



Tool center path

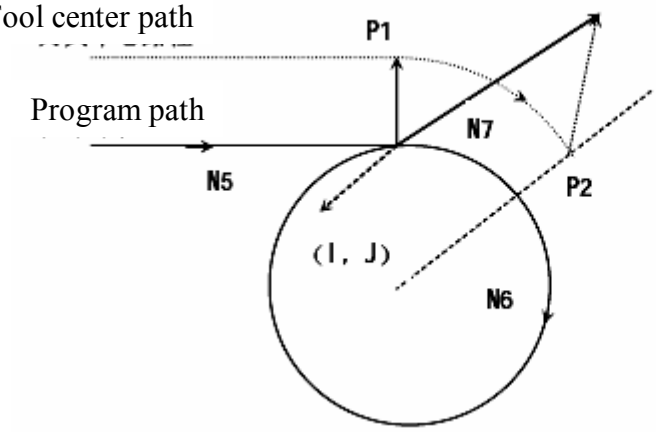
Program path

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

Tool center path



(G41)
N5 G01 G91 X10000;
N6 G02 J-6000;
N7 G40 G01 X5000Y 5000 I-100 J-100;

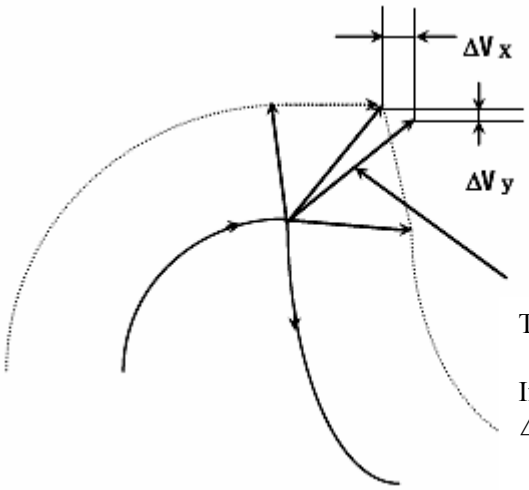
In the above picture, the tool center path is not moved along the circle but the circular arc from P1 to P2.

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.

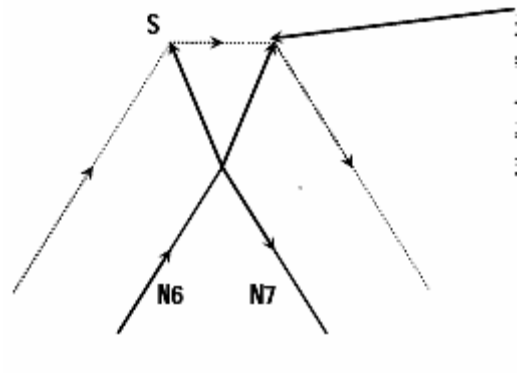


This vector is neglected.

If △Vx≤△V limit
△Vy≤ △V limit

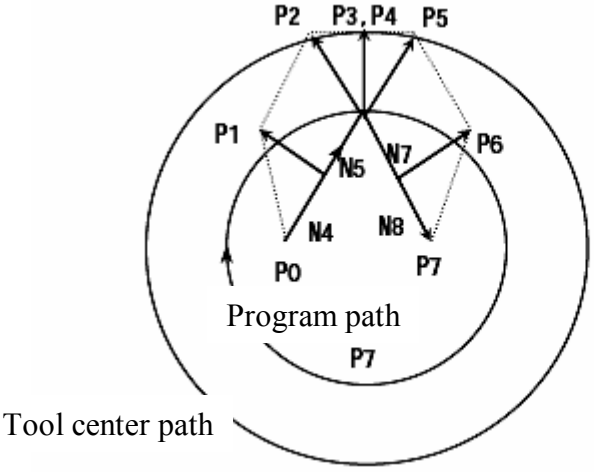
If △VX≤△V limit and △VZ≤ △V limit, the later vector is neglected. △V limit uses the parameter.

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;

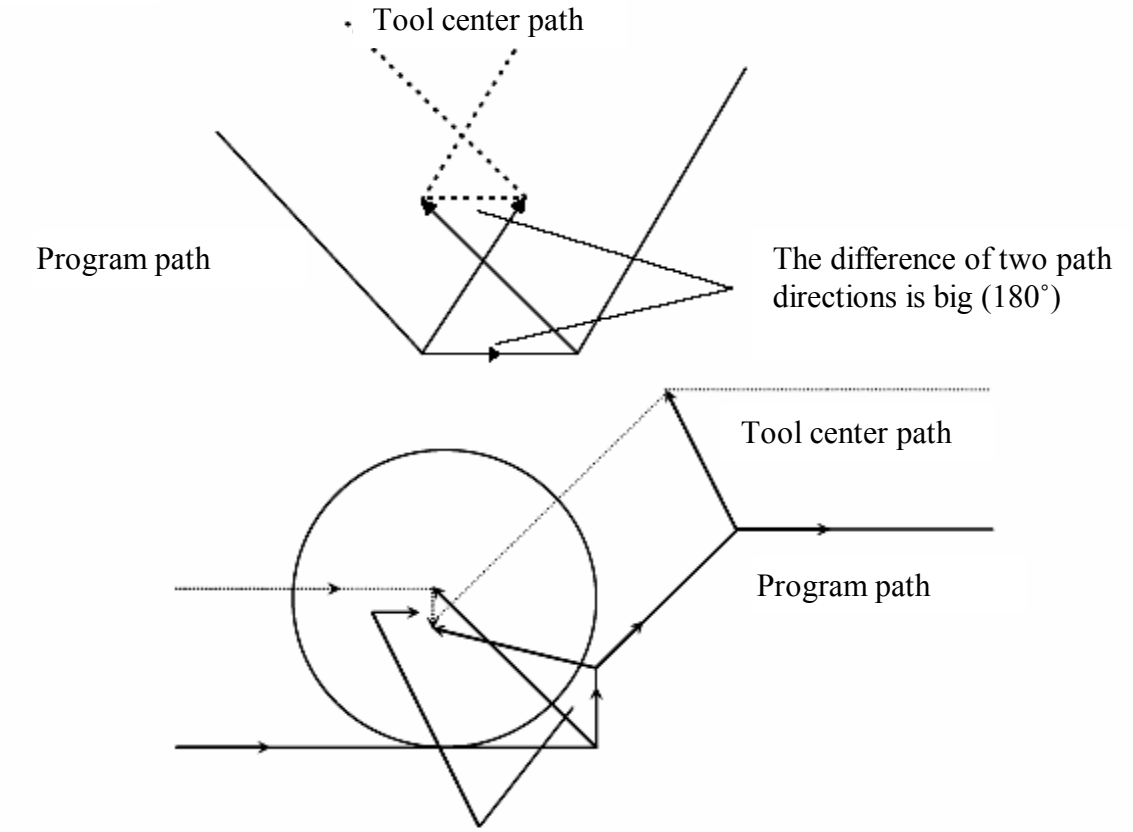
thIf the vector is not neglected, the tool path is as follows:
P0→P1→P2→P3 (Circular arc)→P4→P5→P6→P7
But if the distance between P2 and P3 is neglected, the P3 will be neglected. The tool path is as follows:
P0→P1→P2→P4→P5→P6→P7, circular arc cutting of program segment N6 is neglected.

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.

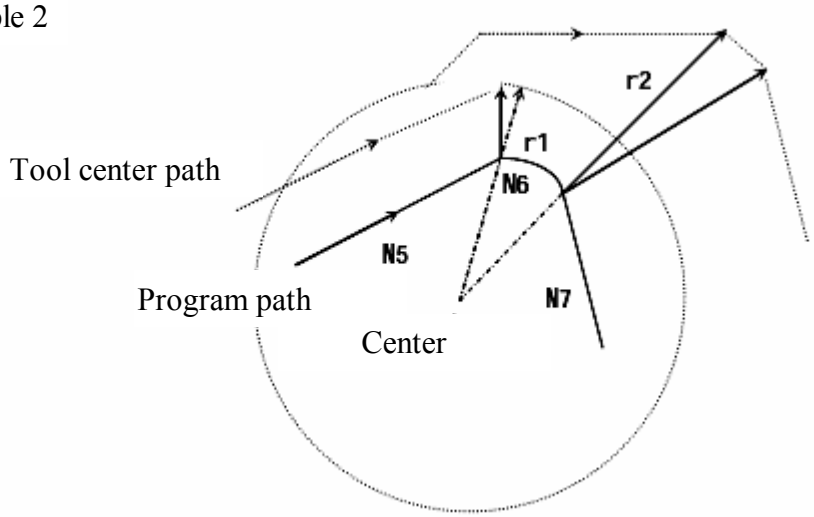
- Basic conditions of interference
  - The tool path direction is different from that of program path. (the inclination of paths is 90˚-270˚.)
  - When processing circular arc, apart from the above conditions, the inclination of starting point and end point of tool center path has a great difference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

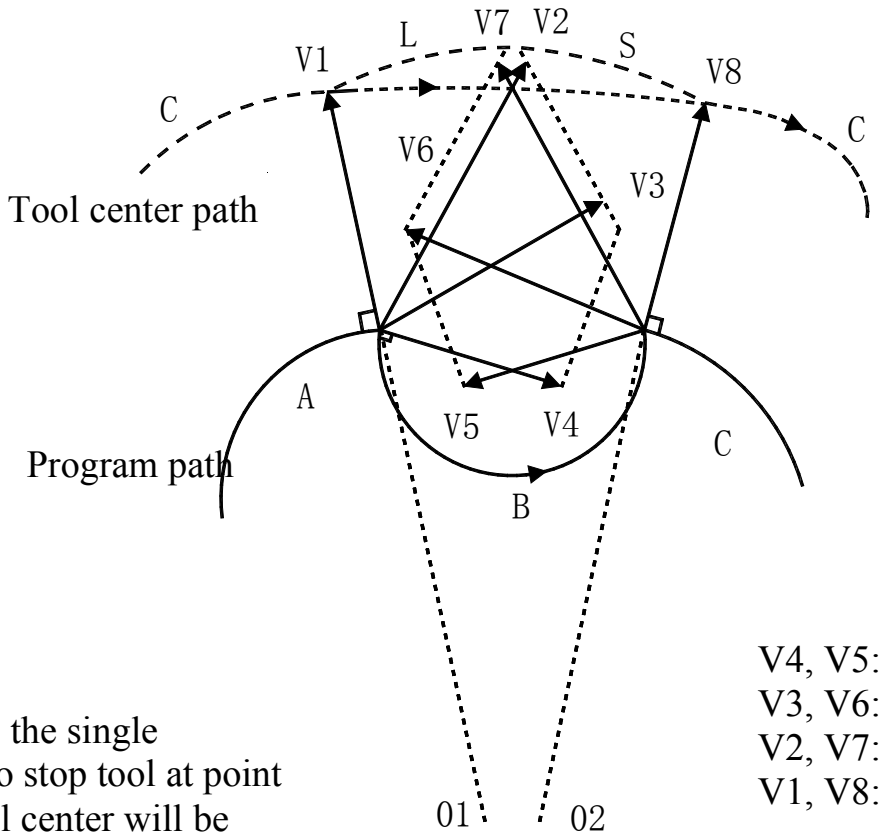
When tool compensation program segment A, B and C are executed, it will produce vector V1, V2,V3, and V4 between A and B, and V5, V6, V7, and V8 between B and C. First, check the latest vector. If there is interference, they will be cleared automatically. If the vector to be neglected is at the end of 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.

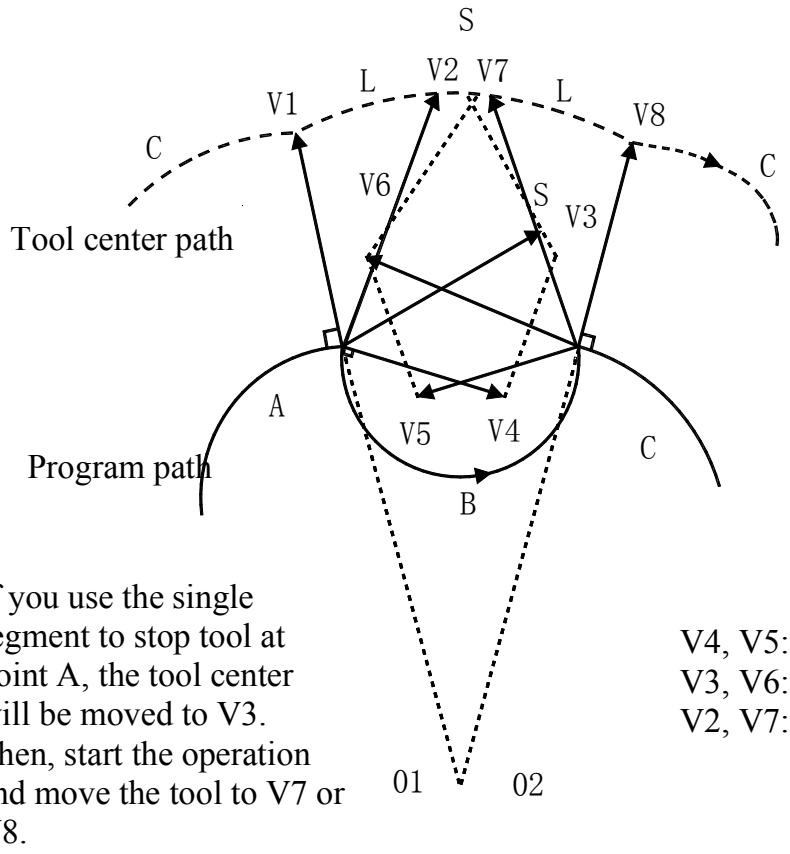
(Example 1) tool moves from V1 to V8 in straight line



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

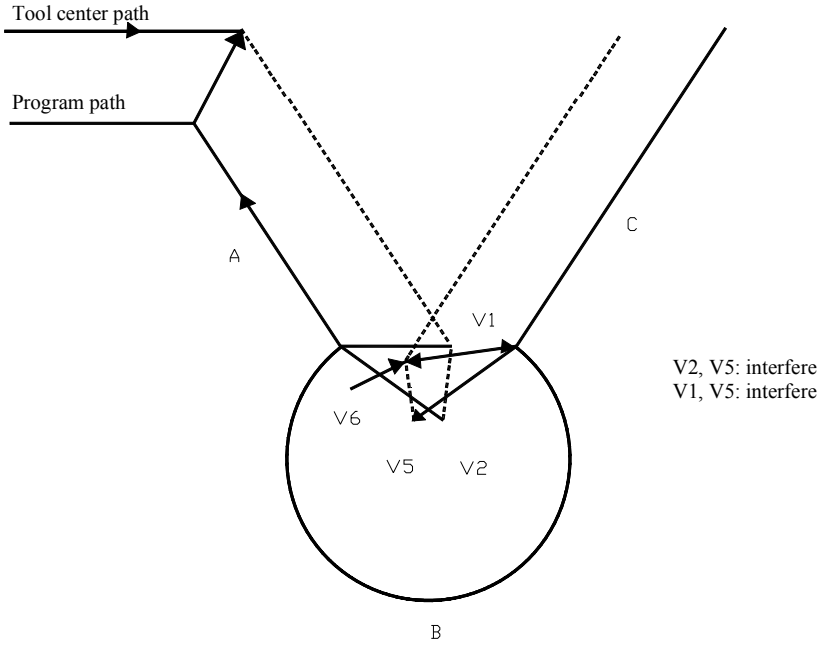
V4, V5: interfere
V3, V6: interfere
V2, V7: interfere
V1, V8: Not interfere

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



- ✧ If 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).

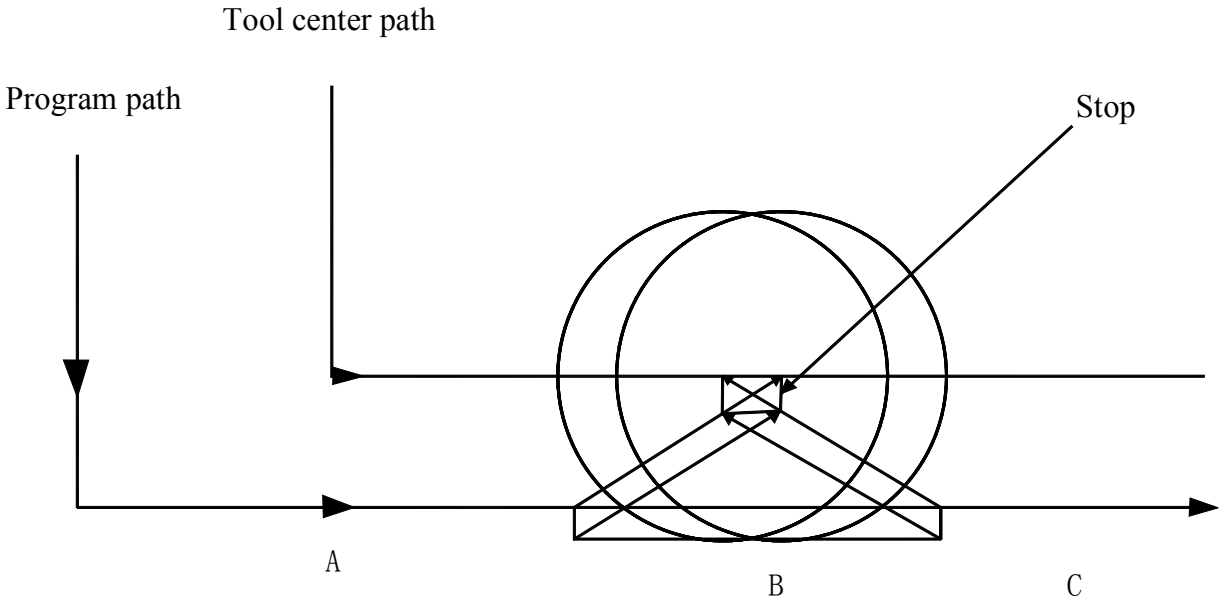
(If executed by single program segment, the tool will stop at the end of the program segment.)



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.

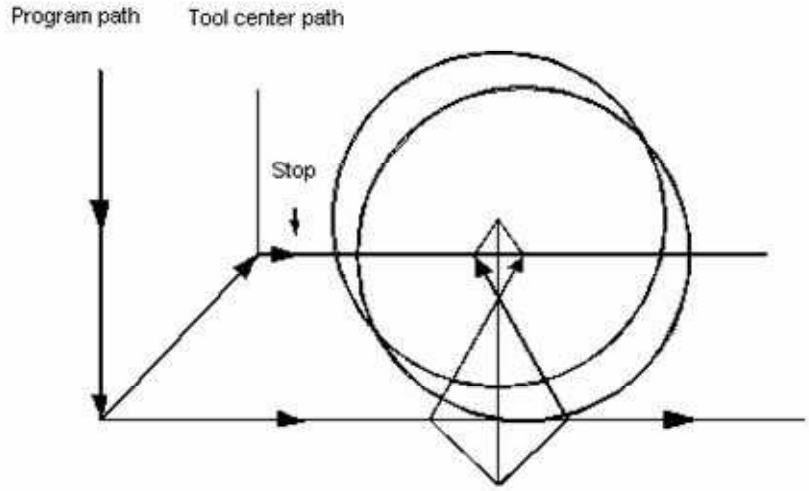
➢ In fact, there is no interference, but doing the interference check
For example:

✧ Recess depth is smaller than compensation



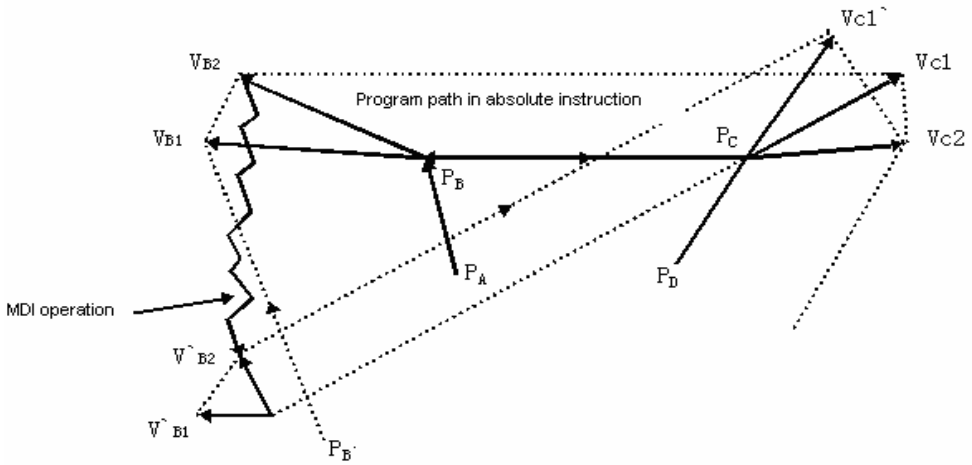
Actually, there is no interference, but because the tool is in program segment B, the program direction is opposite to the path of tool radius compensation, the tool stops and shows an alarm.

✧ 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.

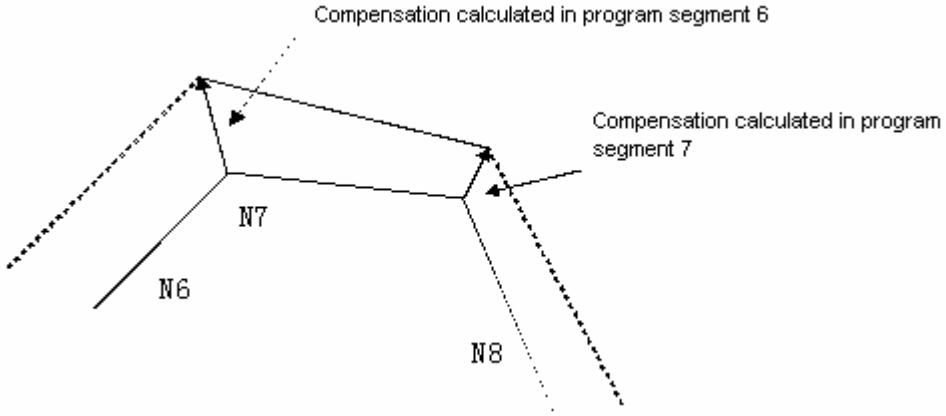
p. Notices for compensation

■ 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

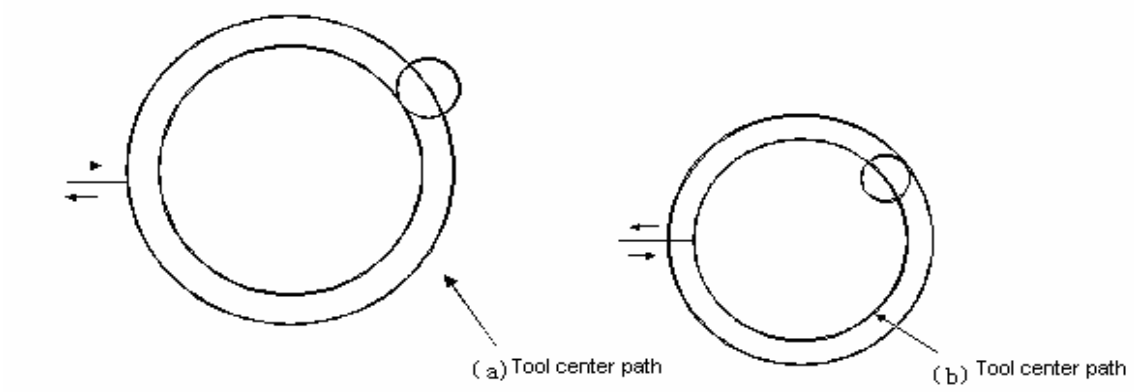
Usually when changing tool, the compensation should be changed in cancellation mode. If compensation is changed in compensation mode, calculate the new compensation at the end point of program segment.



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.

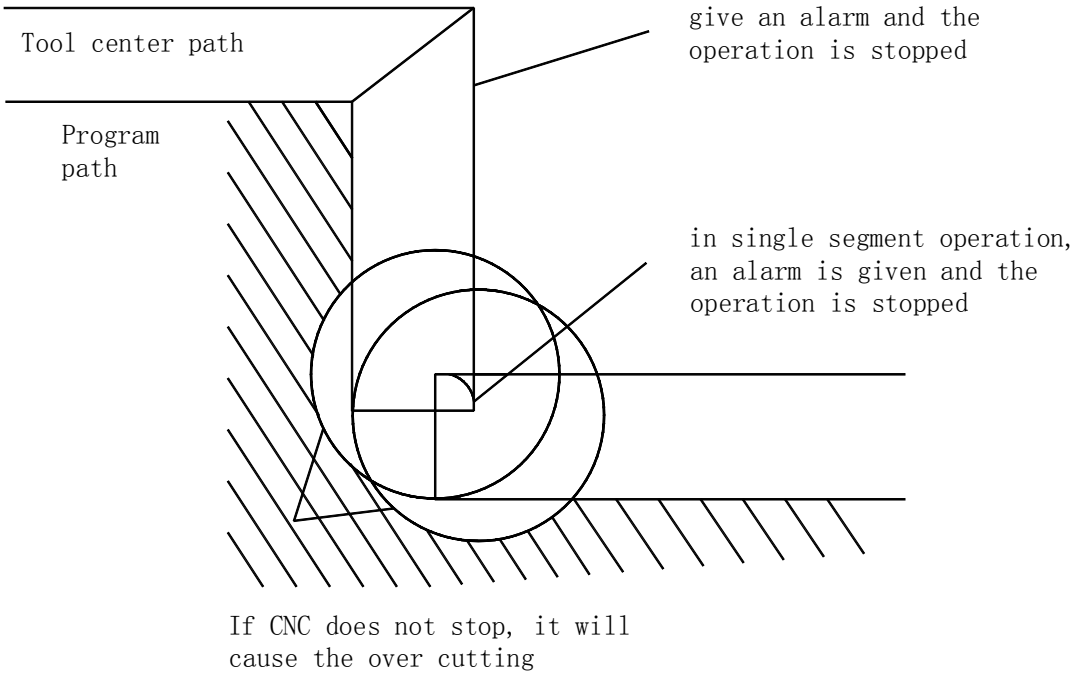
Generally, the compensation is (+) when programming. When tool path is programmed as picture (a), if the compensation is (-), the tool center movement is as picture (b), and vice versa. In this way, the same program can be cut into male or female type, and the gap between them can be adjusted by choosing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



■ Compensate over cutting by tool radius

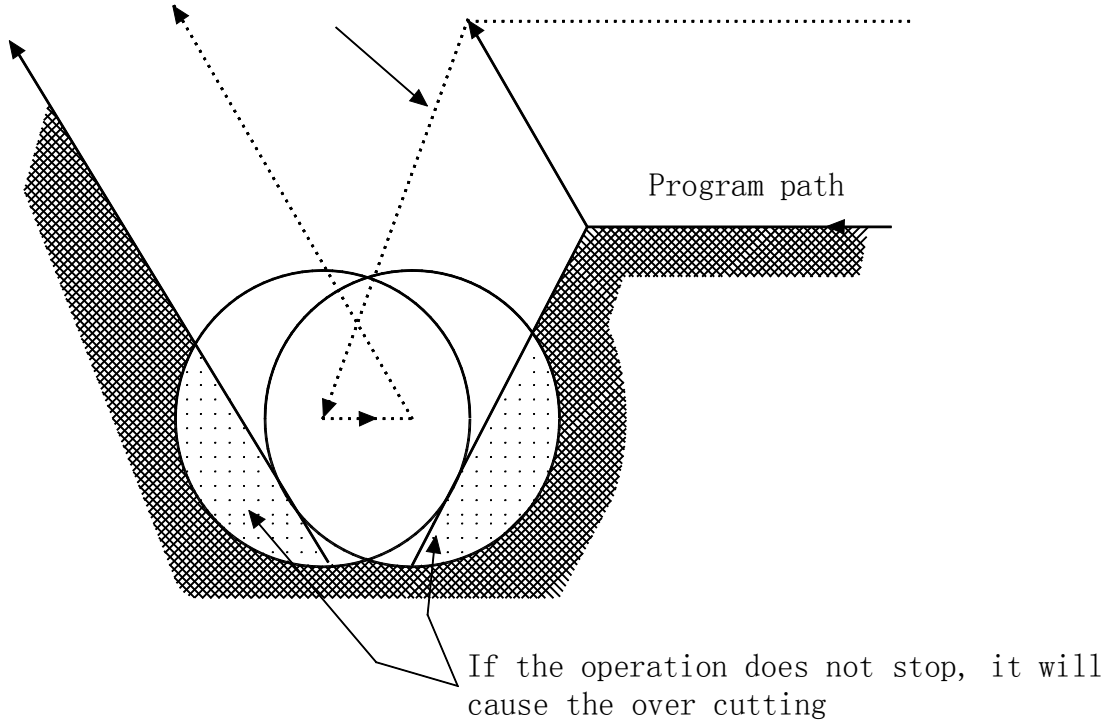
◆ when processing by circular inside of smaller tool radius

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.



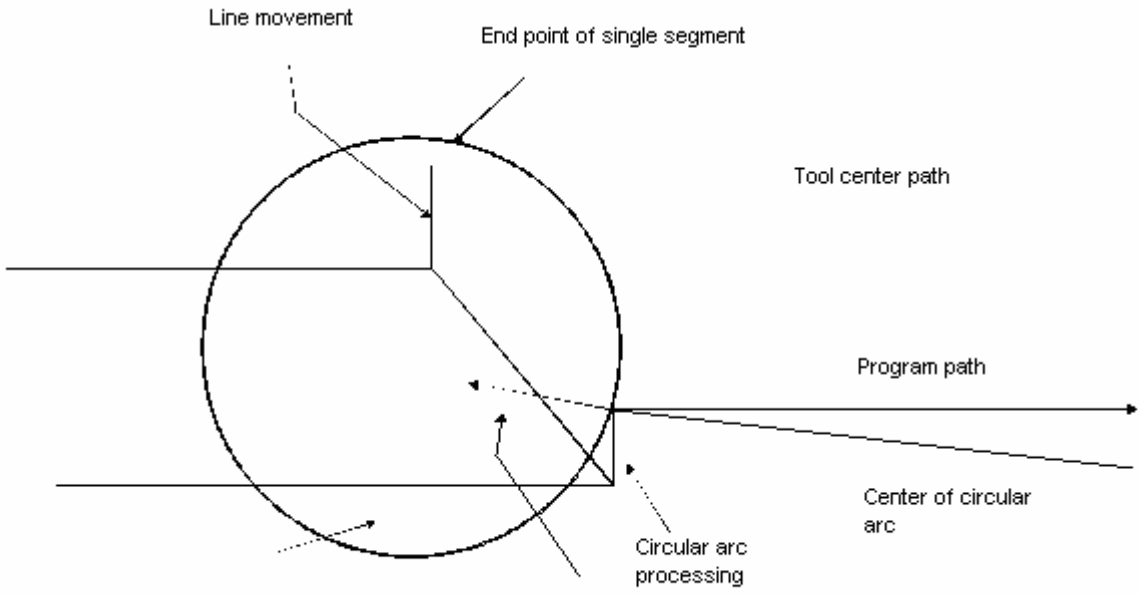
✧ when processing the cut-off trench smaller than radius of tool

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.

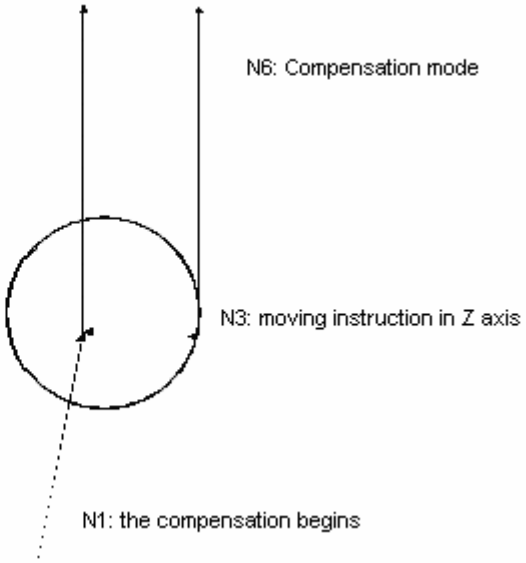


✧ If the initial vector of tool is not neglected, it will generate the over cutting

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:

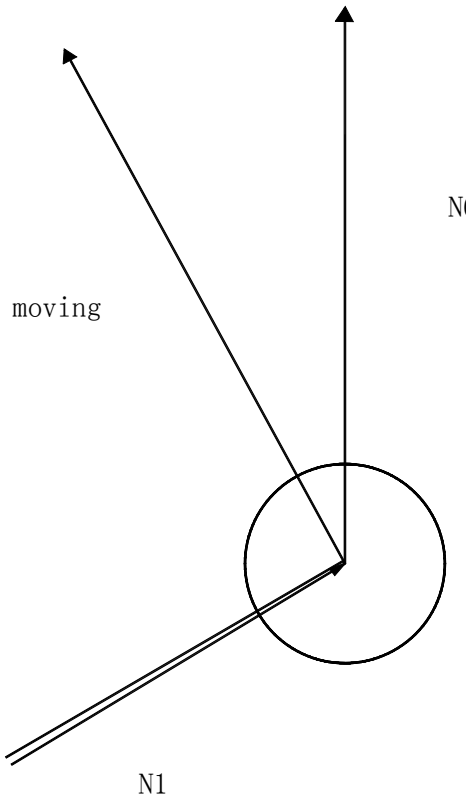
```
N1 G91G00X50000Y50000H01:
N3 G01Z-30000F1:
N6 Y10000F2:
```

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:

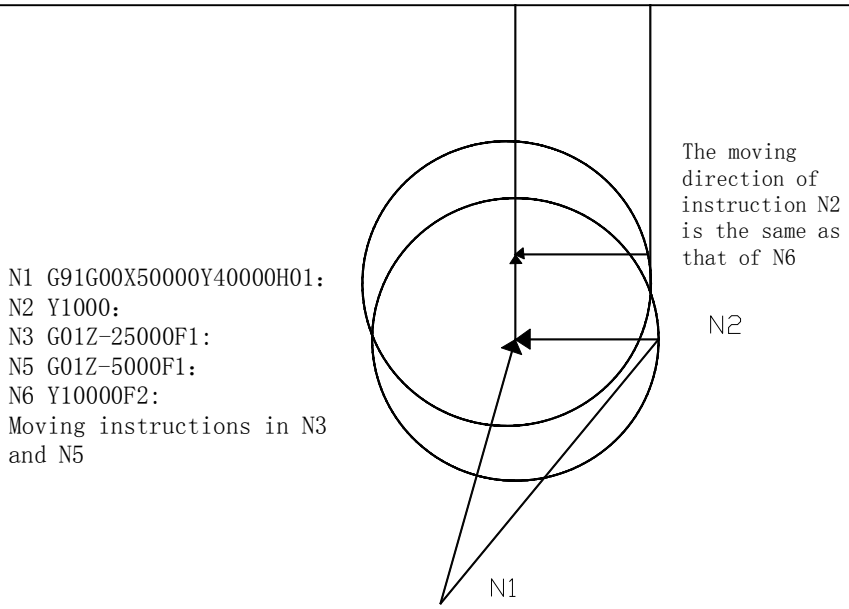
```
N1 G91G00X50000Y50000H01:
N3 Z-250000:
N5 G01Z-5000F1:
N6 Y10000F2:
```



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.

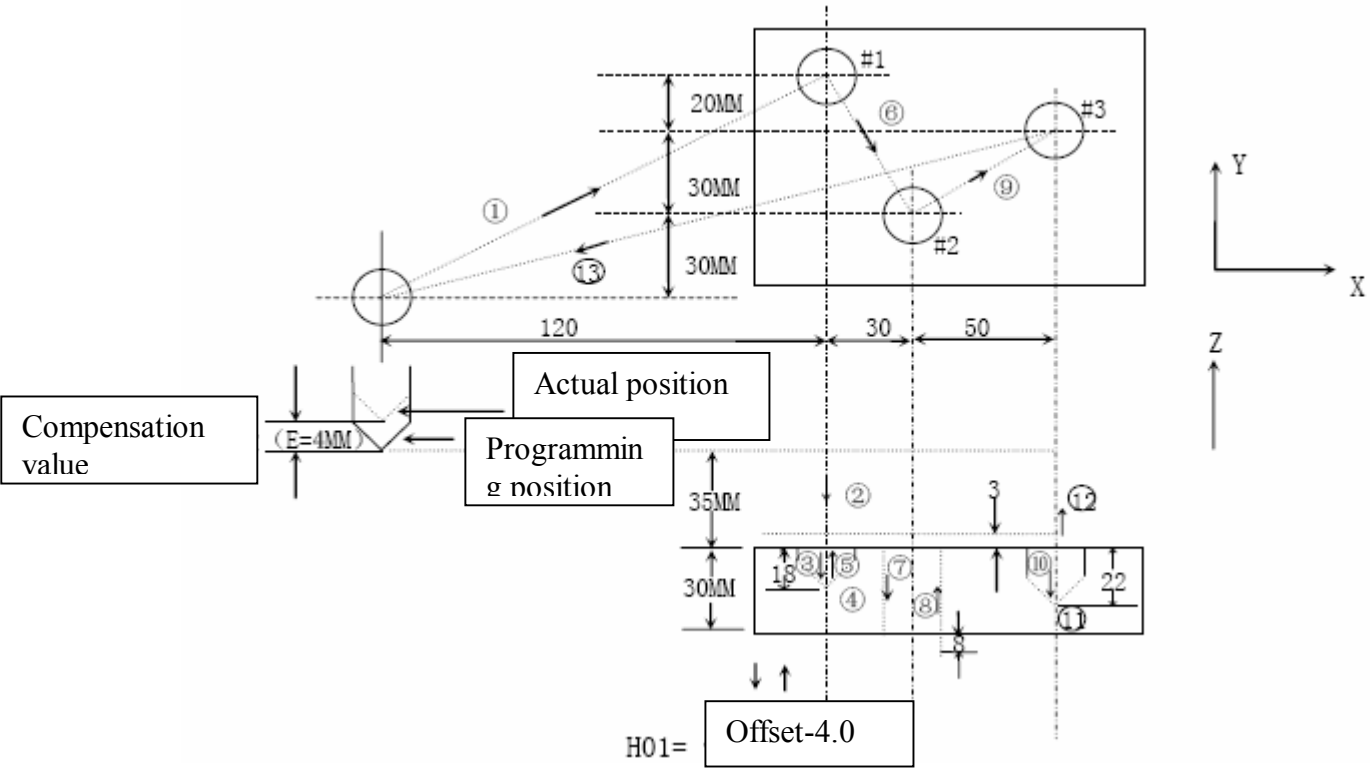
Enter the cuter compensation menu, and preset the corresponding offset number in offset storage.

	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)

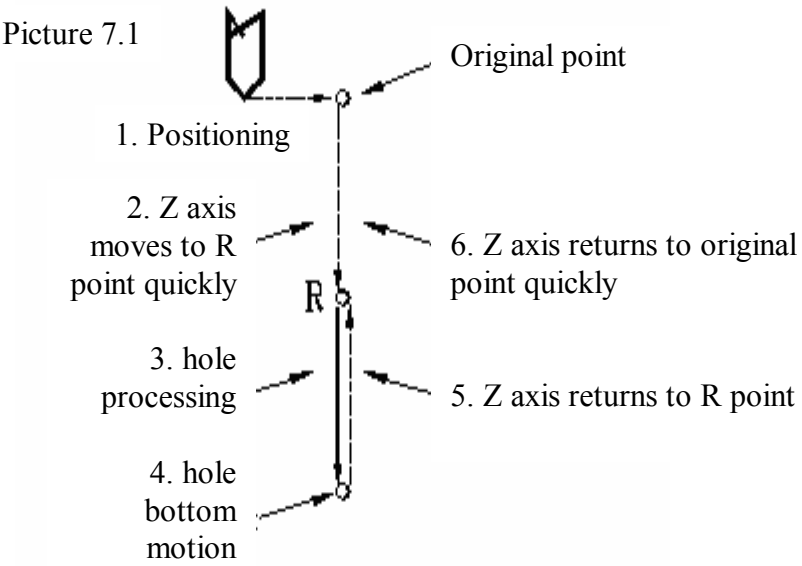
Fixed cycle of hole processing allows functions that should be done with many program segments in other methods to be done in just one program segment. Table 7.1 lists all fixed cycles of hole processing. Generally, one fixed cycle of hole processing finishes the following 5 operations (see Picture 7.1):

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

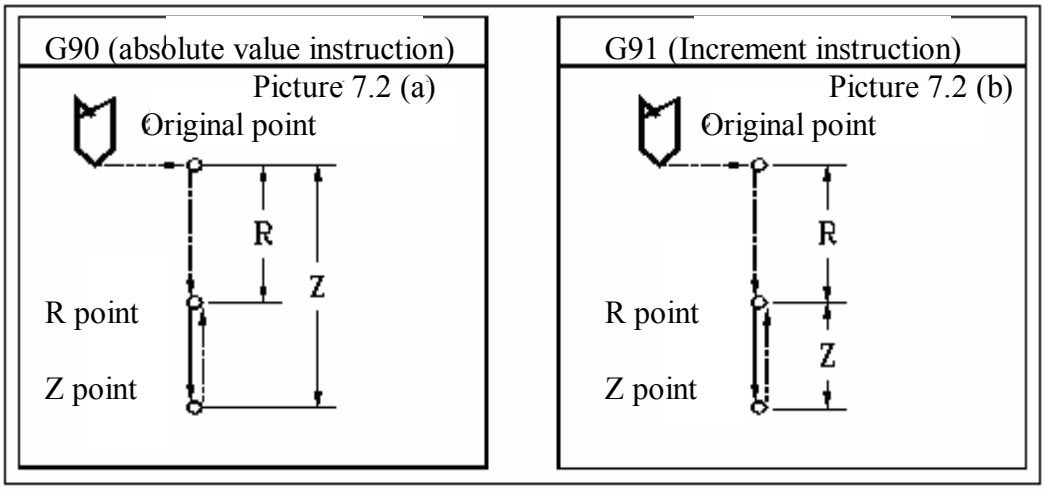
Table 7.1 Fixed cycle of hole processing

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

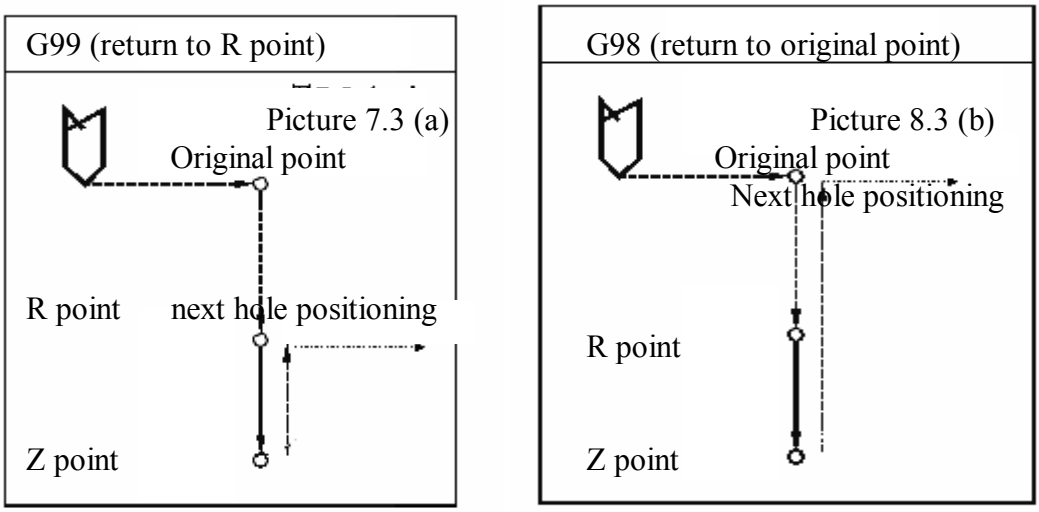


The instructions that influence the execution of instruction for hole processing fixed cycle include G90/G91 and G98/G99 instructions. Picture 7.2(a) and 7.2(b) show the influence of G90/G91 on instruction for hole processing fixed cycle.



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:

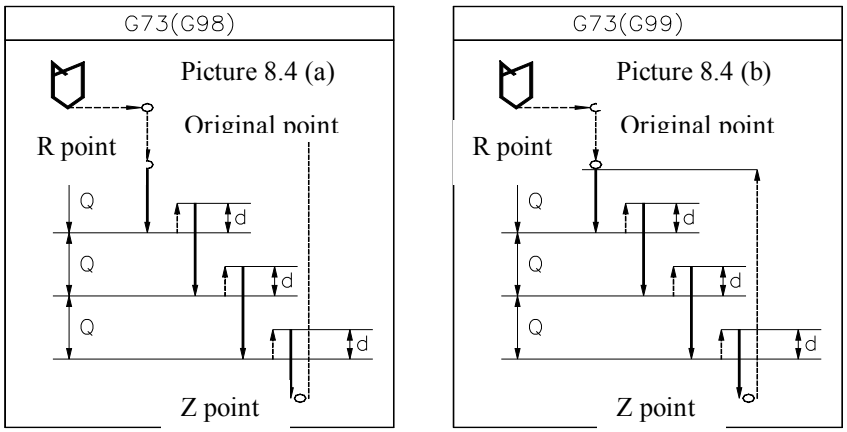
Move at quick feeding rate ·····→

Move at cutting feeding rate ——→

Manual feed - - - - - - - - - - →

➢ G73 (High-speed deep hole drilling cycle)

Format: G73 X_ Y_ Z_ R_ Q_ F_

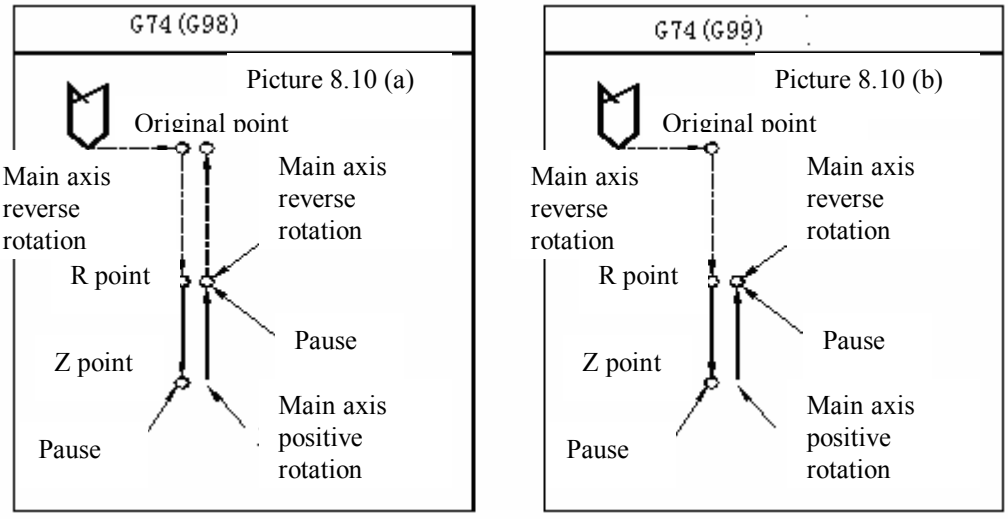


In high-speed deep hole drilling cycle, the feed from R point to Z point is done by segment. After each cutting feed, Z axis will be uplifted a certain distance before performing the cutting feed of next segment. The uplift distance of Z axis is d, which is specified by 531# parameter. The depth of each 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_



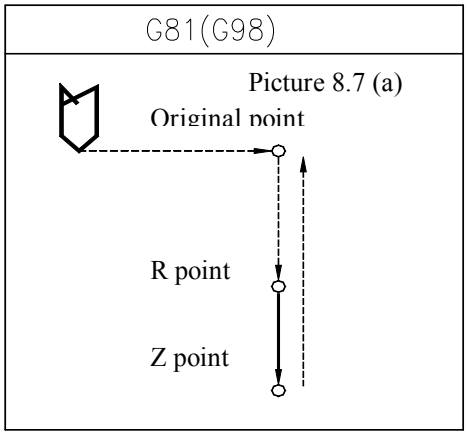
Note: In G74 and G84 cycles, the function of feed rate override and feed hold will be neglected, which means the feed rate will be kept at 100%. It should not be stopped in midway before a fixed cycle is executed. Before the cycle, it is required to instruct to rotate in main axis tapping direction.

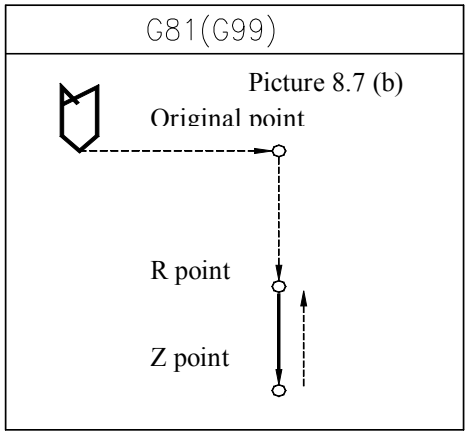
G80 (Cancel fixed cycle)

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_

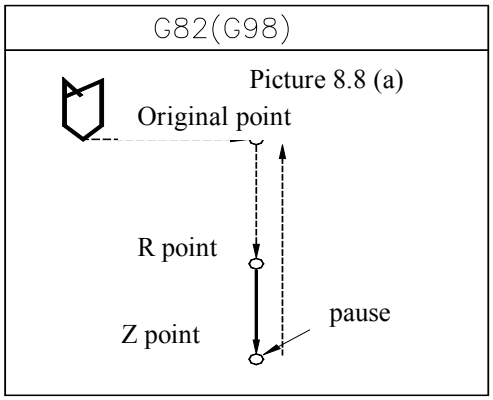


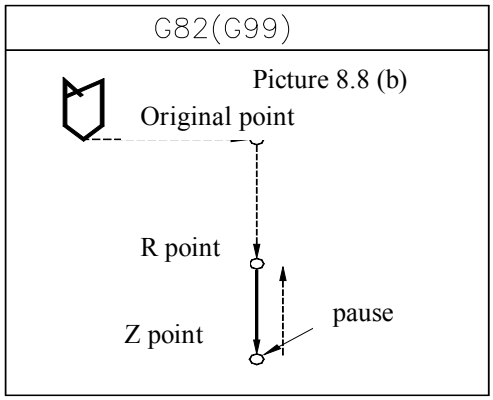


G81 is the most simple fixed cycle, it is executed as follows: X, Y locating, Z axis moved to R point quickly and fed to Z point at F speed, and then returned to original point (G98) or R point (G99) quickly, without hole bottom action.

G82 (Drilling cycle, rough boring cycle)

Format: G82 X_ Y_ Z_ R_ P_F_



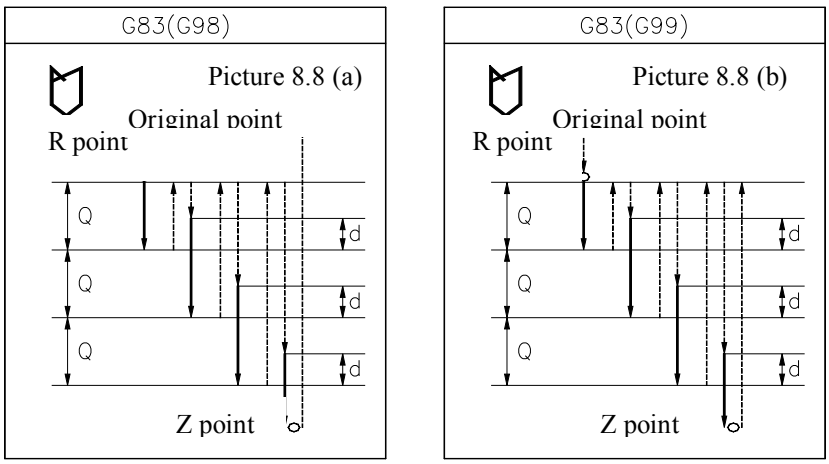


G82 has a pause action at the hole bottom, and apart from this, it is the same as G81. The pause at the bottom of hole can improve the precision of hole depth.

G83 (Deep hole drilling cycle)

Format: G83 X_ Y_ Z_ R_ Q_ F_

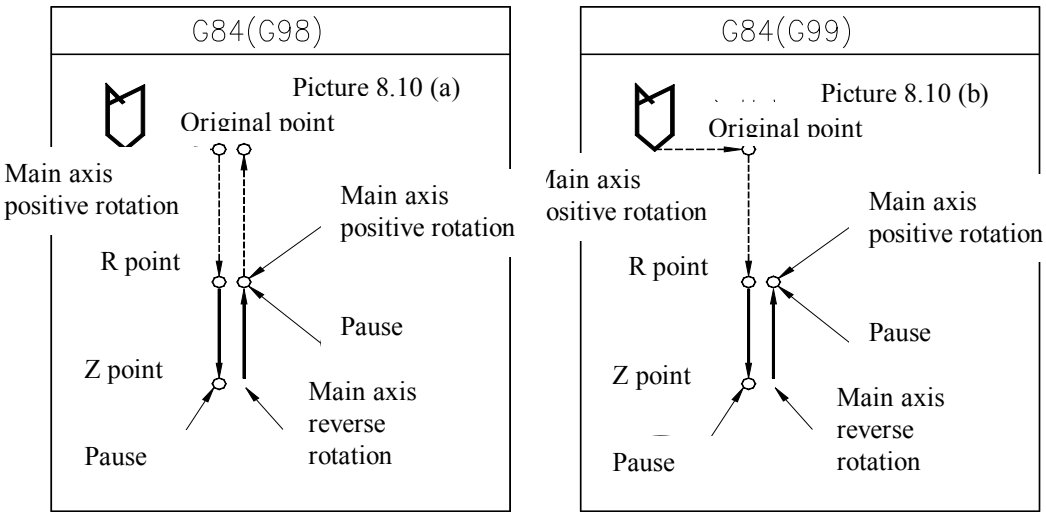
Similar to G73 instruction, the feeding from R point to Z point in G83 instruction is also done by segment. The difference is that after the feed of each segment, Z axis is returned to R point, and then moved at quick feeding rate to d above the feeding origin of the next segment and start the feeding 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_

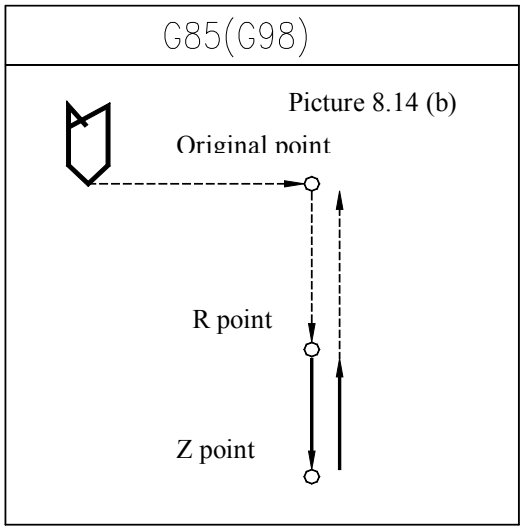


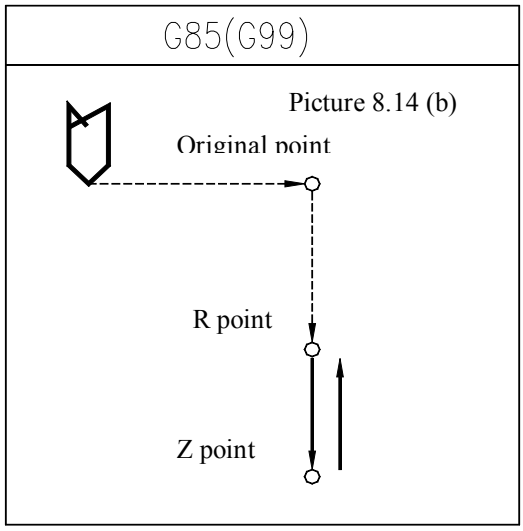
Note: In G74 and G84 cycles, the function of feed rate override and feed hold will be neglected, which means the feed rate will be kept at 100%. It should not be stopped in midway before a fixed cycle is executed. Before the cycle, it is required to instruct to rotate in main axis tapping direction.

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.

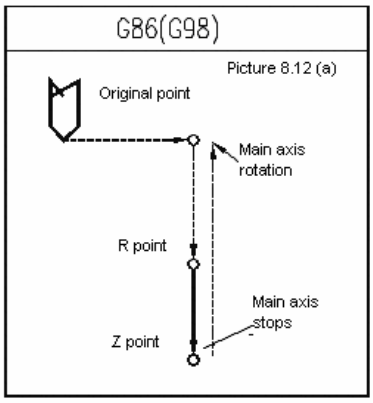


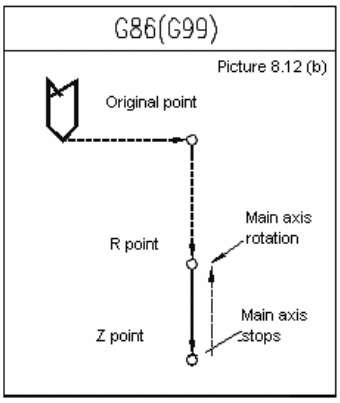


➢ G86 (Boring cycle)

Format: G86 X_ Y_ Z_ R_ F_

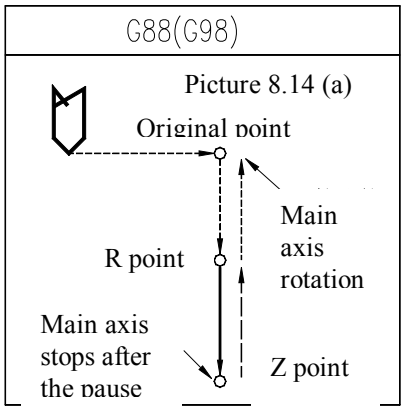
This fixed cycle is executed similarly to G81. what is different is that in G86 when the tool is fed to hole bottom, the main axis will stop, and when it returns to R point or original point quickly, the main axis will rotate at the original speed in the same direction.

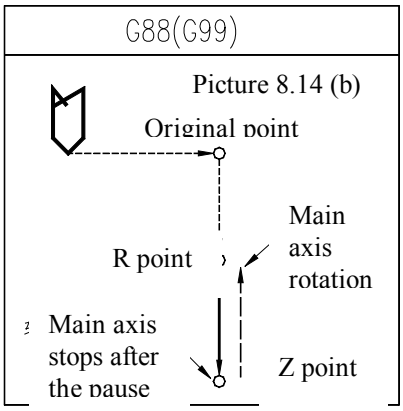




G88 (Boring cycle)

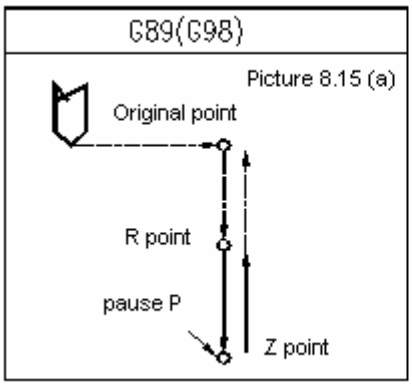
Fixed cycle G88 is provided with manual return function for fixed cycle of drilling (as picture shows):

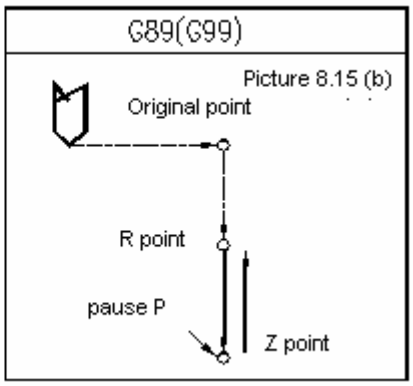




G89 (Boring cycle)

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.

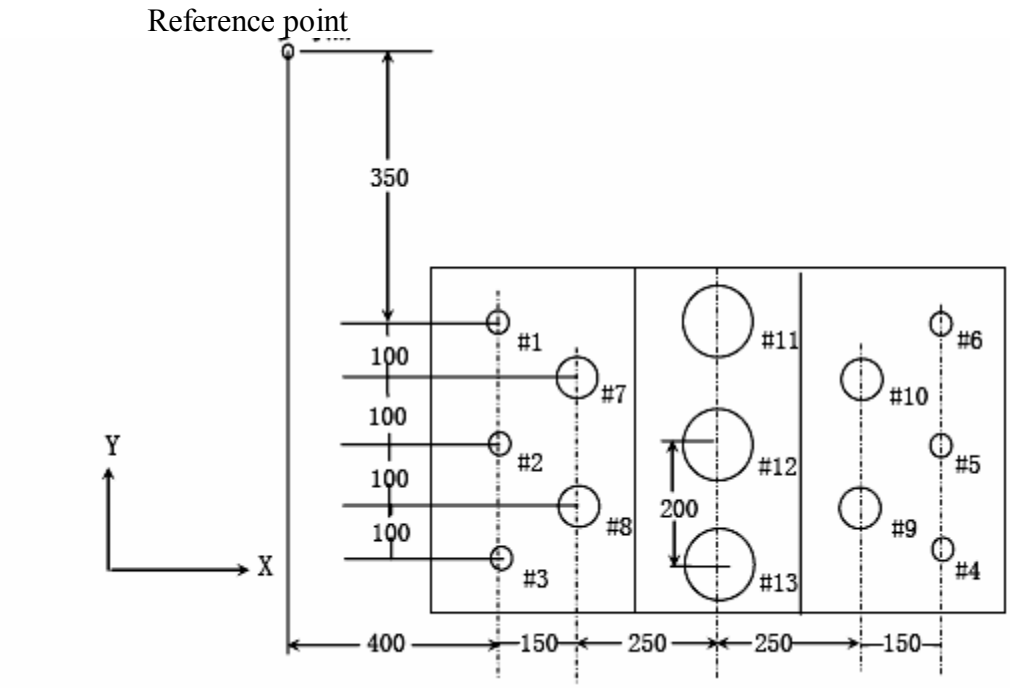
f. If an M code is specified in program segment for executing fixed cycle, M code will be executed at the same time when the fixed cycle is executing the positioning. The signal of completion of M instruction execution will be given out as Z axis returns to R point or the original point. When using K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

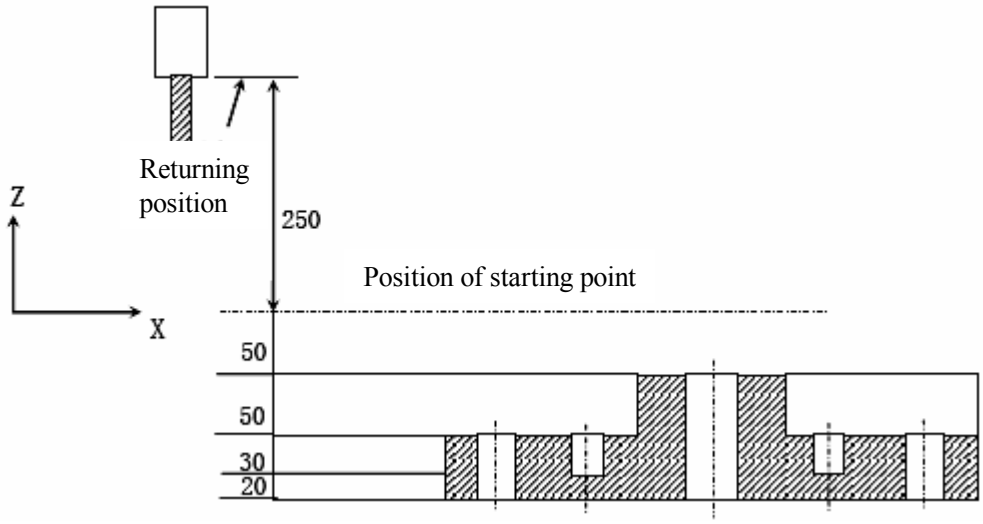
h. When the switch of single program segment is set in upper position, the fixed cycle will stop after executing X, Y positioning, feeding to R point quickly and returning from hole bottom (to R point or original point). This is to say you have to press the cycle activate button for 3 times to complete 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.

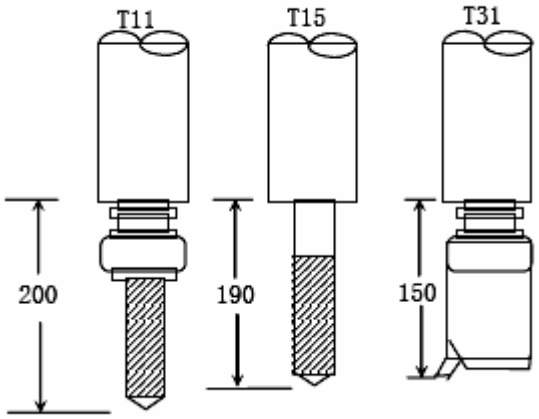
i. When executing G74 and G84 cycles, if you press feed holding button when Z axis is moving from R point to Z point and from Z point to R point, the feed holding indicator will be on immediately, but the machine would not stop and go to holding state until Z axis returns to R point. In addition, in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

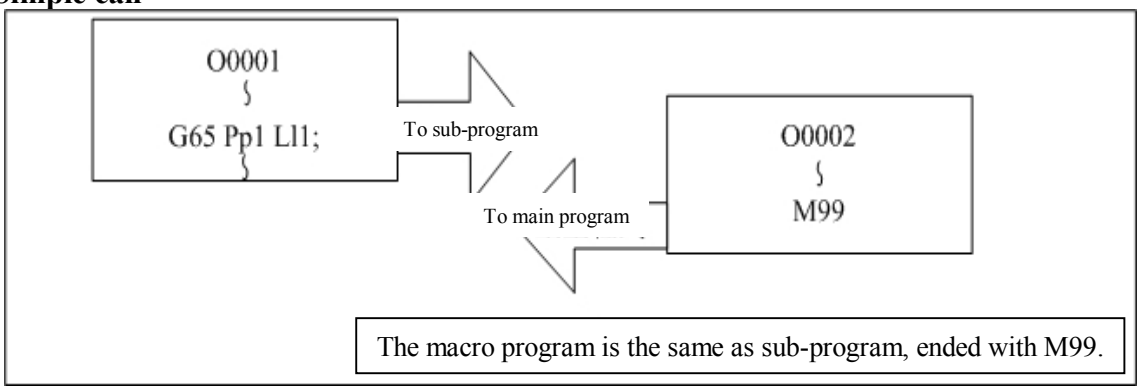
1) Specified by G66 (or G661) instruction, and before G67 (Cancel) instruction, the macro program specified after the execution of single segment with move instruction (or each single segment) will be called.
2) In the same program, G66 (or G661), G67 instructions should be used correspondingly in pair.

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:

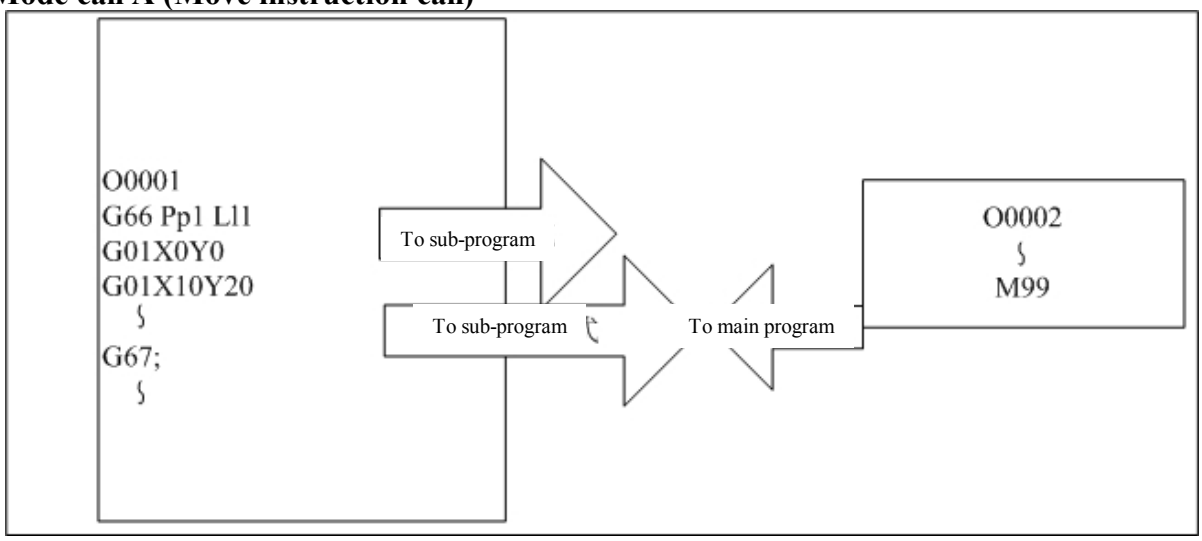
1) Except G,L,N,O,P, all addresses can be specified as argument.
2) Addresses that are not required to be transferred can be neglected.
3) The address information occurs in G65 instruction is considered as the argument of G65. For example: G65P0002N100G01G90X100.Y200.F400R1000,G01 instructions are not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

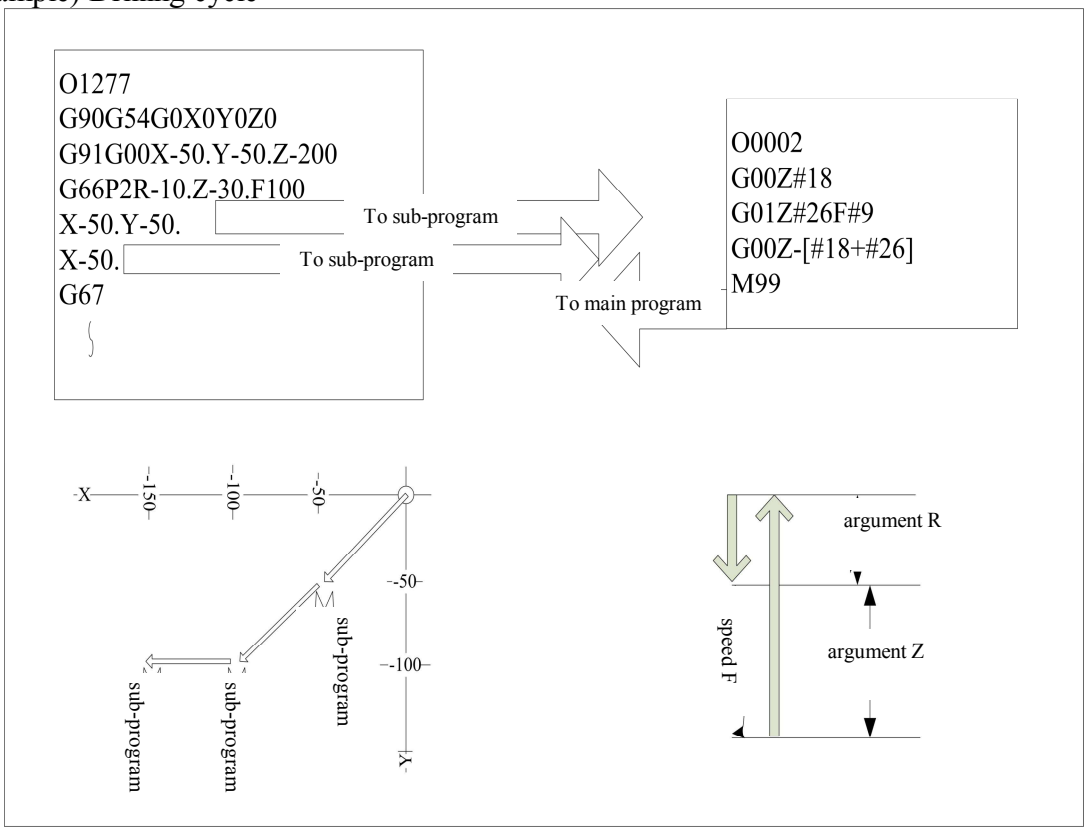
Format description:

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

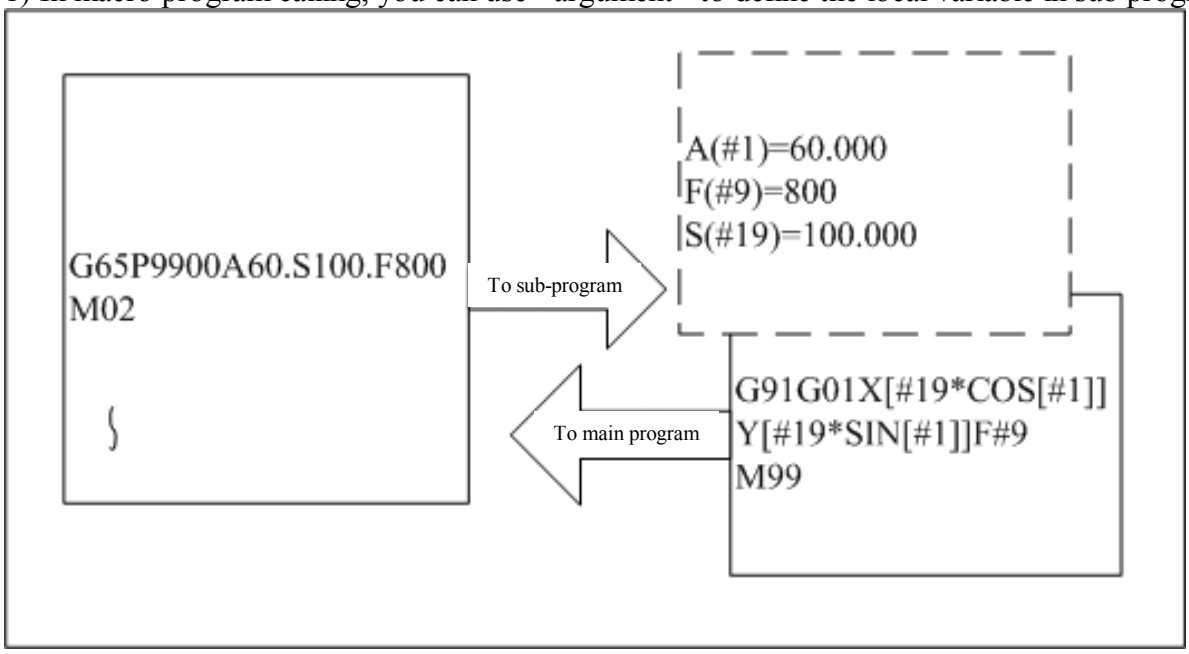
< argument > is Aa1 Bb1 Cc1…Zz1 etc.

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

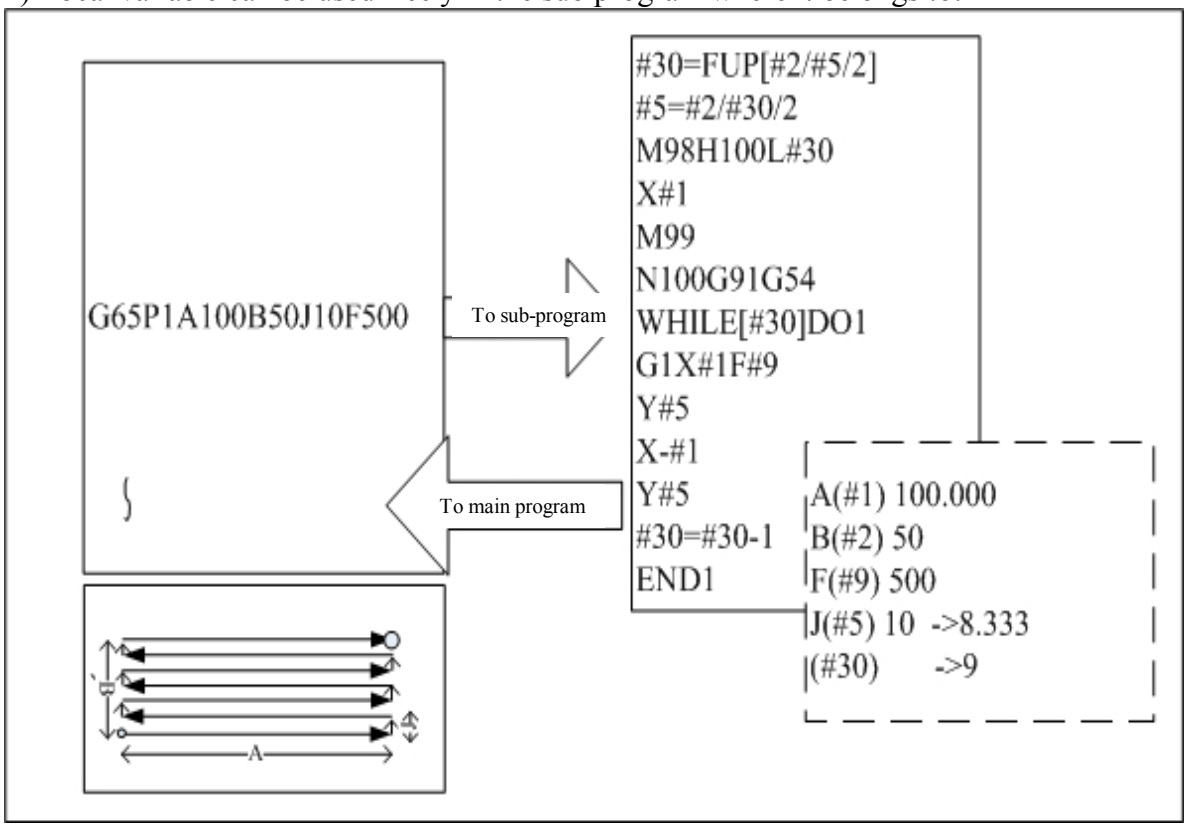
Address	Variable No.	Sub program	Address	Variable No.	Sub program
A	#1	○	N	×	×
B	#2	○	O	×	×
C	#3	○	P	×	×
D	#7	○	Q	#17	○
E	#8	○	R	#18	○
F	#9	○	S	#19	○
G	×	×	T	#20	○
H	#11	○	U	#21	○
I	#4	○	V	#22	○
J	#5	○	W	#23	○
K	#6	○	X	#24	○
L	×	×	Y	#25	○
M	#13	○	Z	#26	○

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.

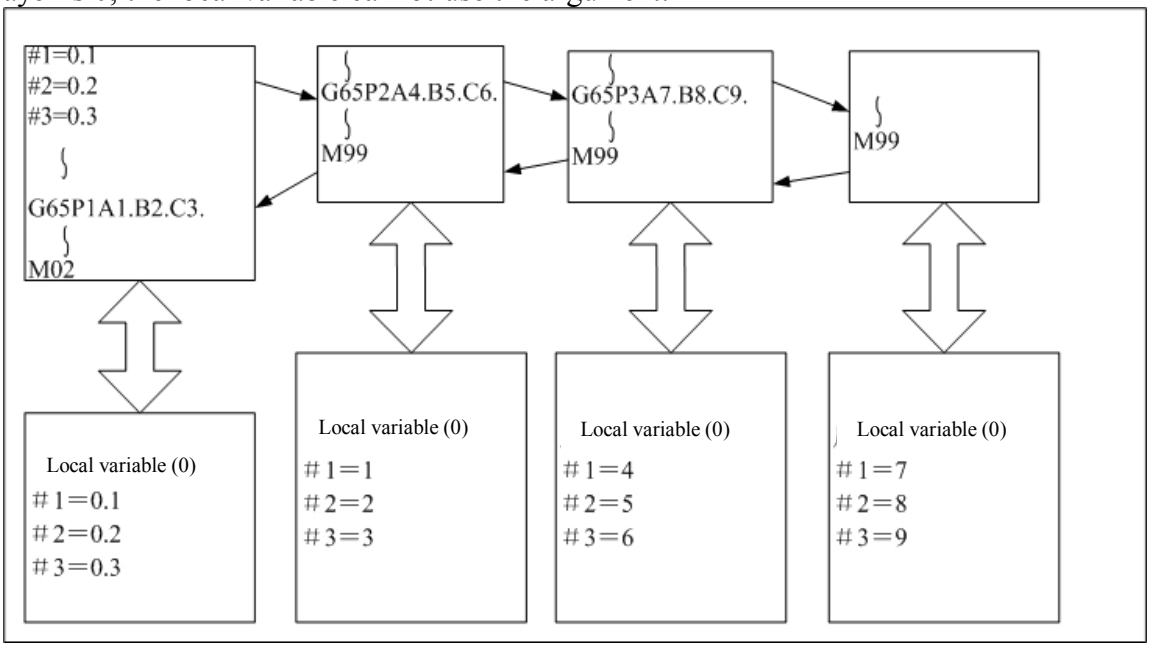


2) Local variable can be used freely in the sub program where it belongs to.



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.

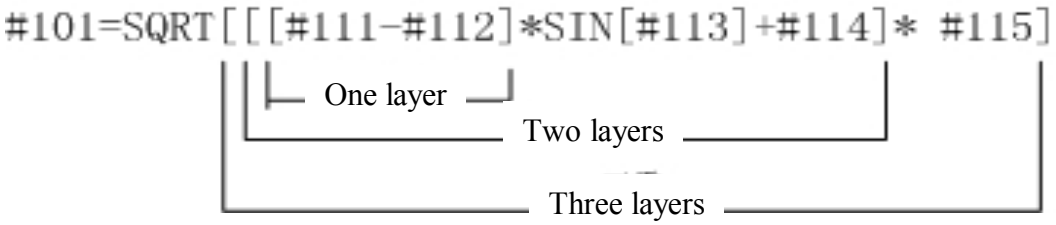
Calculation method	#i=#j	Definition, displacement
Addition & Subtraction	#i=#j+#k #i=#j-#k #i=#j OR #k or #i=#j|#k #i=#j XOR #k or #i=#j^#k	Addition Subtraction 32 bit OR calculation (logical sum) 32 bit XOR calculation
Multiplication & Division	#i=#j*#k #i=#j/#k #i=#j MOD #k #i=#j AND #k or #i=#j & #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.] #565=ATAN[1.732]	#561 60.000 #562 60.000 #563 60.000 #564 60.000 #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

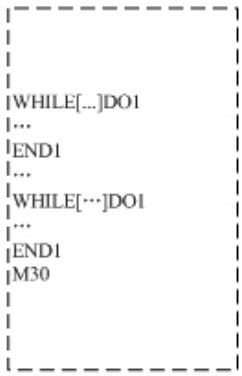
Instruction format:

WHILE [expression]DO m; (m=1,2,3…127)

…

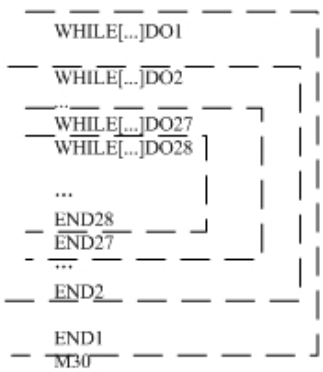
END m;

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.

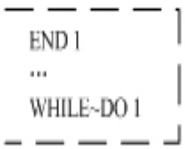
<table>
<tr>
<td>(1) the same identification number can be used repeatedly
 

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
 

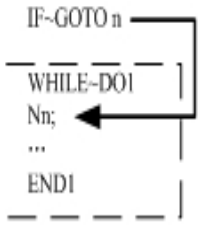
Wrong</td>
</tr>
</table>

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

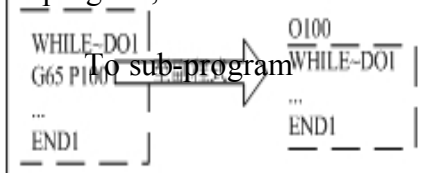
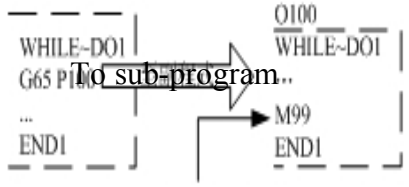
Wrong</td>
<td>(6) WHILE~DO m should be corresponding one by one in the same program
 

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.

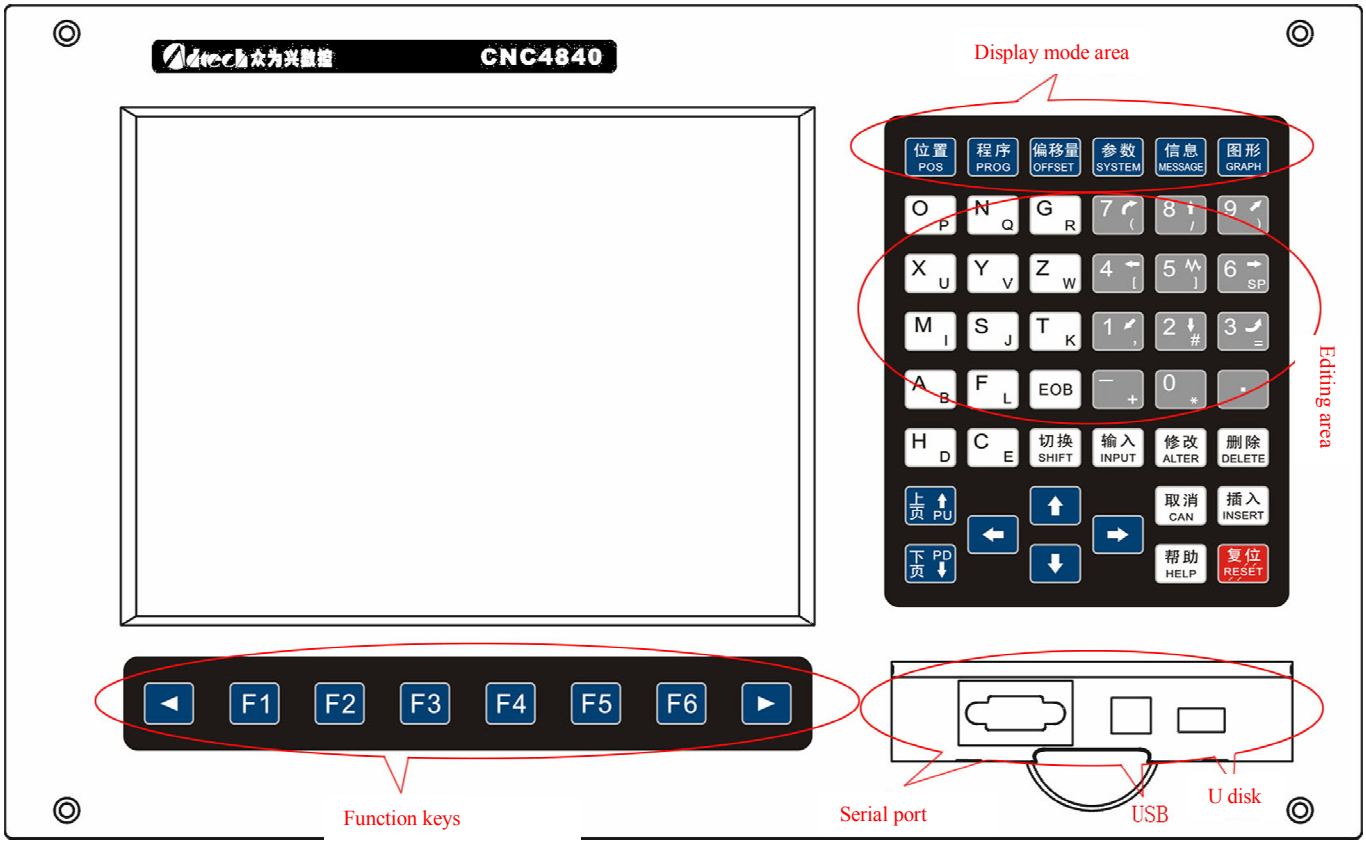
1) Variable Initialization; the variable used in program should be initialized at the beginning of program for transferring the variable value. One intervening variable is also needed for the transfer. this can prevent the program to modify the parameter environment when processing for many times.
2) 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

1. 1. 1 Description of Control Panel

2. 1. 1 LCD panel

See the following picture for the LCD panel of CNC4840.

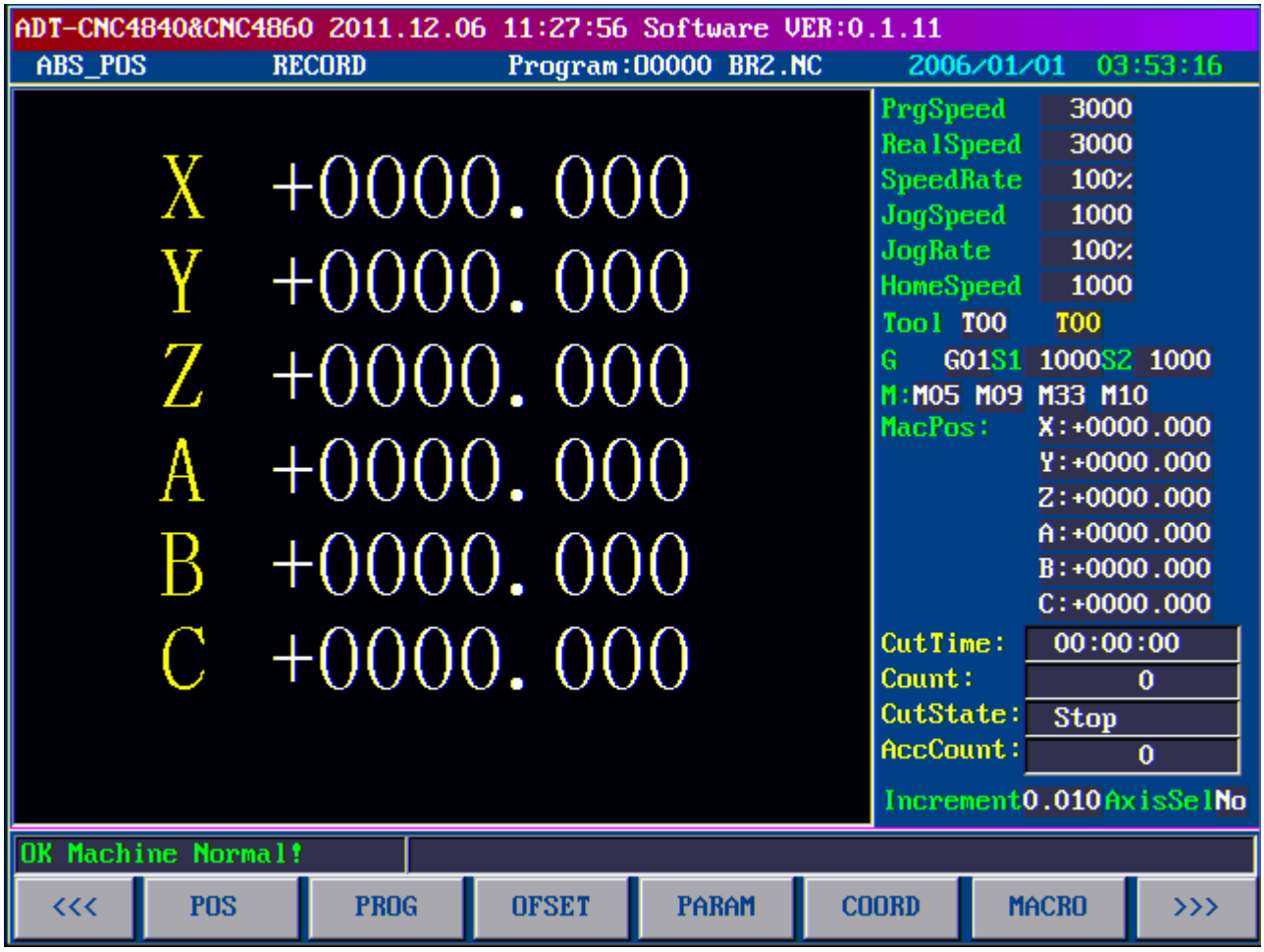


Description of display mode

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.

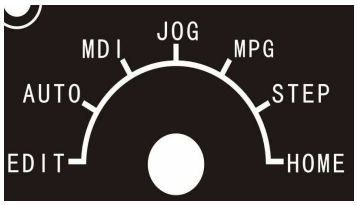
In the middle of the screen, the left is the content of all display interfaces, and the right displays the status of current M code, G code, S code and T code, as well as the processing information.

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 side to exit the sub-menu and return to the main menu. The first page includes [Edit Mode], [Auto Mode], [Manual Mode], [Clear Mode], [Single-step Mode], [Record Mode] (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 position display

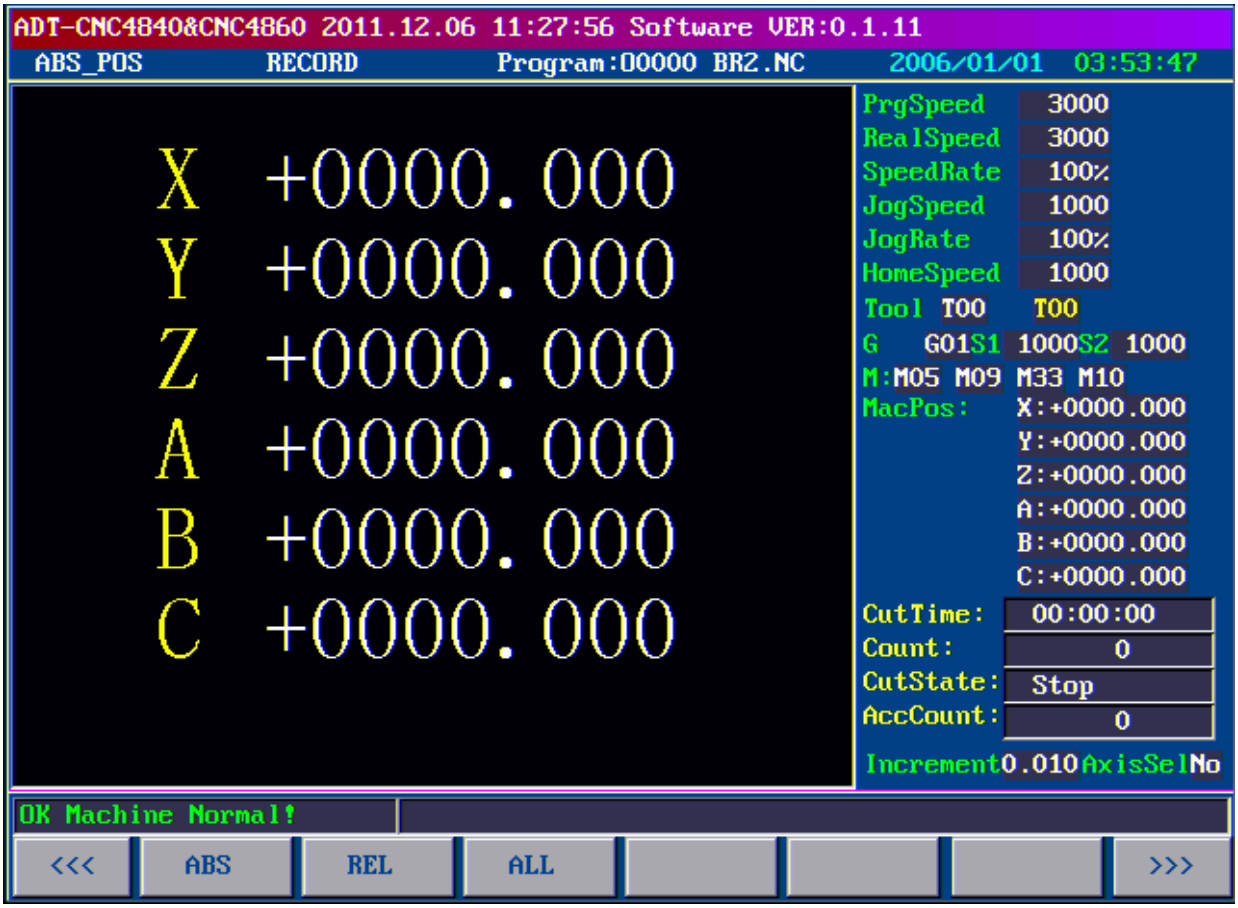
There are three pages for the position interface, you can press F1 F2 F3 to switch among absolute position, relative position, and comprehensive position.

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.

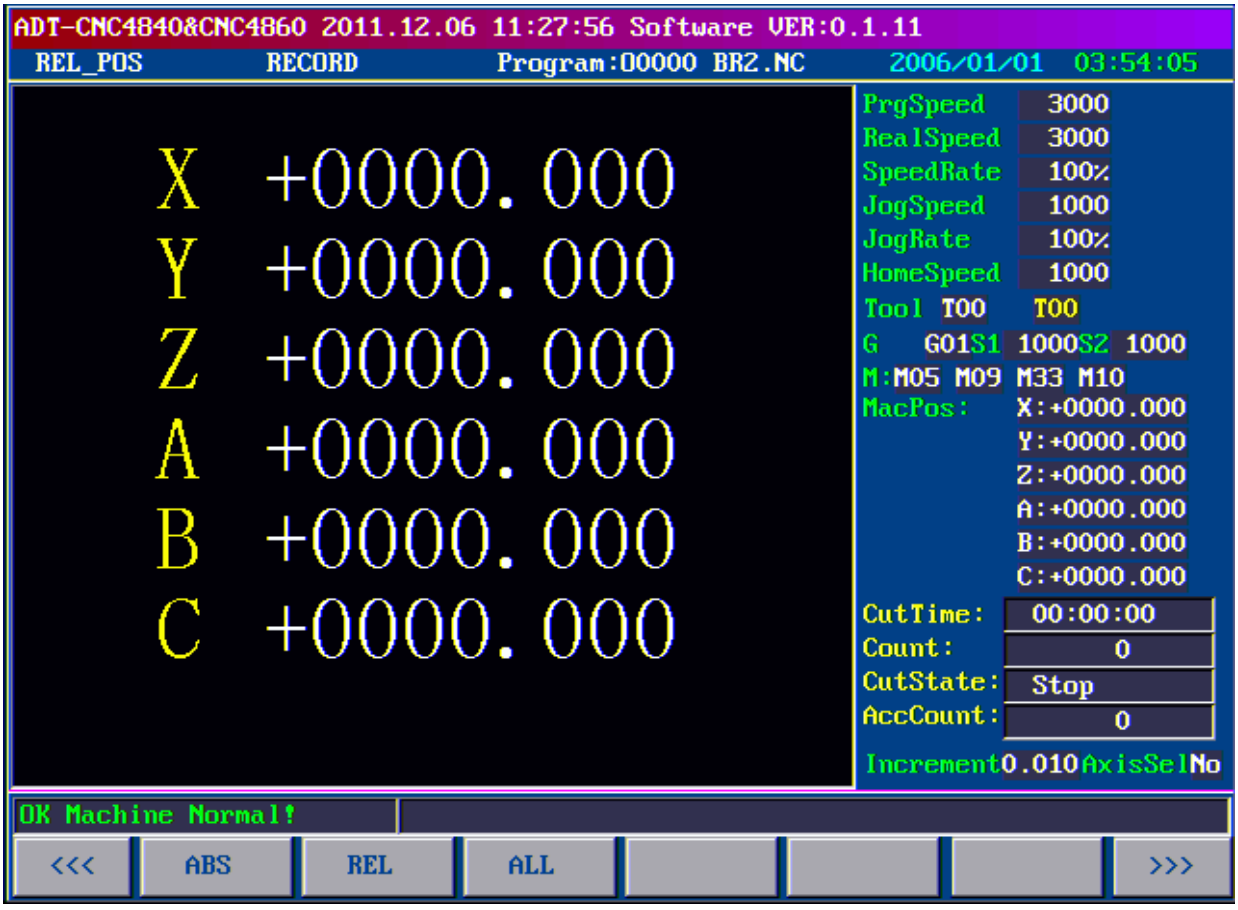
Relative position: display the displacement of address X, Y, Z, A ,B, and C corresponding to work piece coordinate system. It is can cleared at any time in manual mode, and it is for observing the position or for setting the cutter compensation value of counting way.

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

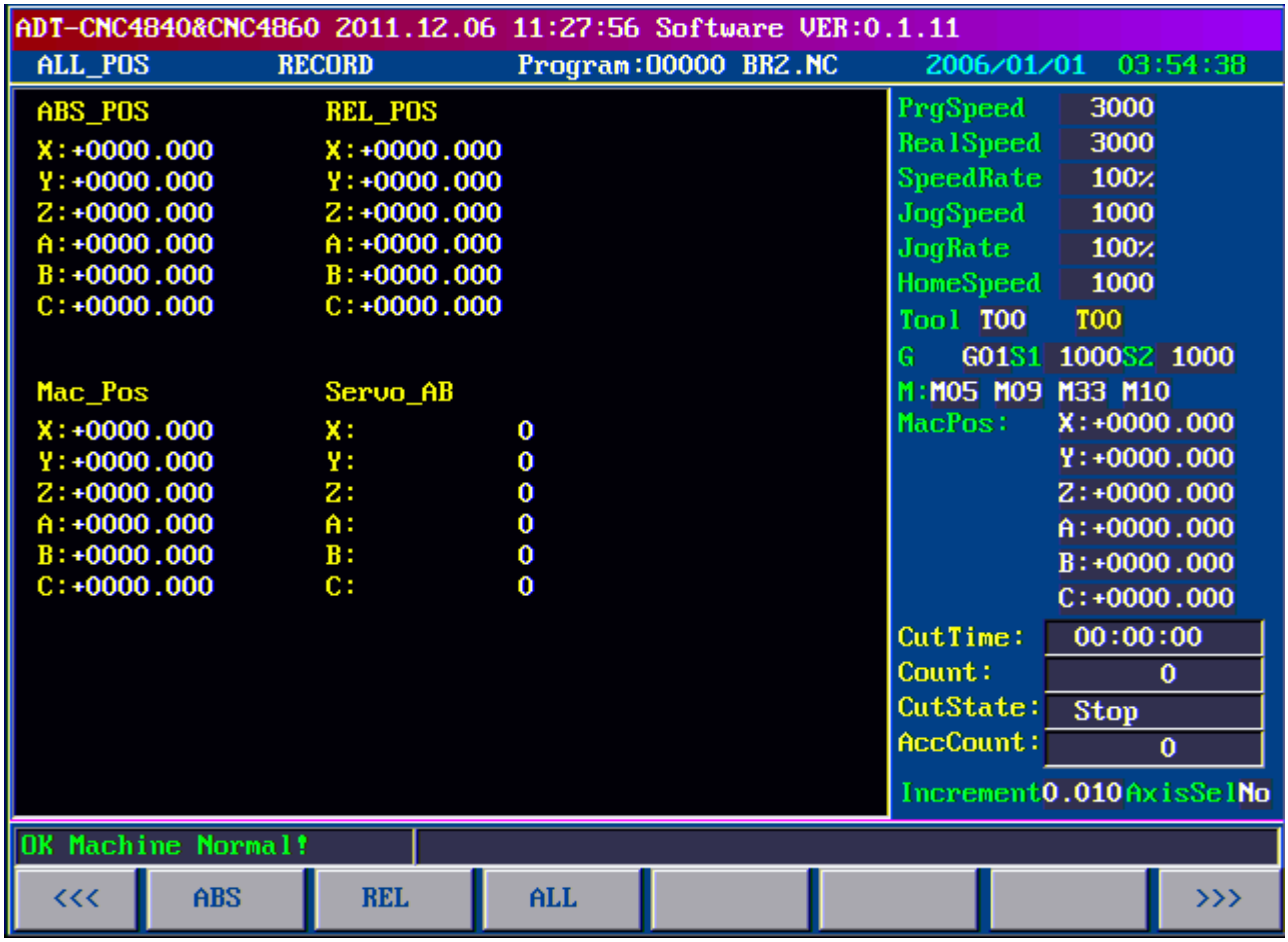


2. Interface of relative position



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.

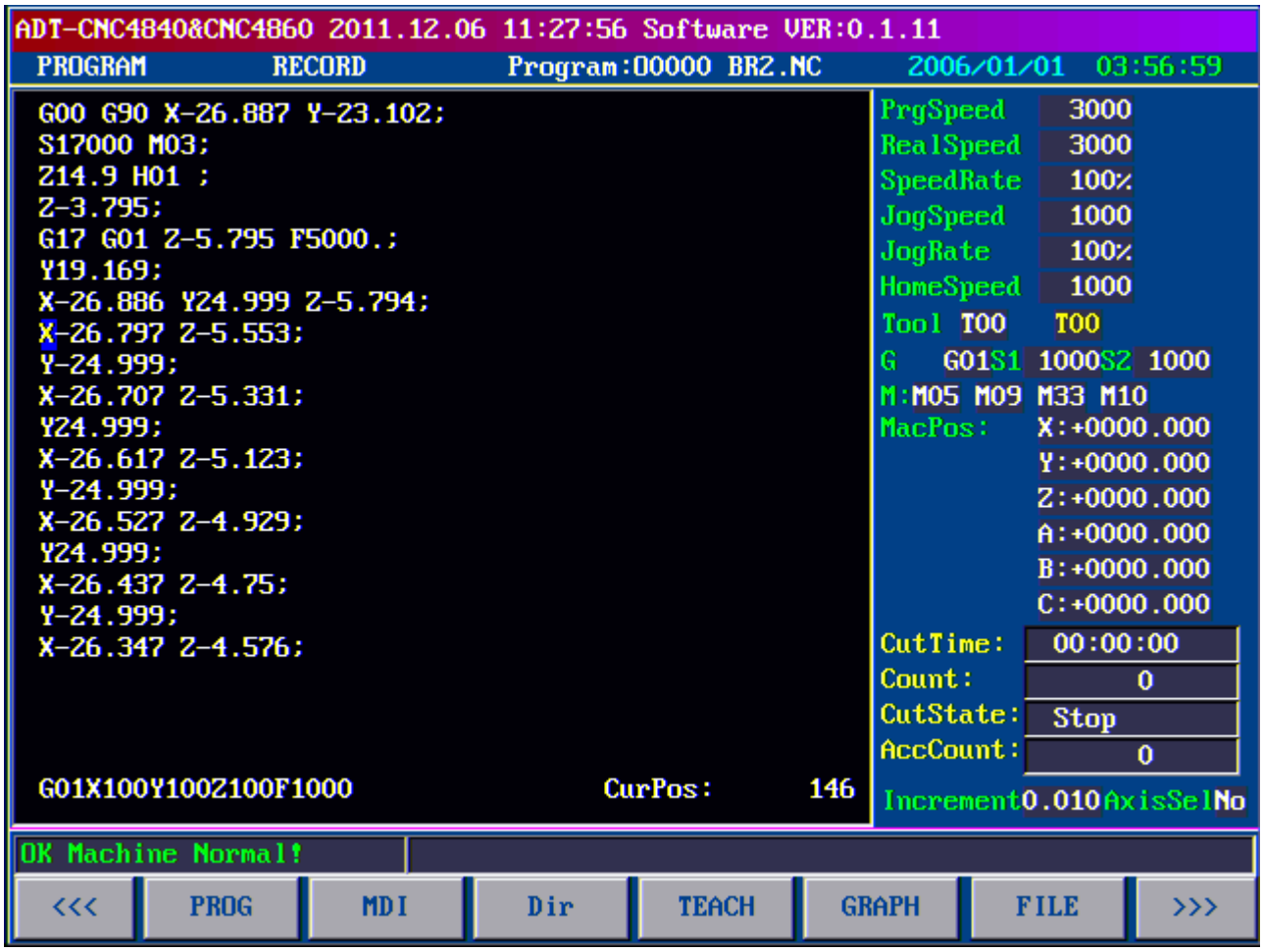
3． Interface of comprehensive position



1.2.2 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.

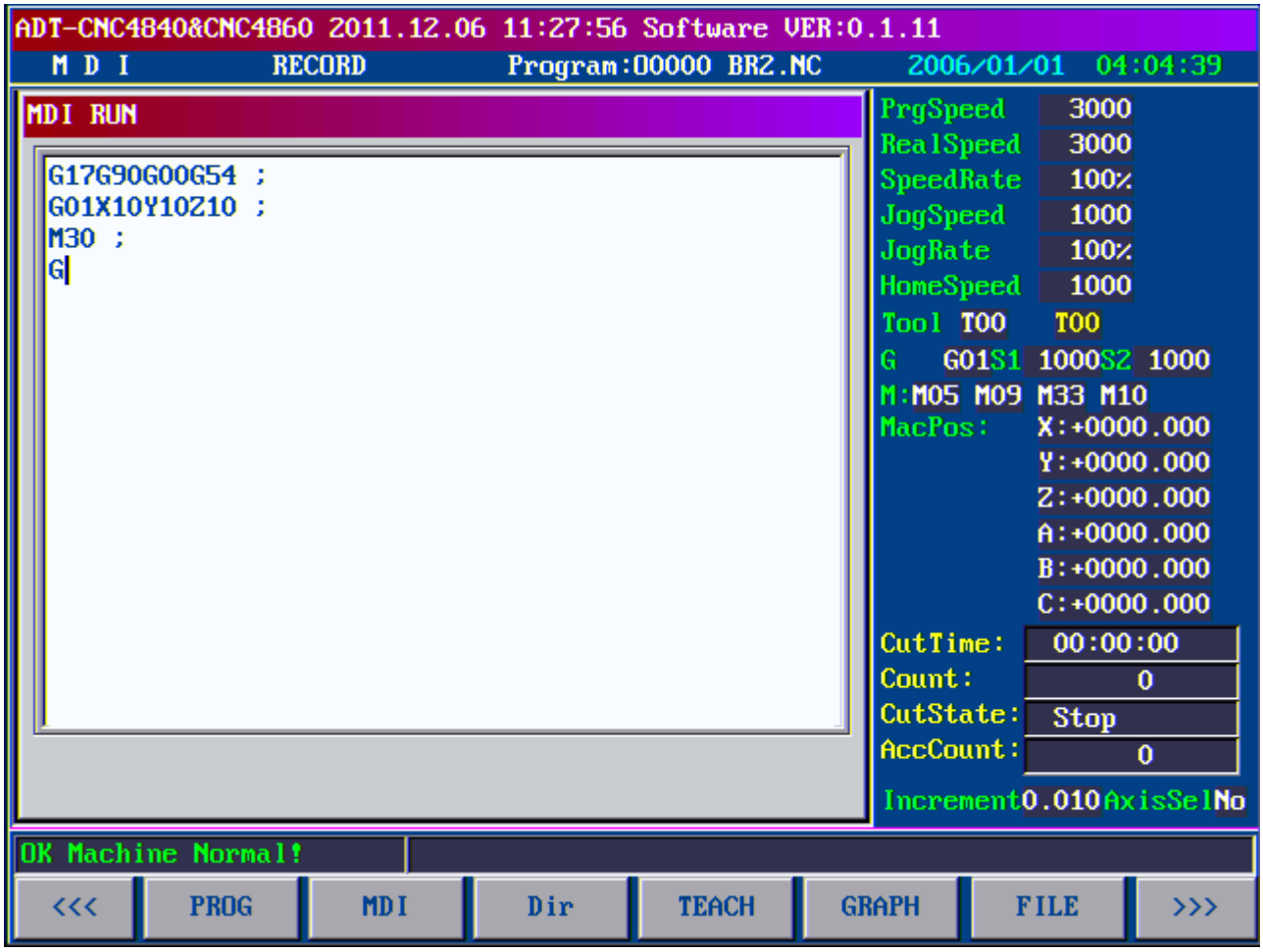


2. MDI

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. Then You can continue performing the MDI operation.

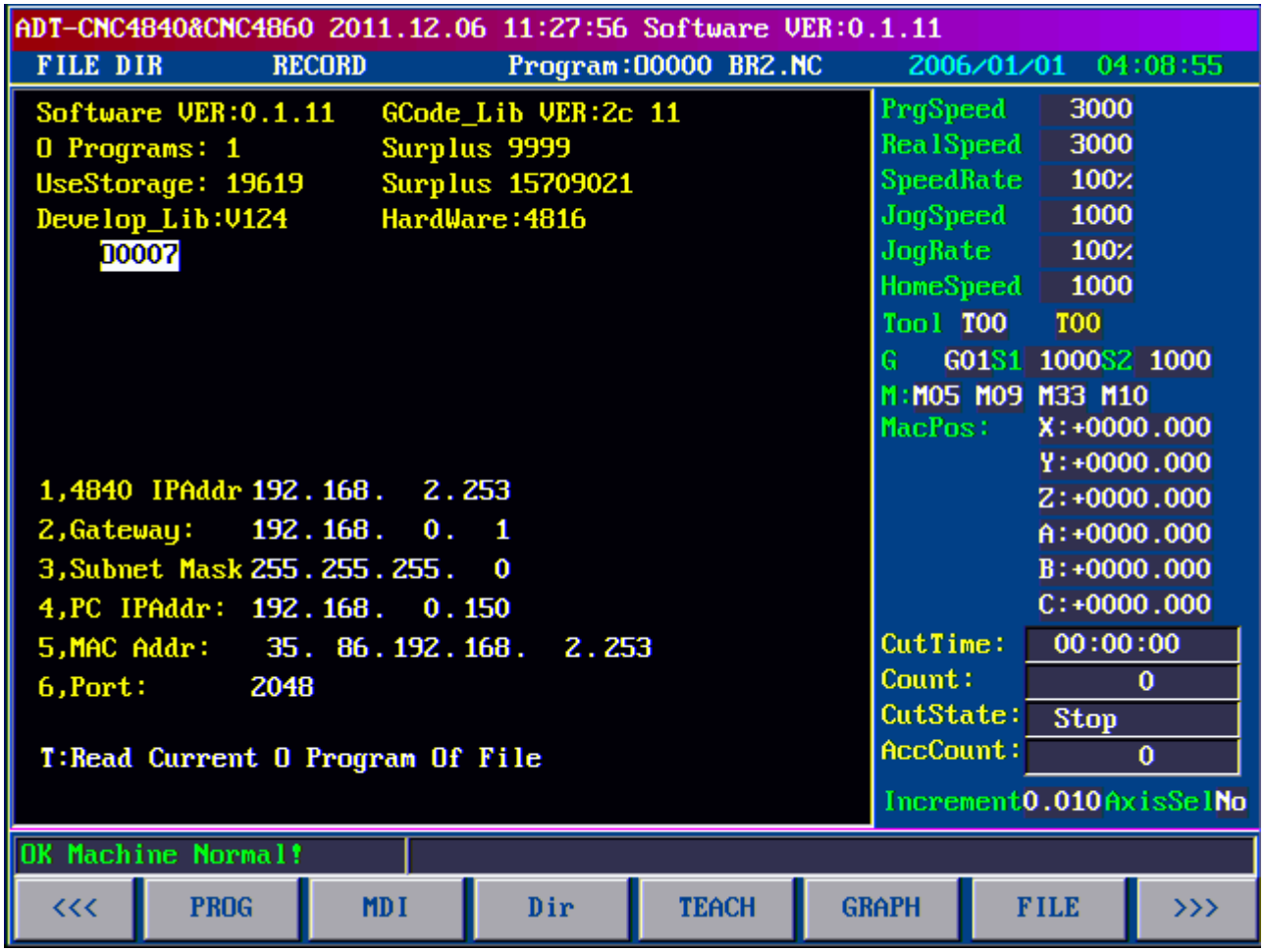


3. Program catalog

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,



4.

5. Graphic Simulation

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.

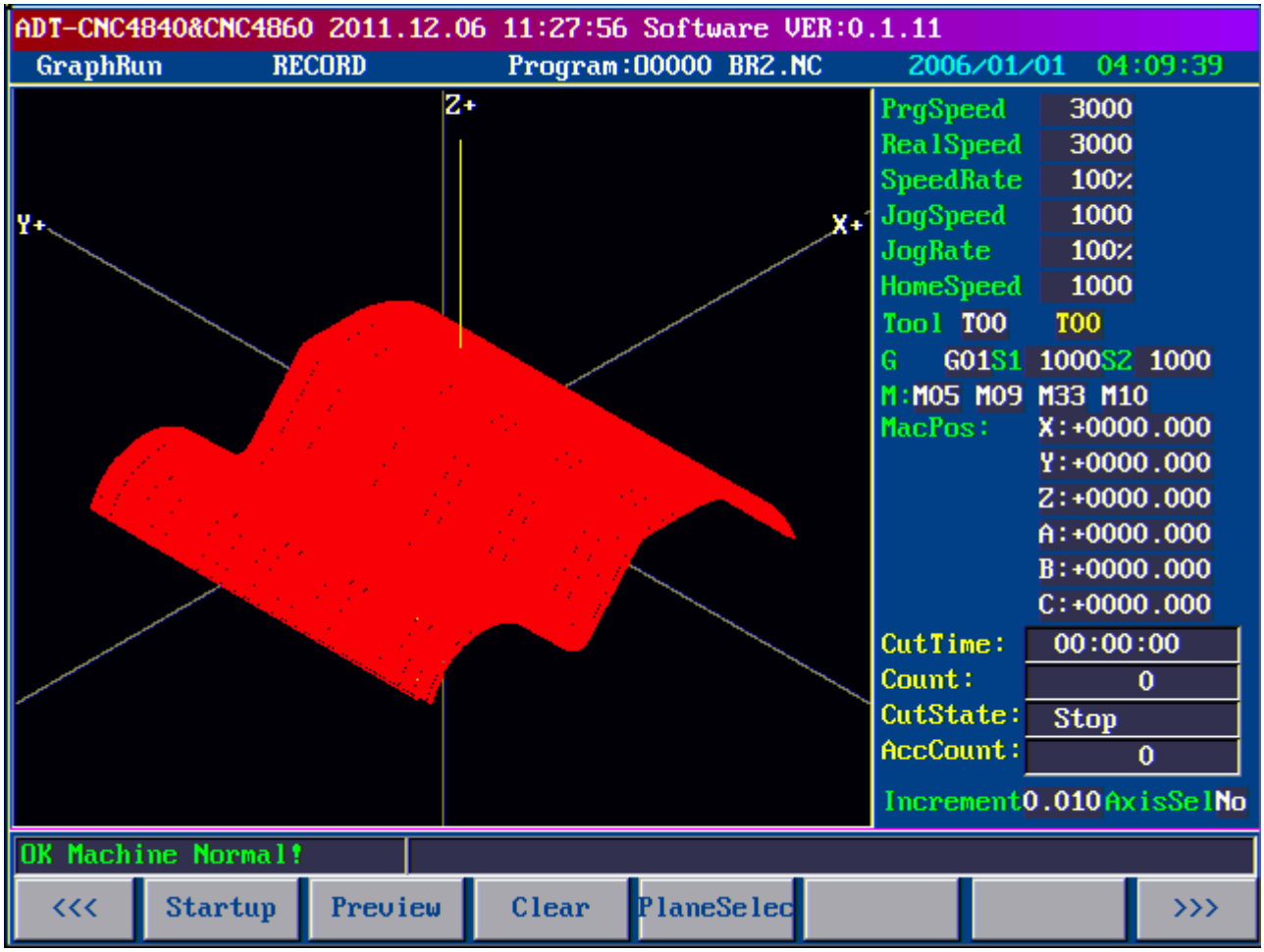
F1: start tracking, start track realtime tracking simulation when G code is processed.

F2: Preview. scan, draw, preview G code track graphics.

F3: Clear screen, clear the screen drawing area.

F4: Plane selection, set two-dimensional or three-dimensional Coordinate selection to draw simulation graphics.

【page up】button: simulation graphic zoom in
【page down】button: simulation graphic zoom out
【→】: simulation graphic move right
【←】: simulation graphic move left
【↑】: simulation graphic move up
【↓】: simulation graphic move down



6. File management

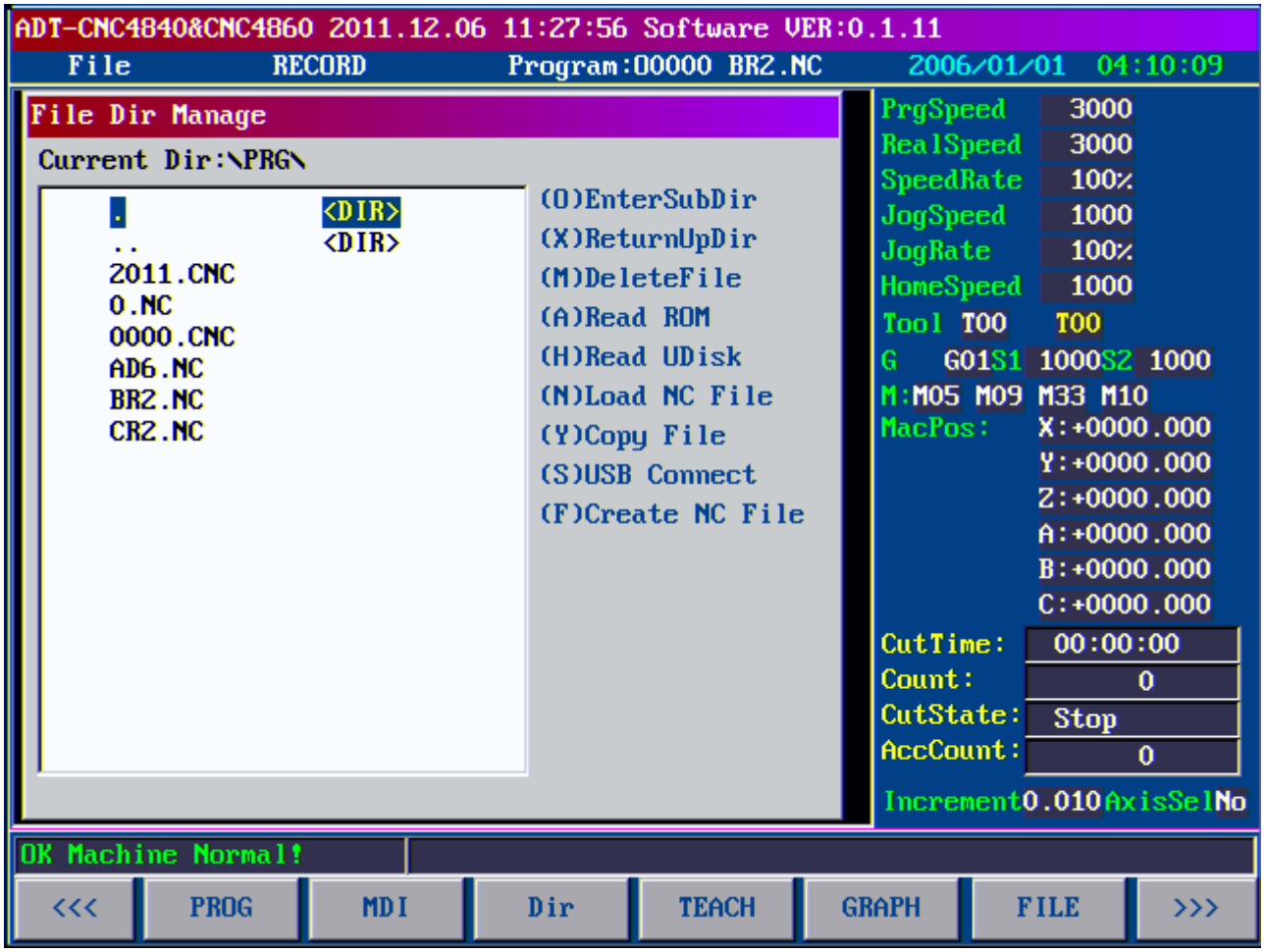
Press [File] F6 key to enter the file management interface as follows. The main function is to operate the external USB disk.

Operating interface of file management:

1. Catalog ADT is to store CNC4840 system program and system parameters, and the files under this catalog cannot be deleted.

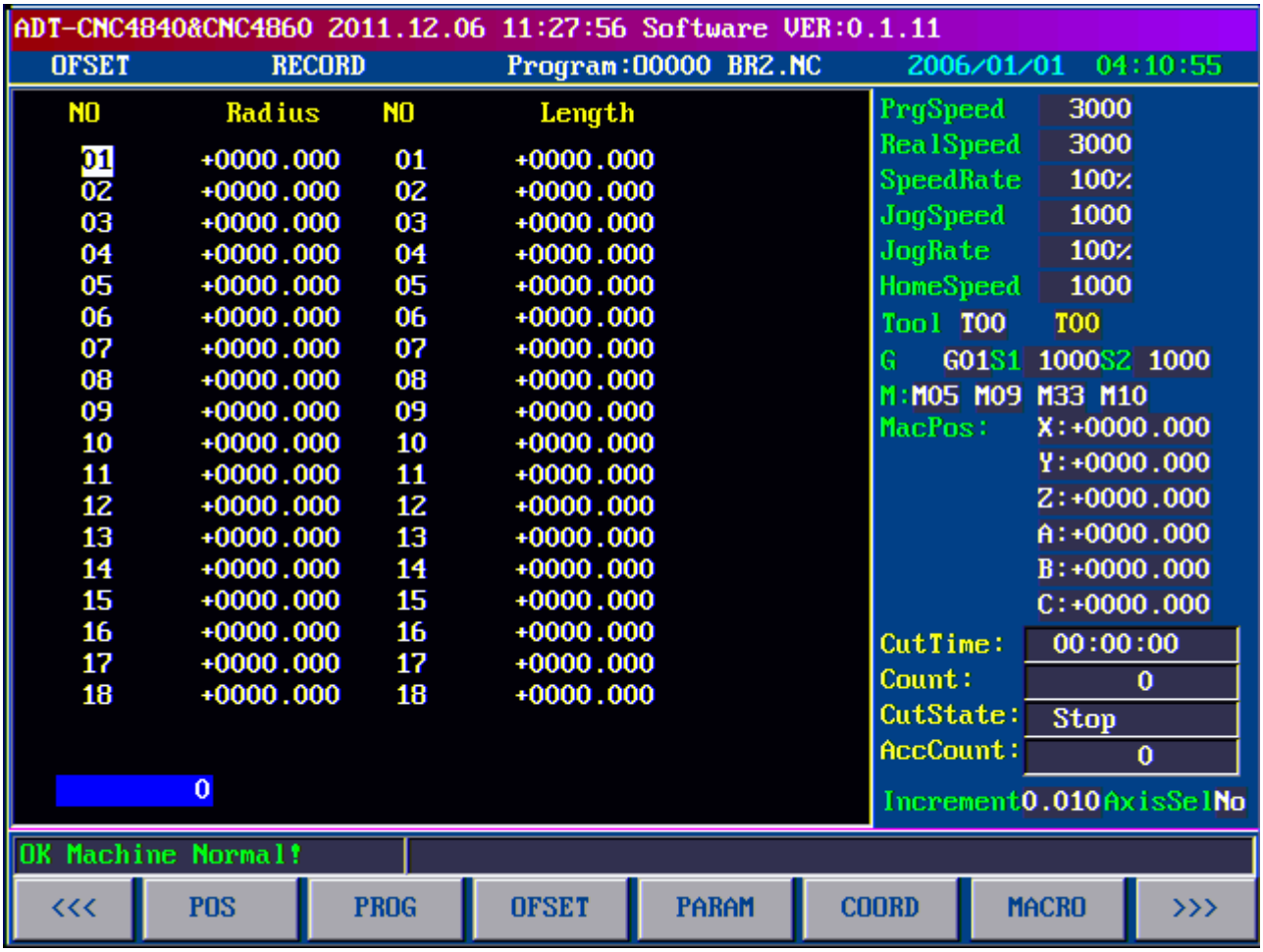
2. Catalog PRG is to store the processing files of system (Note: saved as xxxx.CNC).

3. Operation menu, press OXMAHNYST buttons according to prompts on the right.



1.2.3 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.



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

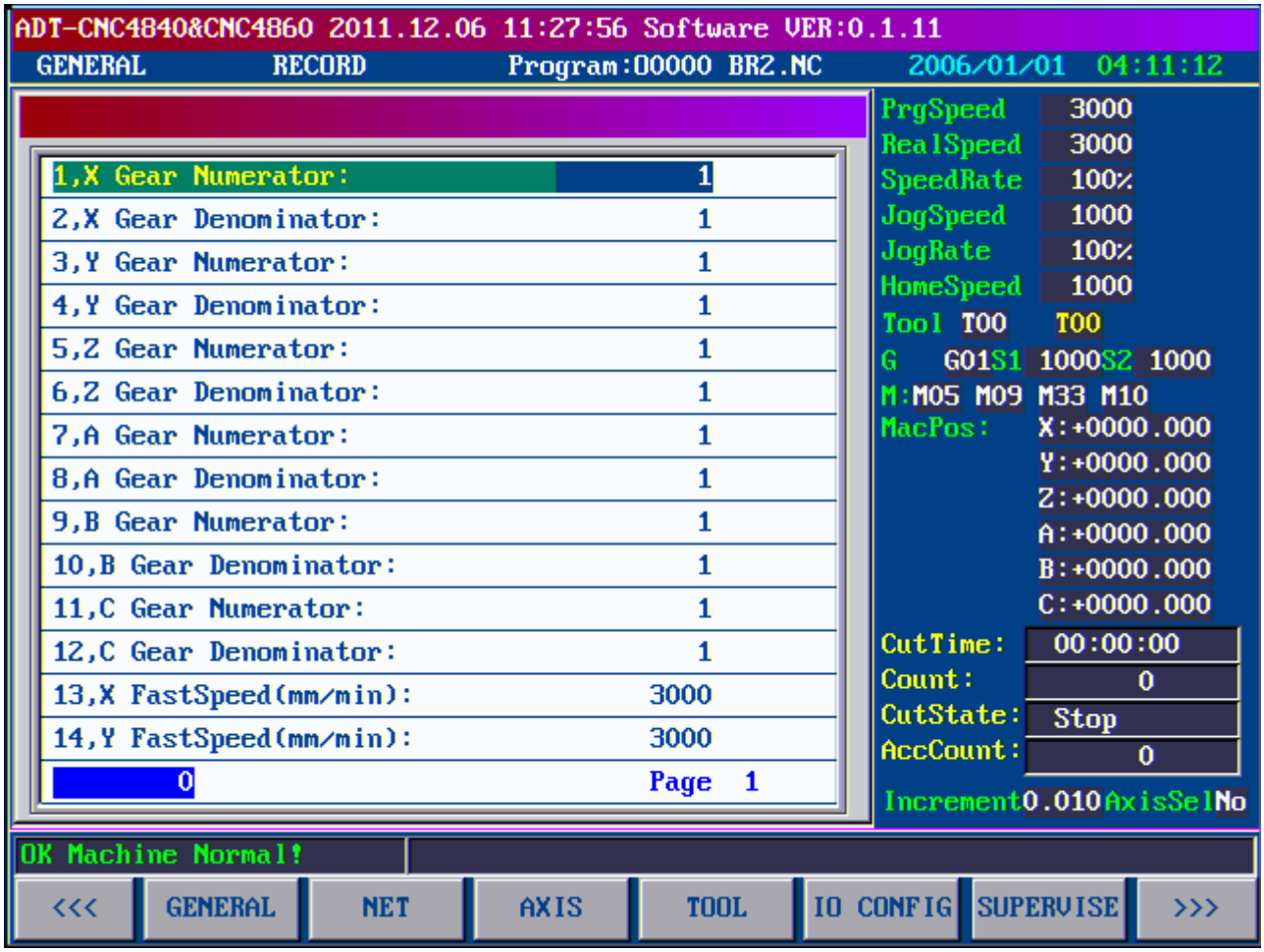
You can press 参数 SYSTEM or F4 key to enter the system parameter interface. To ensure the parameters would not be modified wrongly, the parameters can only be modified in [Record] or [Edit] mode.

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

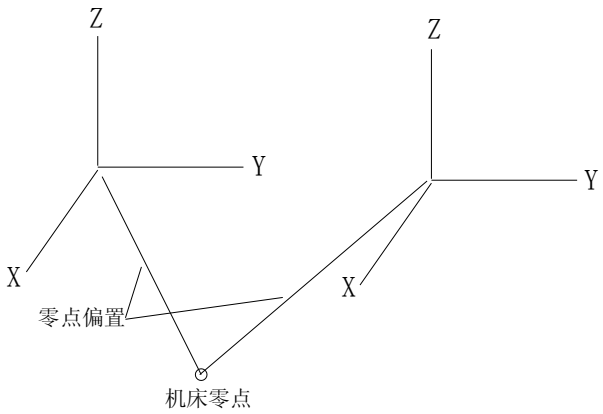
G56.....workpiece coordinate system 3

G57.....workpiece coordinate system 4

G58.....workpiece coordinate system 5

G59.....workpiece coordinate system 6

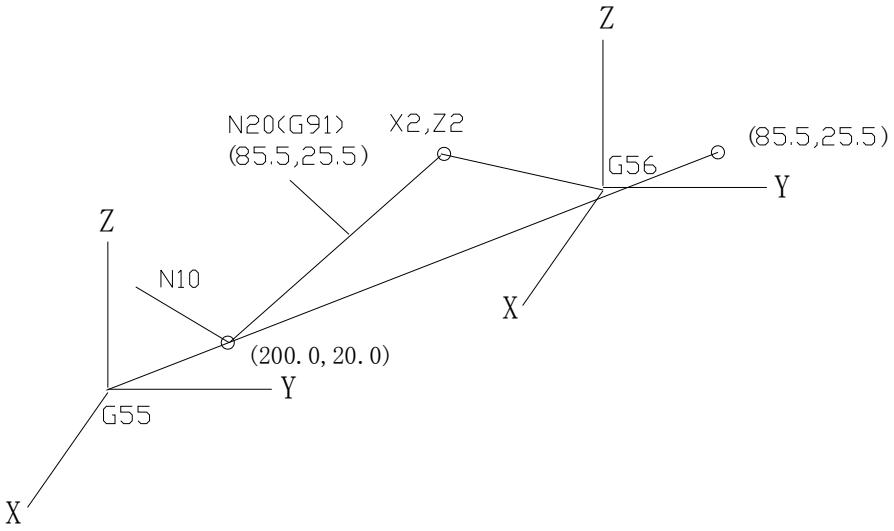
These 6 workpiece coordinate systems are set by distance (Part zero offset) from machine zero to the zero point of each coordinate system .



例：N10G55G90G00X100.0Z20.0

N20G56X80.5Z25.5

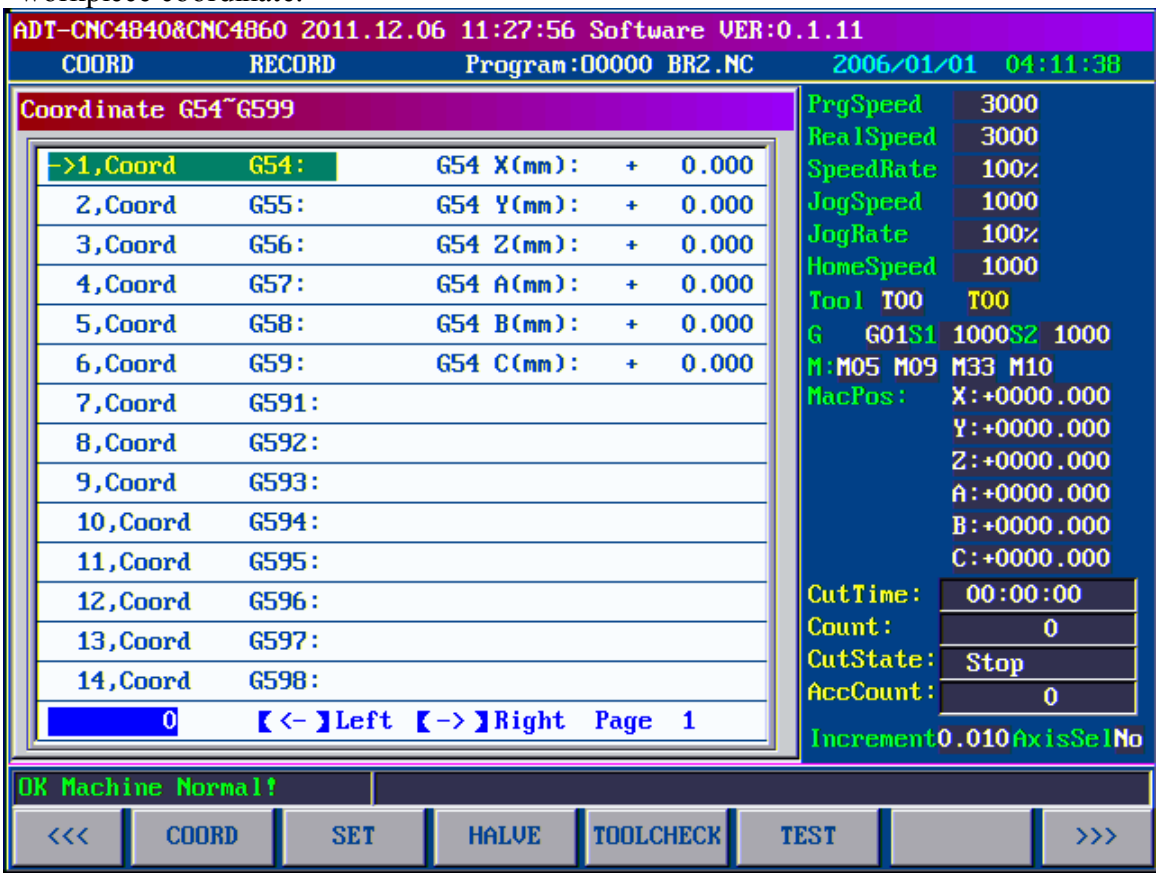
In the above example, the position in workpiece coordinate system 2 (X=100.0, Z=20.0) is quickly located to position in workpiece coordinate system 3 (X=80.5, Z=25.5). If N20 program segment is G91, it's increment movement. When starting to execute the N20 program segment, the absolute coordinate value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

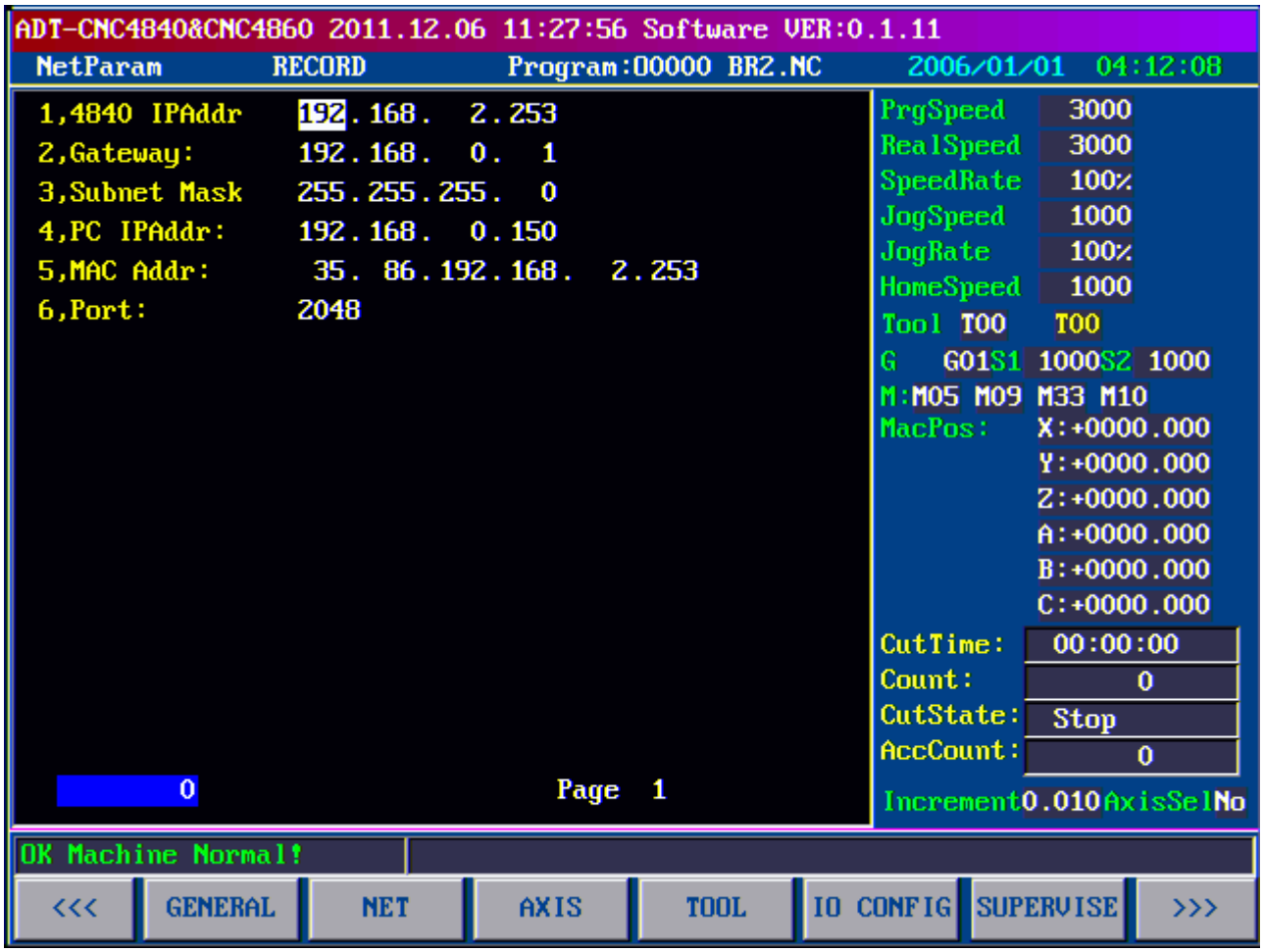
2. Display and input of workpiece zero

In recording or editing mode, you can press  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.



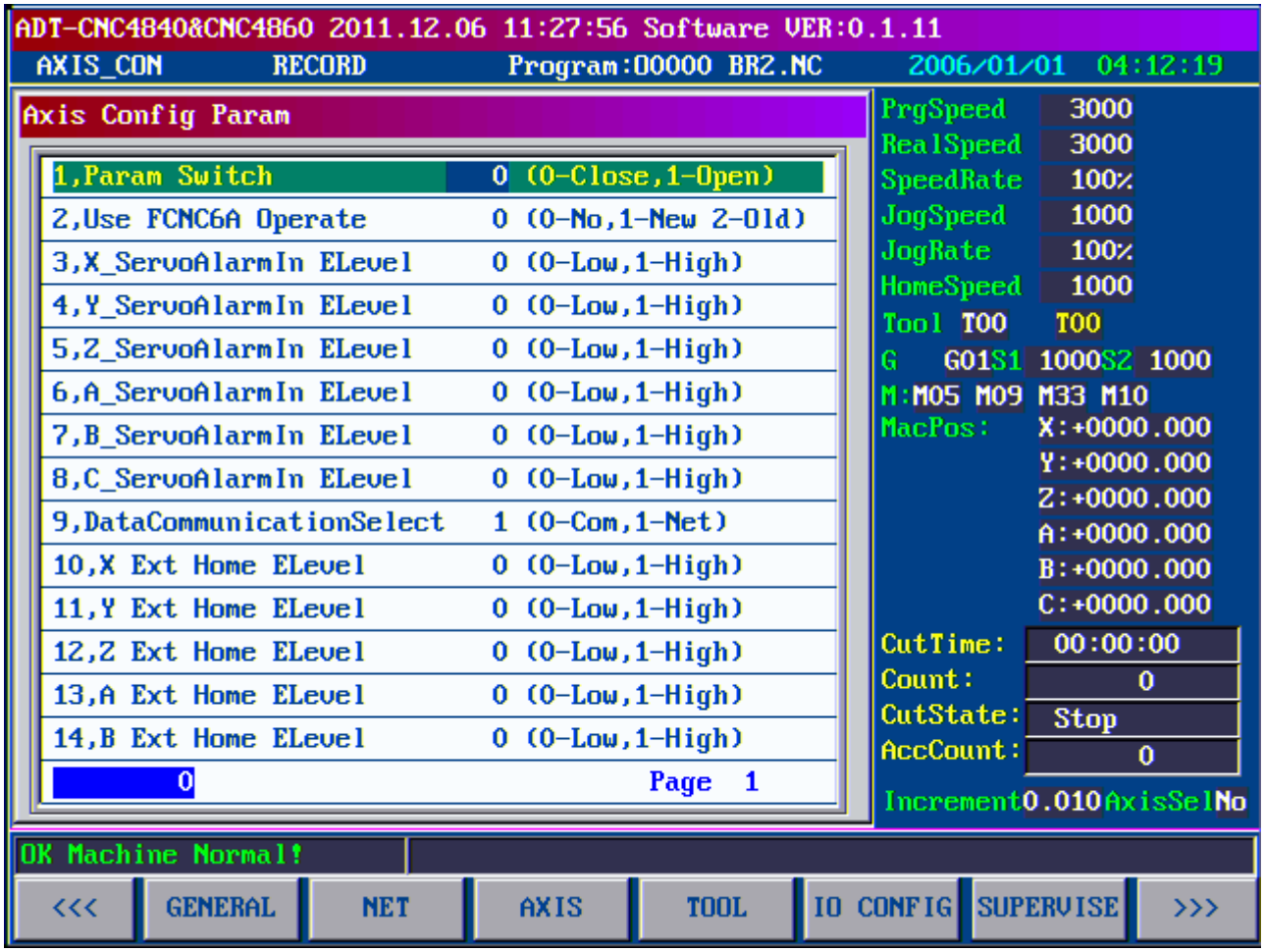
3. Settings of network parameters

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 CNC4840, 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)



4. axis parameter setting

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)



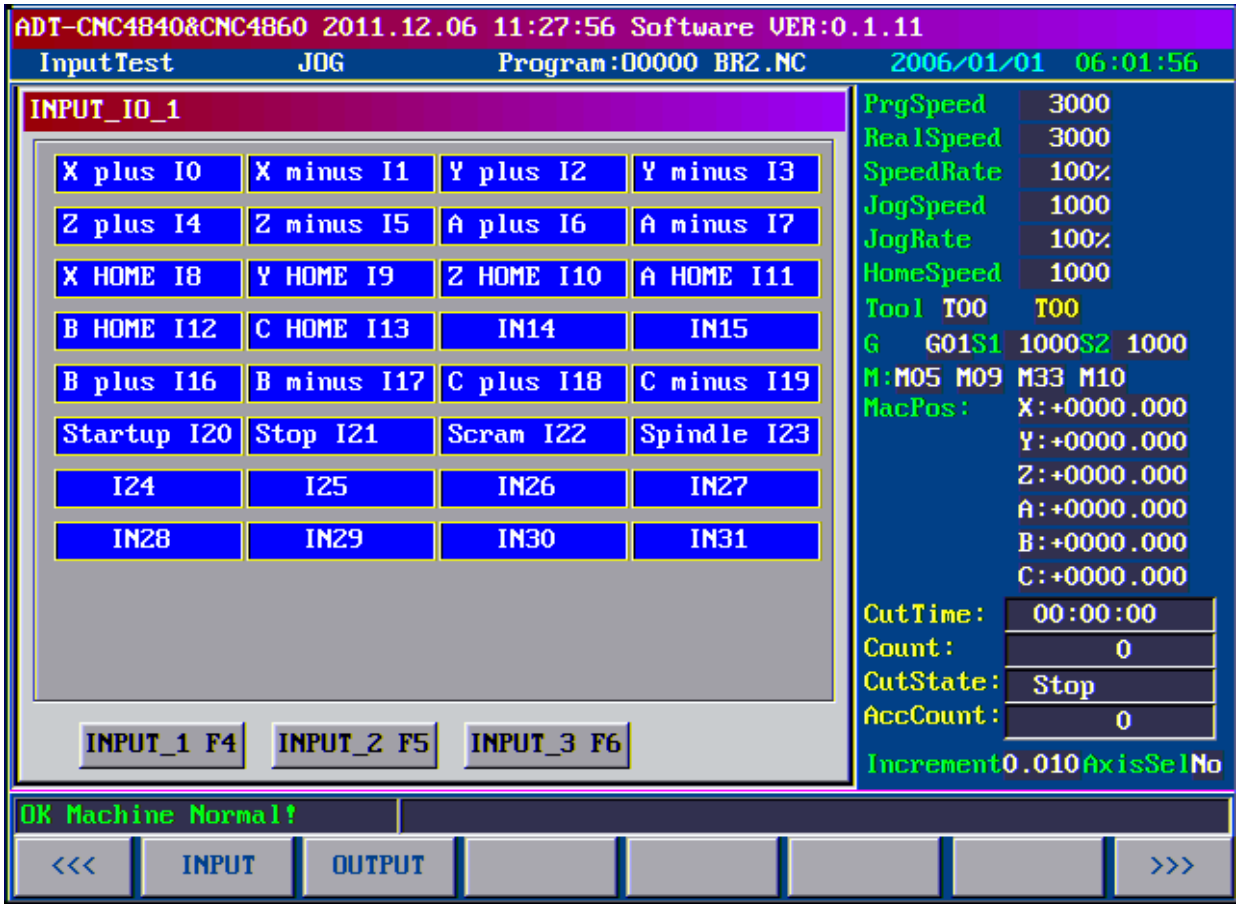
1.2.5 Diagnosis display setting

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.

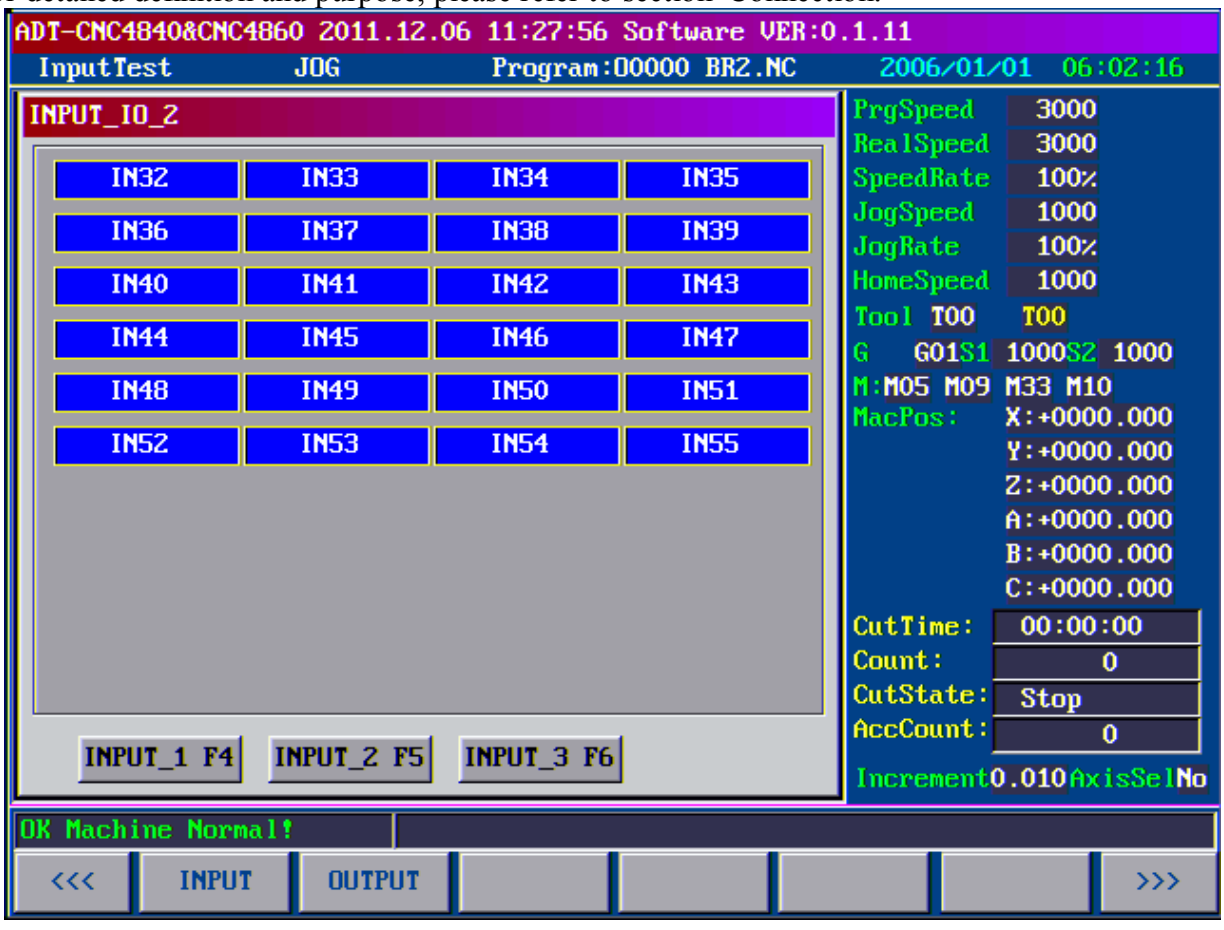
The input diagnosis includes basic input, extended input and other inputs.

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.

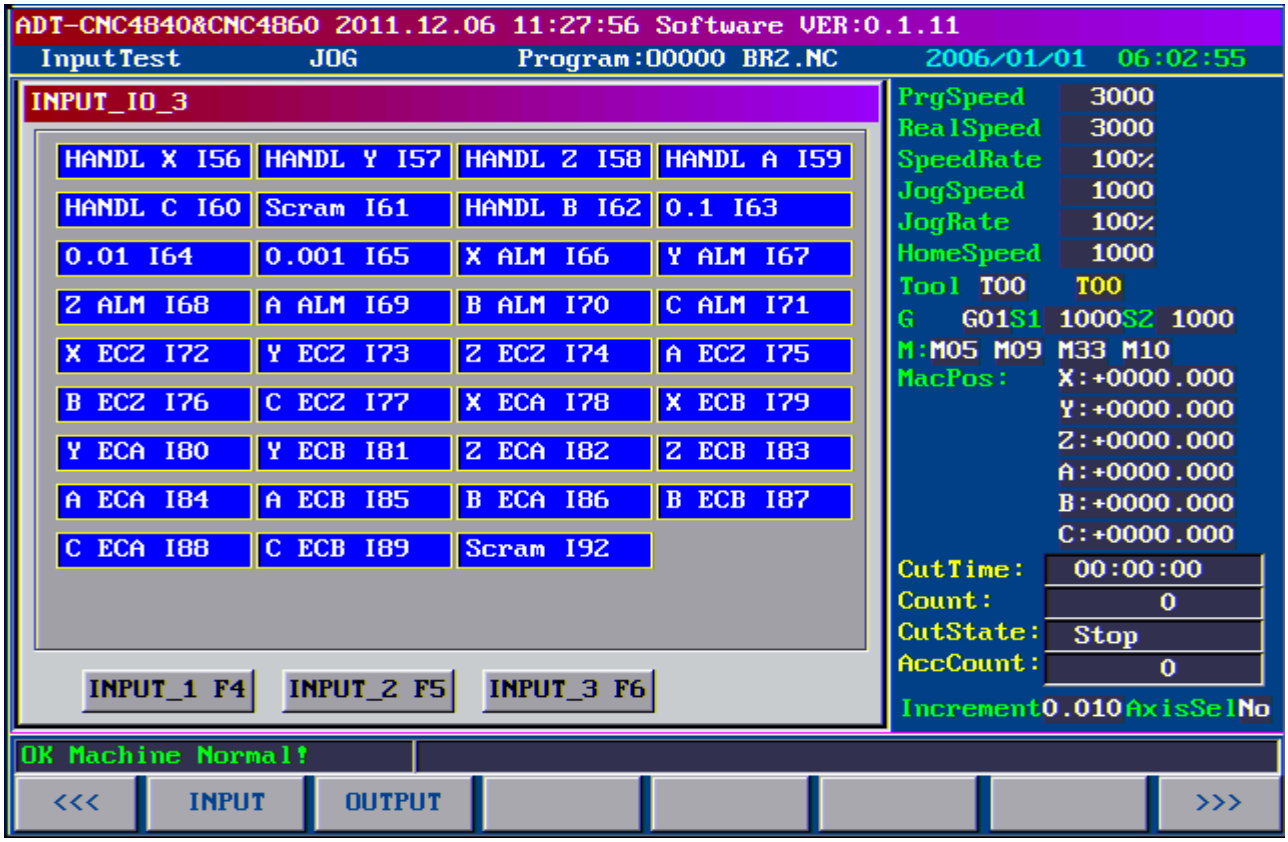
The basic input of machine is corresponding to 37-pin interface of XS1 machine input. For detailed definition and purpose, please refer to section–Connection.



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.



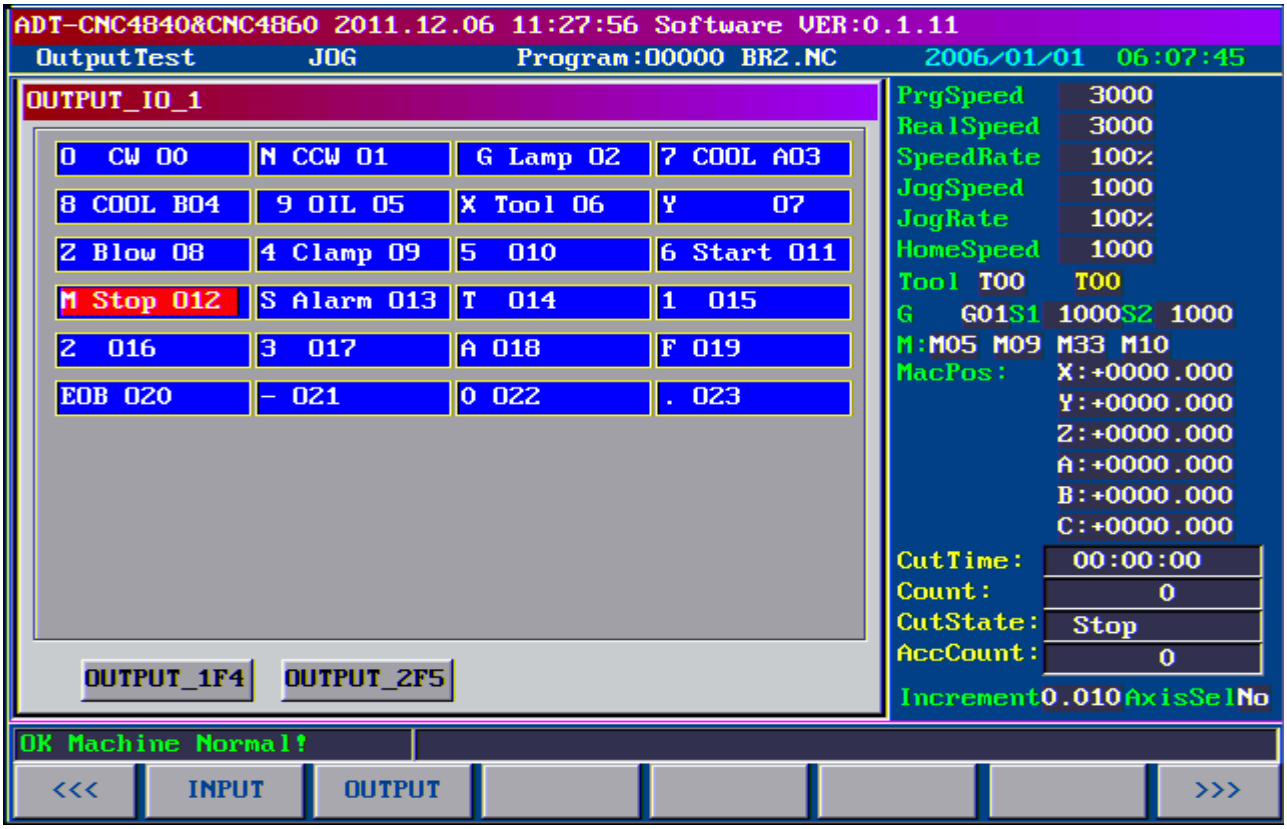
Other inputs are corresponding to XS6 handheld box, XS7 X axis, XS8Y axis, XS9Z axis, XS10A axis, XS11B axis, and XS12C axis pulse interfaces. For detailed definition and purpose, please refer to section–Connection.



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.



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.

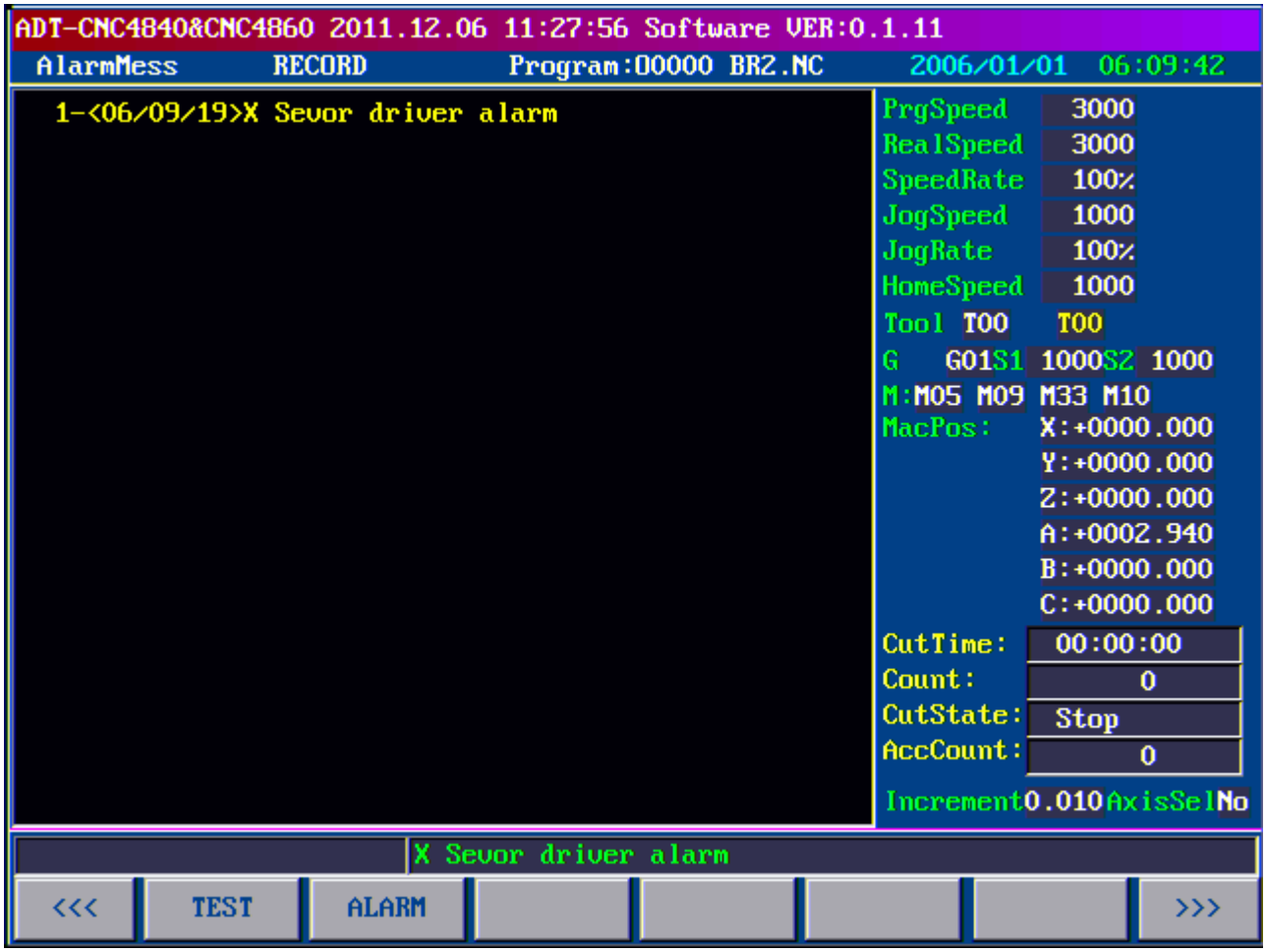
The extended and pulse outputs are corresponding to 25-pin interface of XS3 machine extended output, as well as XS7 X axis, XS8Y axis, XS9Z axis, XS10A axis, XS11B axis, and XS12C axis pulse interfaces. For detailed definition and purpose, please refer to section–Connection.

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).

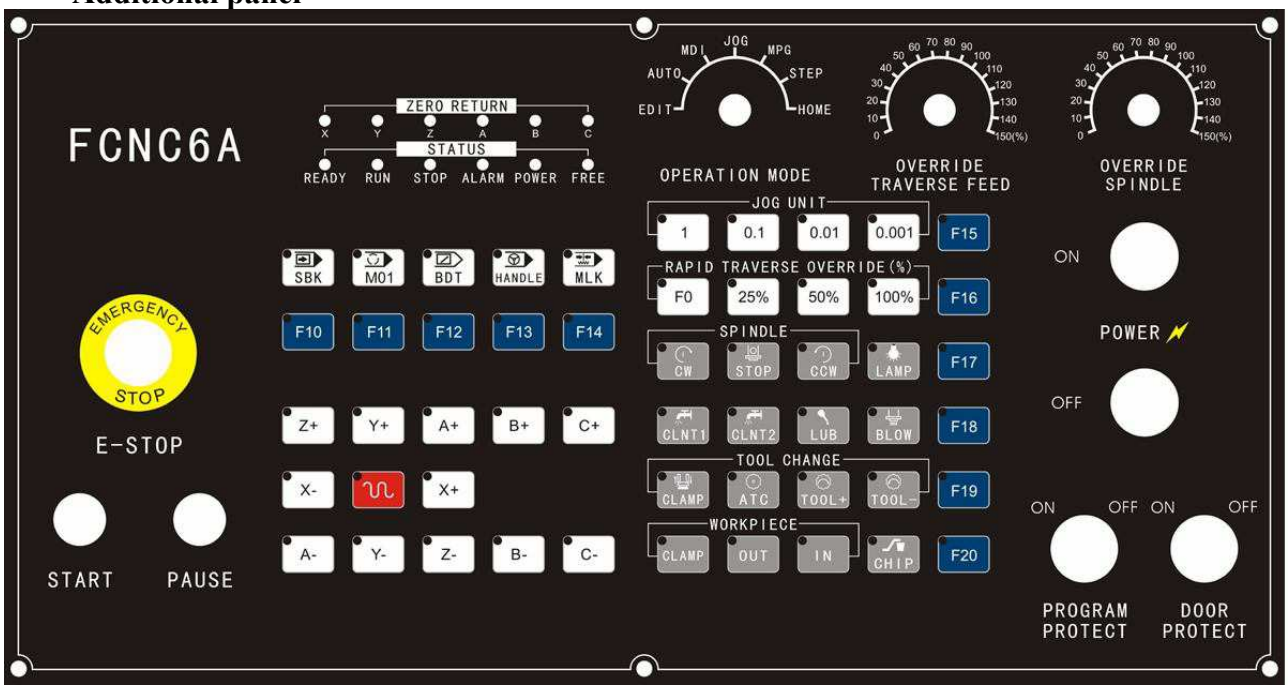
1.2.6 Alarm display



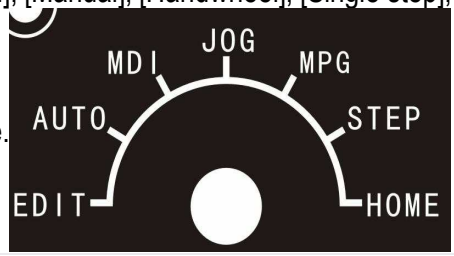
1. 3 Description of operating mode

1.2.1 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.

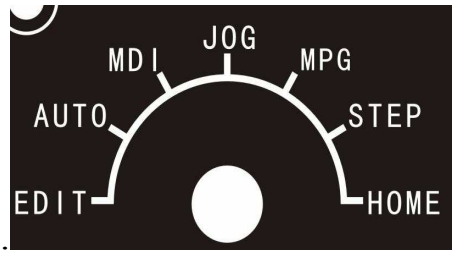


S/N	Name	Purpose
1	[Edit]	edition of program, and the setting of system parameters, coordinate parameters, offset parameters, network parameters, setting parameters and administrative password
2	[Auto]	start/stop the processing code program, adjust the main axis speed and feed rate
3	[Manual]	manual 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

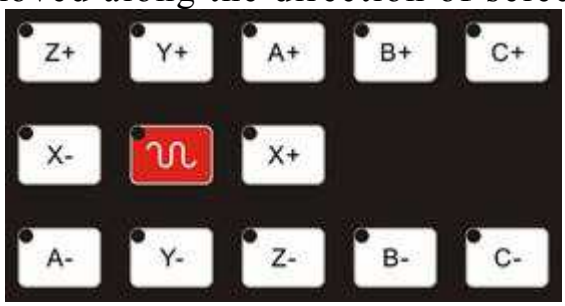
1.4 Manual operations

Continuous manual feed operations

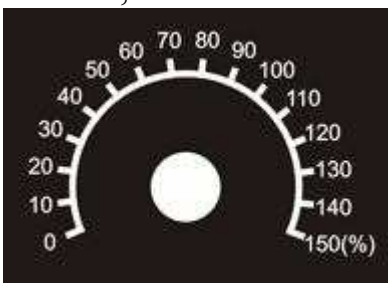
Rotate the mode shift to manual mode to select the manual operation mode.



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%.



1.4.2 reset relative position

Reset relative position: press X U, Y V, Z W or A B key on CNC4840 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.

1.4.3 Operations of manual auxiliary functions

1 换刀

In [Manual] mode, press 

key and the tool rest will be rotated for changing for the next tool.

2 Main axis positive rotation

In [Manual] mode, press 

key and the main axis will be rotated in positive direction.

3 main axis stop

In [Manual] mode, press 

key and the main axis will be stopped.

4 main axis reverse rotation

In [Manual] or [Single-step/Handwheel] mode, press 

key and the main axis will be rotated in reverse direction.

5 Cooling 1, Cooling 2

In [Manual] mode, press any one of 

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 

key and the corresponding lubrication will be on while press it again the lubrication will be off.

Others, such as 

illumination、 

blowing、 

adjustor、 

tool magazine +、 

tool magazine -、 

adjusting materials、 

dumping、 

feeding、 

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>.

1.5 Auto operation

1.5.1 start of program

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.

MDI operating

Select the recording mode, press [MDI] 

key to enter MDI operating interface, input the words to be executed and press 

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.

1.5.2 stop of auto operation

1.5.2 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

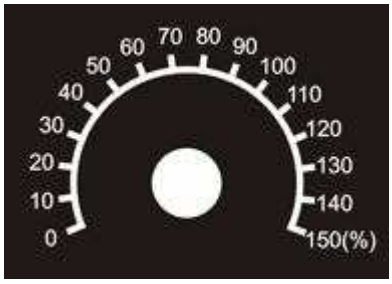
3)after executing the action of M, S, and T, stop press [Start] and the program will continue.

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.

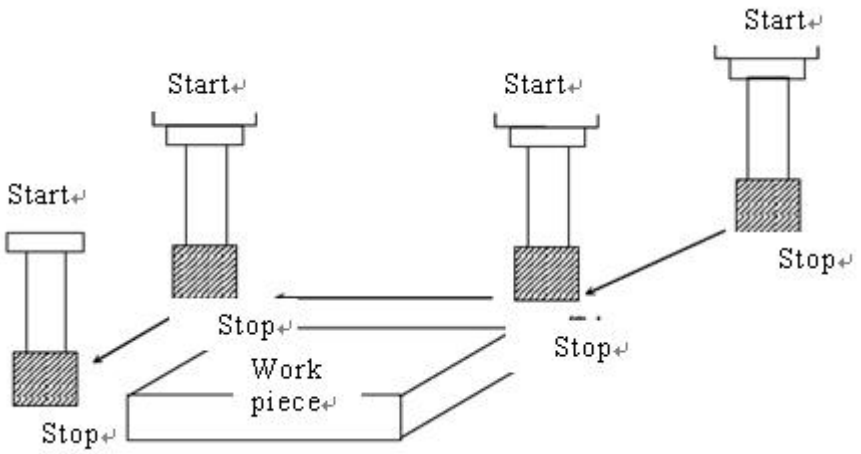
1.5.3 feed rate adjustment in auto operation

1.2.2 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.



1.5.4 Single program segment

In auto mode, press MLK 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.



1.5.5 Skip the program segment

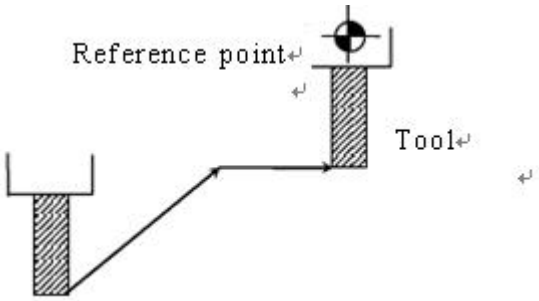
In auto mode, press 

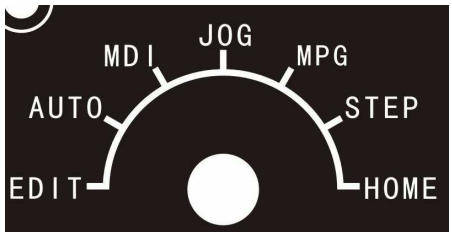
key and the indicator will be on. At this time, the system will skip the program segment automatically when it is executed to program segment with "/" and start executing the next one. This function is to make the program segment with "/" in program to be invalid.

1.6 Zero fill

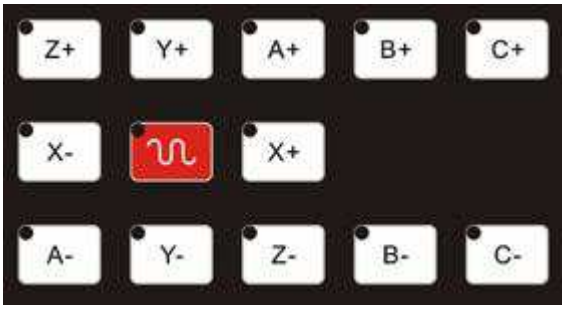
1.6.1 return to reference point manually

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".

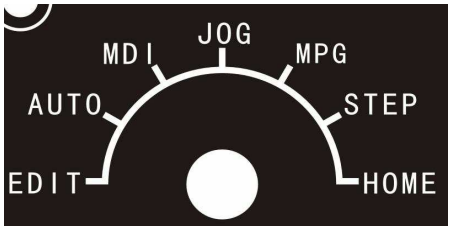
When the value is 0, it means there is no zero switch.

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.

1. 7 Single-step/Handwheel operation

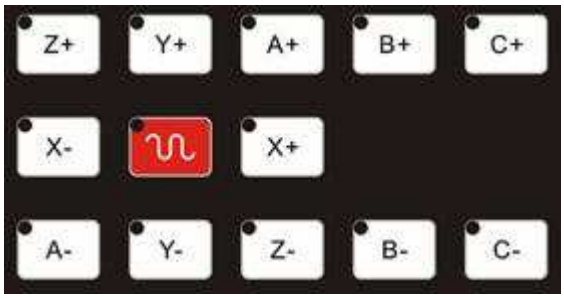
1. 7. 1 Single-step feed

Switch the mode shift 

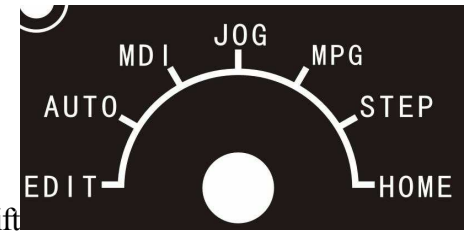
key to "Single-step" mode, and select [Single-step] operating mode. The operating mode column on LCD will show words "Single-step mode".

Select the moving distance 

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

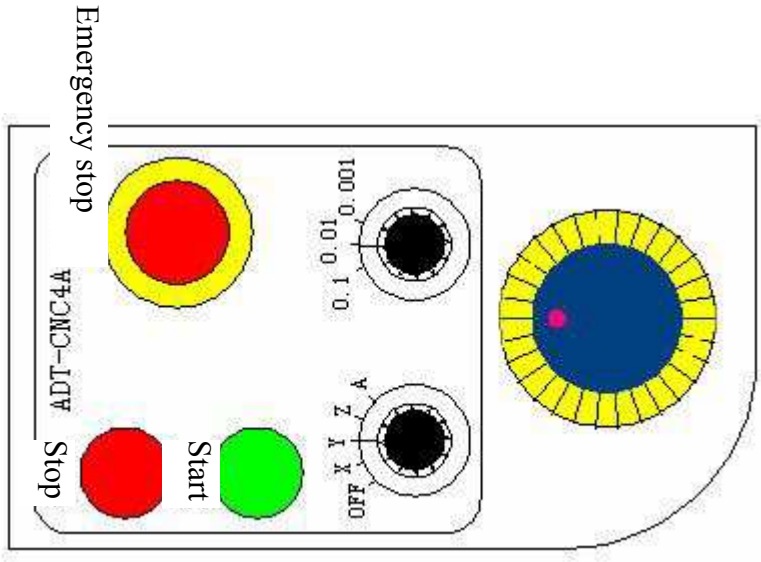


1. 7. 2 Handwheel feed

Switch the mode shift 

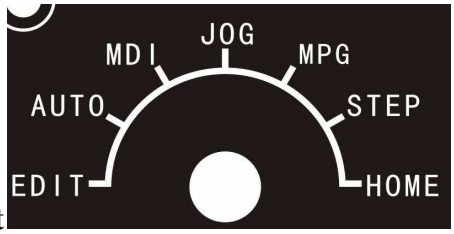
key to "Handwheel" mode and select the [Handwheel] operating mode. The operating mode column on LCD will show words "Handwheel mode".

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.



1.8 Edition operation

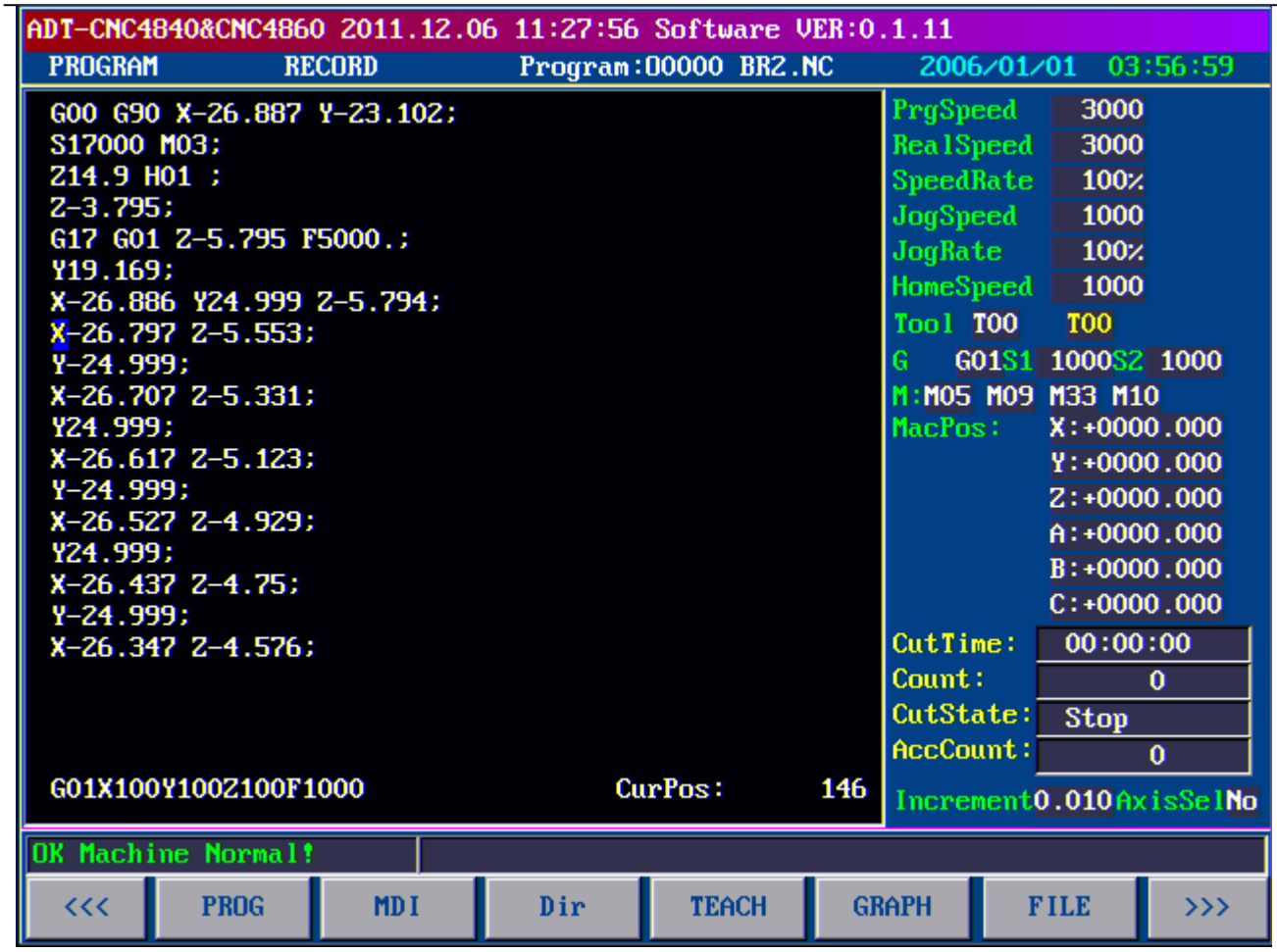
1.8.1 Preparation before program storage and edition

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

1.8.2 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.



1.8.3 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.

1.8.4 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.

1.8.5 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.

1.8.6 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.

1.8.7 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.

1.8.8 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: 72 groups, in which length compensation has 36 groups and radius compensation has 36 groups.

1.8.9 Download of program

1、Download via USB communication:

(1)Use USB cable to connect the computer and CNC4840 controller, rotate the mode shift selection controller to <Record> mode -→press 程序 PROG →press F6, to show 'File catalog' interface →press S J (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 twice, then press 位置 POS to go back to position interface.(Note: File name of OXXXX should not be repeated; otherwise, only 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

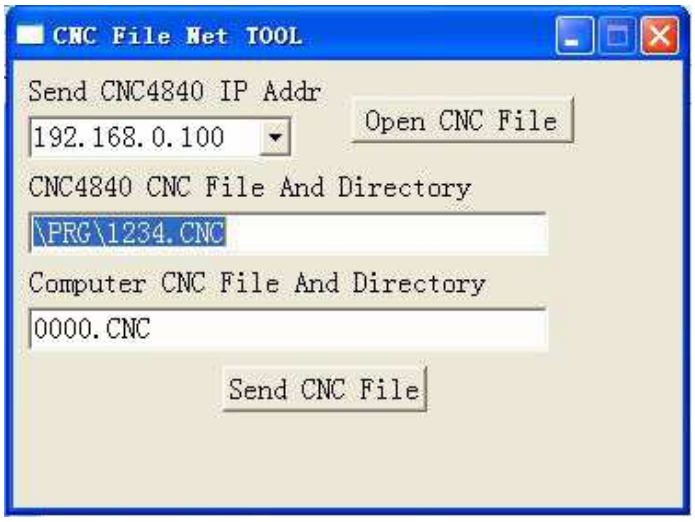
M30

%

(Note: File name of OXXXX should not be repeated; otherwise, only 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 CNC4840 IP Addr: It's the IP address of CNC4840 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 CNC4840 controller in accordance with the above format in IpAddr.ini file.

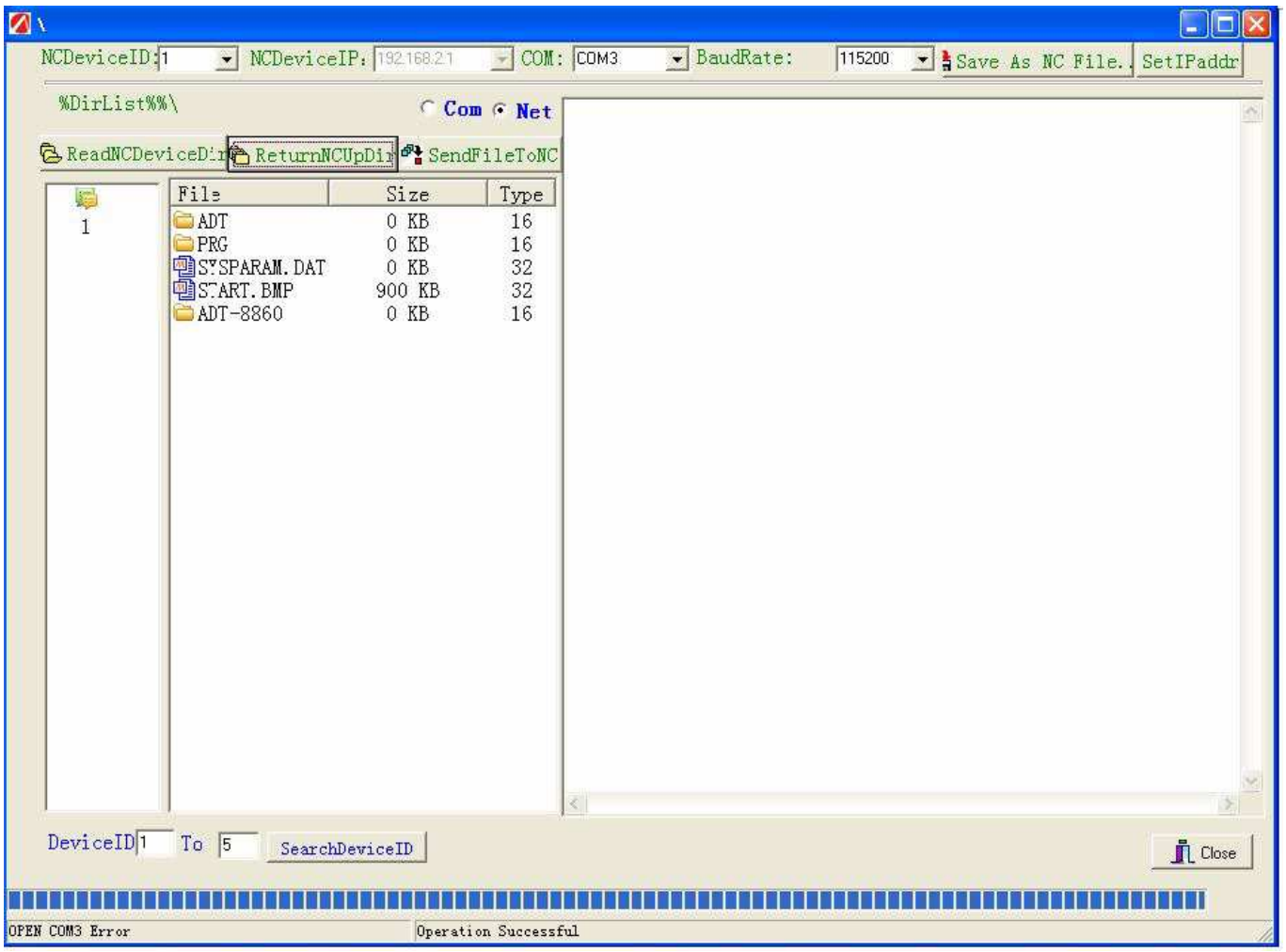
CNC4840 CNC File And Directory: the directory and file name of CNC files that sent to CNC4840 controller (note: the directory file is \PRG\xxxx.CNC, x is number or character, for example 0000.CNC 0001.CNC).

Computer CNC File And Directory: the file directory and name of CNC processing files in local computer that sent to CNC4840 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 CNC4840 controller, but the IP address should be corresponding to that of relevant CNC4840 controller.

3、brief operation methods of Serial network port download application software V1.02

Install the software in PC, then double click Project1.exe to open the software, displayed as below :



This software can manage system files through serial port or network port

When choose com serial port to download, set system axis configuration parameter P3.009 : 0 ; when choose net port to download, set system configuration parameter P3.009 : 1

Get serial port or net port communication line ready, set system relevant serial port Baud rate, controller ID(when multi CNC controllers networking, controllers' ID should be different), network relevant parameters.

Run [SearchDeviceID]

[ReadNCDeviceDir]: list of file directory of chosen CNC controller.

[NCDeviceID]: chosen controller ID

[NCDeviceIP]: chosen controller IP address(when using network transmission)

[COM]: chosen controller's serial port number(when using serial port transmission)

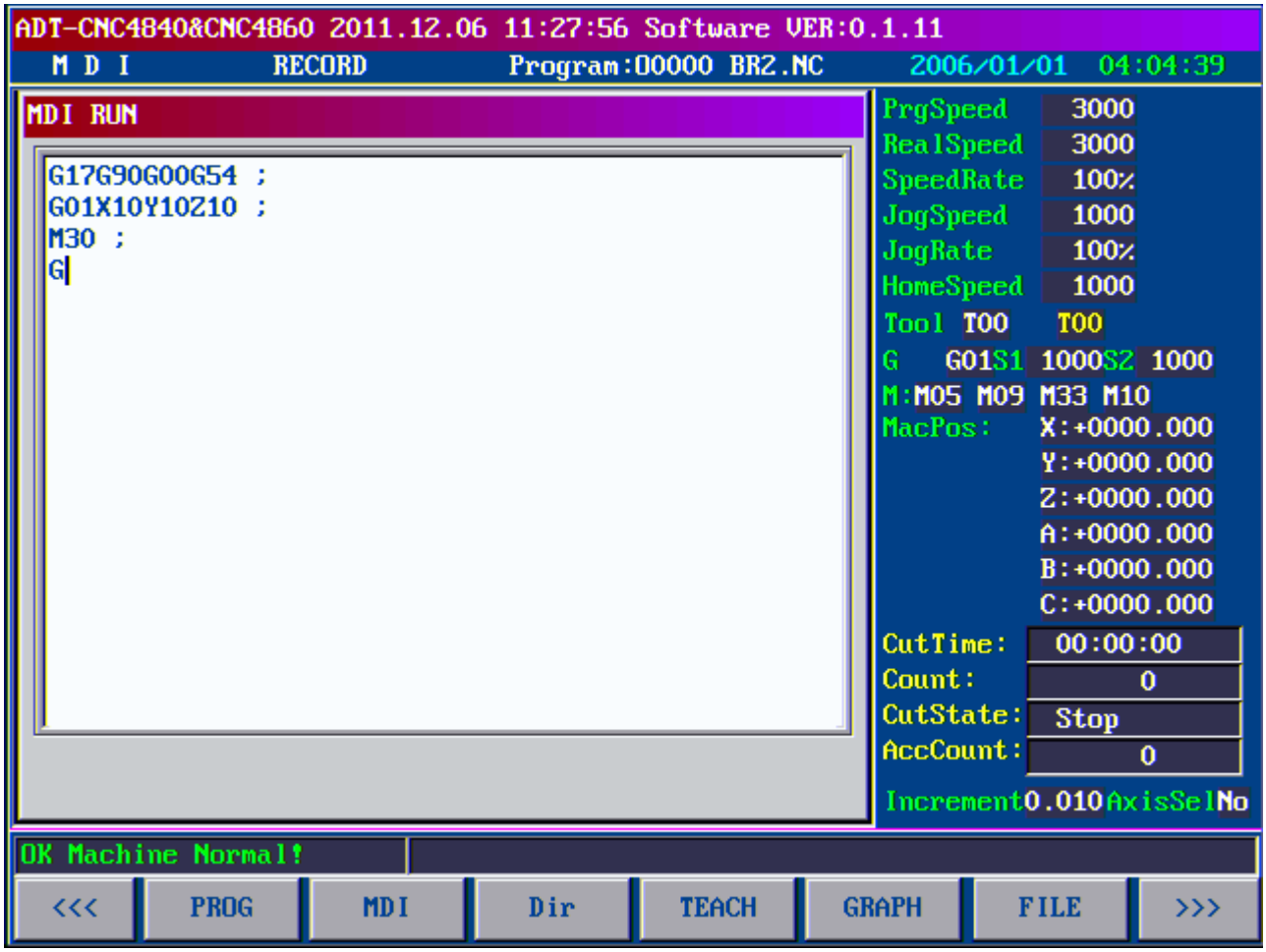
[BaudRate]: chosen controller's serial port communication baudrate(this baudrate should be the same as CNC system synthetical parameter P1.094 setting.)

[SetIPaddr]: controllers not in use, their IP address should be set as 0.

1. 9 Recording operation

MDI operation

In [Record] mode, press [MDI] F2 key to enter the MDI interface, input the words to be executed and press [Insert] 插入 INSERT. After enter all required data, press [Start], the system will then run the program segment. When the operation is stopped, it can also continue executing the MDI operation.



1. 10 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]		Switching between Single Segment and Continuous
			Program started
			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
			M3 (main axis rotated normally)
			M4 (main axis rotated reversely)
			M5 (main axis stops)
[Clear]	[Position]		X axis returned to mechanical zero
			Y axis returned to mechanical zero
			Z axis returned to mechanical zero
			A axis returned to mechanical zero
			B axis returned to mechanical zero
			C axis returned to mechanical zero
[Single-step]	[Position]		Single step increment 0.001 to 1.000 grade switch
work mode switch	[Position]	[输入 INPUT]key	Switch among recording, auto, manual, MPG, single

1.11 Parameters

According to application and function, there are : Synthetical parameter, network parameter, axis config parameter, toolchanger parameter, IO config parameter, manage parameter and so on.

1.11.1 GenralParam (P1.)

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

Setting range : 1～65535

unit : none

permission : operation manager above

preset value : 1

Take effect time : immediately

explain :

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.

013		Axis X's fast rate
014		Axis Y's fast rate
015		Axis Z's fast rate
016		Axis A's fast rate
017		Axis B's fast rate
018		Axis C's fast rate
019		Axis X's start rate

020		Axis Y's start rate
021		Axis Z's start rate
022		Axis A's start rate
023		Axis B's start rate
024		Axis C's start rate
025		Axis X's acceleration
026		Axis Y's acceleration
027		Axis Z's acceleration
028		Axis A's acceleration
029		Axis B's acceleration
030		Axis C's acceleration

Setting range : 1～9999, 1～9999, 1～8000

unit : mm/min, mm/min, mm/sec

permission : operation manager above

Preset value : 3000, 200, 1500

Take effect time : immediately

Explain : This parameter is setting of Trapezoid acceleration and deceleration, used in GOO instruction, and manual and zero motion

About start rate, 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.

031		Axis X's positive soft limit
032		Axis X's negative soft limit
033		Axis Y's positive soft limit
034		Axis Y's negative soft limit
035		Axis Z's positive soft limit
036		Axis Z's negative soft limit
037		Axis A's positive soft limit
038		Axis Z's negative soft limit
039		Axis B's positive soft limit
040		Axis B's negative soft limit
041		Axis C's positive soft limit
042		Axis C's negative soft limit

Setting range : -9999～9999

Unit : mm

permission : Operation manager above

Preset value : ± max

Take effect time : immediately

explain :

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.

043	Feed speed
044	Start feeding speed
045	acceleration of feed
055	Maximum feed speed

Setting range : 1～9999, 1～9999, 1～8000, 1～9999

unit : mm/min, mm/min, mm/sec, mm/min

permission : Operation manager above

Preset value : 3000, 200, 1000, 3000

Take effect time : Immediately

explain : 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.

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.

046	Axis X's reverse gap compensation
047	Axis Y's reverse gap compensation
048	Axis Z's reverse gap compensation
049	Axis A's reverse gap compensation
050	Axis B's reverse gap compensation
051	Axis C's reverse gap compensation

Setting range : 1～20000

unit : Pulse

permission : Operation manager above

Preset value : 0

Take effect time : immediately

explain : Compensating the coordination gap of the machining shaft.

052	Zero

Setting range : 0～1

unit : None

permission : Upon operating administrators

Preset value : 0（Program zero）

Take effect time : Immediately

explain : 0 Program zero

1 Mechanical zero

Program zero means the system is zeroed when the coordinate value becomes zero.

To realize mechanical zero, the external inspection switch is needed to position the zero point. To that effect, the system moves to the set zeroing direction at fast speed, and moves back at low speed when the signal is detected. Then, the system will advance slowly after disconnected upon the detection of signal, and the zeroing process ends till the signal becomes effective again. When the switch is enabled to start by the inspection of servo phase Z in the IO configuration parameters, the mechanical zero mode will automatically start phase Z to position the zero point after the signal from the inspection device reaches.

053	MPG valid mode whether prior

Setting range : 0～1

unit : Unit

permission : Upon operating administrators

Preset value : 0

Take effect time : Take effect after reboot

explain : To set when MPG axis choose valid, whether switch to MPG mode

0:when MPG axis choose valid, not auto switch to MPG mode.

1: when MPG axis choose valid,auto switch to MPG mode.

054	Manual speed

Setting range : 1～9999

Unit : mm/min

permission : Upon operating administrators

Preset value : 1000, 9000

Take effect time : immediately

explain : Set manual speed of XYZABC axis

Start rate and acceleration are up to parameters from 019 to 030

056	M code waiting time

Setting range : 1～9999

Unit : ms

permission : Upon operating administrators

Preset value : 100

Take effect time : Immediately

explain : set persistance time after M code executed

063	**Line number increment**

Setting range : 0～64

unit : None

permission : Upon operating administrators

Preset value : 0

Take effect time : Immediately

explain : When edit G code manual, auto add a Nxxxxx line number when change line.
0 represents prohibiting generate the line number automatically.

064 Maximum speed of spindle (RPM)

Setting range : 0～30000

unit : RPM

permission : Upon operating administrators

Preset value : 6000

Take effect time : immediately

explain : Set max rotation speed of spindle, when S value of process code exceeds this value, this value is used to limit max rotation speed of spindle.

065 Line number of spindle encoder

Setting range : 64～9999

unit : none

permission : Upon operating administrators

Preset value : 1024

Take effect time : immediately

explain : Line number encoder receives when spindle makes 1 revolution.

Like common encoder, can only receive AB phase pulse,encoder line number must be pulse per revolution /4 frequency doubling.

This parameter setting shall influence G74,G84 tapping command, so this parameter must be set correctly.

066	return to zero direction of X axis
067	return to zero direction of Y axis
068	return to zero direction of Z axis
069	return to zero direction of A axis
070	return to zero direction of B axis
071	return to zero direction of C axis

Setting range : 0～1

unit : None

permission : Upon operating administrators

Preset value : 1,1,0,0,0,0

Take effect time : Immediately

explain : It's used to set the machinery to zero direction of X, Y, Z, and ABC axes.

0 Positive direction return to zero

1 Negative direction return to zero

072	**Feed of circular interpolation**

setting range : 0～1

unit : mm

permission : Upon operating administrators

preset value : 0.2

take effect time : immediately

explain : Set Equivalent circular split

If the value is set too small, arc Approximation precision higher, but calculated amount too big, will cause obvious pause when processing, affects processing results.

073	G73 cutter retract value in fixed cycle
074	G83 cutter retract value in fixed cycle

setting range : 0.1～100

unit : mm

permission : Upon operating administrators

preset value : 2.000

take effect time : immediately

explain : Set cutter retract value after Q value feed in G73, G83 command, set according to actual Scraps discharge effect.

075	Auto-home mode configuration (Bit)

setting range : 0～4294967295

unit : none

permission : Upon operating administrators

preset value : 772

take effect time : immediately

explain : 00000000 00000000 00000011 00000100-->772 first Z axis return to zero, then XY axis return to zero at the same time

00000000 00000000 00001011 00000100-->2820 first Z axis return to zero, then XYA axis return to zero at the same time

00000000 00000000 00001101 00000010-->3330 first Y axis return to zero, then XZA axis return to zero at the same time

00000000 00000000 00111011 00000100-->15108 first Z axis return to zero, then XYABC axis return to zero at the same time

076	interpolation speed mode

setting range : 0～1

unit : none

permission : Upon operating administrators

preset value : 0 (accelerate and deaccelerate)

take effect time : immediately

explain : In not preprocess mode (P1.077 set to 0) , this parameter decides accelerate and deaccelerate mode of G01 command.

0: Trapezoid acceleration and deceleration;

1: constant speed;

077	Code processing mode

setting range : 0～2

unit : Unit

permission : Upon operating administrators

preset value : 0 (realtime process)

take effect time : Immediately

explain : 0 Realtime process,suitable for debug machine.

1 Pretreatment mode. Two seconds will be buffered for pre-reading after the System enters the machining state. In this pretreatment mode, only the direction and size of the feeding line segment can be judged. Therefore, the speed can be automatically adjusted to optimize the speed of automatic running. (speed optimize adopts space angle optimize)

2 same as 1(but speed optimize adopts : speed restrain optimize by max feed rate and acceleration)

078	Handwheel reference speed of X axis
079	Handwheel reference speed of Y axis
080	Handwheel reference speed of Z axis
081	Handwheel reference speed of A axis
082	Handwheel reference speed of B axis
083	Handwheel reference speed of Caxis

setting range : 200～30000

unit : none

permission : Upon operating administrators

preset value : 1,1,0,0,0,0

take effect time : immediately

explain : Set axis' turn hand wheel reference speed in MPG mode.

084	X axis zero pulse offset
085	Y axis zero pulse offset
086	Z axis zero pulse offset

087	A axis zero pulse offset
088	B axis zero pulse offset
089	C axis zero pulse offset

setting range : -9999～9999

unit : mm

permission : Upon operating administrators

preset value : 0

take effect time : immediately

explain : Set compensate zero offset after axis return to zero

After machinery zero, then offset the set distance, then set the point as machinery zero.

When Program zero, this parameter is invalid

090	lube oil pump Timing open time (Min)
091	Hold time setting after lube oil pump opens (sec)
092	Point start type pump on/off period (ms)

setting range :

unit :

permission : Upon operating administrators

preset value : 0

take effect time : immediately

explain : ➢ Set the schedule start and hold time parameter of the automatic oil pump of the system

➢ The schedule start parameter starts timing in minute when the system is started, and the oil pump outputs M32(OUT5) when the timing reaches specified value.

➢ The output signal stops (reverse phase) after holding the seconds specified in parameter P1.091

➢ If point start type pump switch cycle is set,then output OUT5 switch signal according to the set switch cycle,if it's set as 0,then output low level.

093	MPG empty running process speed

setting range : 0～10000

unit : Mm/min

permission : Upon operating administrators

preset value : 100

take effect time : immediately

explain : In MPG empty running process mode, execute G code program by turning MPG,set speed of G00、G01、G02、G03 motion command. Axis fast move speed set by synthetical parameter P1.013~P1.018 shall be invalid,feed speed F value set by G01 G02 G03 shall also be invalid.

094	serial port communication baud rate

setting range : 9600～115200

unit : none

permission : Upon operating administrators

preset value : 115200

take effect time : reboot

explain : communication rate setting when DNC or other upper computer software communicates with the controller by RS232.

095	Controller communication equipment ID

setting range : 1～255

unit : None

permission : Upon operating administrators

preset value : 1

take effect time : reboot

explain : Controller’s ID setting when DNC or other upper computer software communicates with controller by MODBUS.

096	Whether reset IO when system alarm

setting range : 0～1

unit : None

permission : Upon operating administrators

preset value : 0

take effect time : immediately

explain : 0:not reset 1: reset

097	Arc speed clamping radius factor(mm)
098	Arc speed clamping speed factor(mm/sec)

setting range : 10～100 10～10000

unit : mm mm/sec

permission : Upon operating administrators

preset value : 50 100

take effect time : immediately

explain : Set clamping G02 or G03 arc track process speed :

MaxCircleF = sqrt(Rel_R/CircSpeed_R)* CircSpeed_V
(Rel_R:G02 G03 radius of arc command, CircSpeed_R: Arc speed clamping radius factor, CircSpeed_V: Arc speed clamping speed factor, MaxCircleF:G02 G03 arc process allowing max feed speed)

In processing when F feed speed of G02 G03 command exceeds MaxCircleF arc process feed speed, to limit G02 G03 arc feed speed. So it won't be too fast to cause deformation and overcut.

099	preprocess buffer section number

setting range : 10～7000

unit : none

permission : Upon operating administrators

preset value : 500

take effect time : immediately

explain : In preprocess mode(P1.007 set as 1) Used to set the codes that the system pre-reads during pretreatment

The higher this value is, the longer the pre-reading time is, and the response may be delayed when just enter G01 instruction

100	arc interpolation mode

setting range : 0～1

unit : none

permission : Upon operating administrators

preset value : 0

take effect time : immediately

explain : G02 G03 interpolation way of arc interpolation command(0:time split,1:pulse step split)

101	X axis return to zero speed
102	Y axis return to zero speed
103	Z axis return to zero speed
104	A axis return to zero speed
105	B axis return to zero speed
106	C axis return to zero speed

setting range : 0～90000

unit : mm/min

permission : Upon operating administrators

preset value : 1000

take effect time : immediately

explain : When X、Y、Z、A、B、C axis return to zero, speed of searching machinery zero.

107	preprocess foresight section numbers

setting range : 10～200

unit : none

permission : Upon operating administrators

preset value : 20

take effect time : immediately

explain : In process mode, To set speed foresight to calculate buffer amount.

108	G00 shift interpolation pattern enable

setting range : 0～1

unit : none

permission : Upon operating administrators

preset value : 0

take effect time : immediately

explain : To set G00 fast move command shift to G01 interpolation command to execute.

109	spare parameters

setting range :

unit : none

permission : Upon operating administrators

preset value : 0

take effect time :

:

110	X axis max restrain acceleration
111	Y axis max restrain acceleration
112	Z axis max restrain acceleration
113	A axis max restrain acceleration
114	B axis max restrain acceleration
115	C axis max restrain acceleration

setting range : 1~90000

unit : mm/sec

permission : Upon operating administrators

preset value : 500

take effect time : immediately

explain : When P1.077 parameter (code process mode) is set as 2, set max acceleration restrain value of speed optimize.

116	X axis max restrain speed
117	Y axis max restrain speed
118	Z axis max restrain speed
119	A axis max restrain speed
120	B axis max restrain speed
121	C axis max restrain speed

setting range : 1~90000

unit : mm/sec

permission : Upon operating administrators

preset value : 10000

take effect time : immediately

explain : When P1.077 parameter (code process mode) is set as 2, set max speed restrain value of speed optimize.

1.11.2 Network parameter(P2.)

001	Local IP address

setting range : 000.000.000.000～255.255.255.255

unit : none

permission : Upon operating administrators

preset value : 192.168.0.176

take effect time : reboot

explain : Set system IP address when network communication. Ip address can't be repeated in LAN.

002	Gateway

setting range : 000.000.000.000～255.255.255.255

unit : None

permission : Upon operating administrators

preset value : 192.168.0.1

take effect time : reboot

explain : When using route to connect to LAN,set route gateway address.

003	subnet mask

setting range : 000.000.000.000～255.255.255.255

unit : None

permission : Upon operating administrators

preset value : 255.255.255.0

take effect time : reboot

:

004	PC IP address

setting range : 000.000.000.000～255.255.255.255

unit : None

permission : Upon operating administrators

preset value : 192.168.0.150

take effect time : Reboot

explain : Upper computer IP address this IP address parameter is not used yet, no need to set.

005	MAC address

setting range	:	000.000.000.000.000.000～255.255.255.255.255.255
unit	:	None
permission	:	Upon operating administrators
preset value	:	192.168.0.150
take effect time	:	reboot
explain	:	MAC(Media Access Control) address, to set network equipment position.

006	Network port number

setting range	:	0～65535
unit	:	None
permission	:	Upon operating administrators
preset value	:	2048
take effect time	:	reboot
explain	:	Use preset value, no need to change.

1.11.3 Axis parameter configuration (P3.)

001	parameter switch open or not

setting range	:	0～1
unit	:	none
permission	:	Upon operating administrators
preset value	:	0
take effect time	:	immediately

explain : Set this parameter to prevent P1.synthetical parameter unwilling change for safe reason.0: P1. synthetical parameter allowed to be changed;1: P1. synthetical parameter not allowed to be changed.

002	use additional panel or not

setting range : 0～1

unit : None

permission : Upon operating administrators

preset value : 0

take effect time : reboot

explain : Set system to use additional panel ADT-FCNC6A or not.0:not use;1: use.

003	Electrical level that indicates alarm of servo axis X is effective
004	Electrical level that indicates alarm of servo axis Y is effective
005	Electrical level that indicates alarm of servo axis Z is effective
006	Electrical level that indicates alarm of servo axis A is effective
007	Electrical level that indicates alarm of servo axis B is effective
008	Electrical level that indicates alarm of servo axis C is effective

setting range : 0～1

unit : LOGIC VOLTAGE LEVEL

permission : Upon operating administrators

preset value : 0

take effect time : immediately

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

009	data communication select

setting range : 0～1

unit : None

permission : Upon operating administrators

preset value : 0

take effect time : reboot

explain : Set communication physical medium of upper computer transmission software,0:RS232 serial port;1:RJ45 ethernet.

010	zero of X effective electrical level
011	zero of Y effective electrical level
012	zero of Z effective electrical level
013	zero of A effective electrical level
014	zero of B effective electrical level
015	zero of C effective electrical level

setting range : 0～1

unit : LOGIC VOLTAGE LEVEL

permission : Upon operating administrators

preset value : 0

take effect time : Immediately

explain : Set the induction effective electrical level of the external machinery zero induction switch when the system returns to machinery zero. .

0 Low electric level

1 High electric level

016	X axis limit effective electric level
017	Y axis limit effective electric level
018	Z axis limit effective electric level

019		A axis limit effective electric level
020		B axis limit effective electric level
021		C axis limit effective electric level
022		X-axis positive limit of hardware enable
023		Y-axis positive limit of hardware enable
024		Z-axis positive limit of hardware enable
025		A-axis positive limit of hardware enable
026		B-axis positive limit of hardware enable
027		C-axis positive limit of hardware enable
028		X-axis negative limit of hardware enable
029		Y-axis negative limit of hardware enable
030		Z-axis negative limit of hardware enable
031		A-axis negative limit of hardware enable
032		B-axis negative limit of hardware enable
033		C-axis negative limit of hardware enable

setting range : 0～1

unit : None

permission : Upon operating administrators

preset value : 0

take effect time : reboot

explain : There are two kinds of modes of hard limit, one is hardware response mode, the other is software scanning mode

- The hardware response mode is self-bring of the motion chip, it is triggered by the effective level of the circuit examination limiting displacement foot, so the real time is very high, but it also brings a disadvantage, when it is disturbed heavily by the outside condition, it will disturb pulse, the system will not alarm due to it has no time to read the fault, it will cause the false appearance of pulse lose; So when using this function normally requires constant-closed wire connection of the connection switch, namely effectiveness is high level; Considering the complexity of the field environment, the default is shuted down.

- The system is equipped with the scan mode and it can no be screened. The scan mode adopts the appointing function visiting input signal, it adopts software anti-jamming examine technology and is able to estimate if whether it is the limiting displacement or disturbing no-action. This need some time to estimate, so real time effective is not better than the stop-type limiting displacement. But most of the time (when the processing speed is 10mm/min), it can satisfy the safety examination

- The hardware response function of the hardware limiting displacement take precedence of scan response function, namely if the hardware response starts, it will quicken the response speed, noteworthiness, the hardware response function only can stop the pulse with immediate-stop mode, so when the speed is very fast, the effectiveness of immediate-stop may cause the vibration of the tool. But the software scanning mode adopt the mode of maximum acceleration decelerating stop, decelerate according to the maximum speed of each axis set by user(parameter P2.074~077), so will be some overshoot phenomenon.

034	X axis pulse direction mode
035	Y axis pulse direction mode
036	Z axis pulse direction mode
037	A axis pulse direction mode
038	B axis pulse direction mode
039	C axis pulse direction mode

setting range : 0～1

unit : None

permission : Upon operating administrators

preset value : 1

take effect time : Reboot

explain : Set pulse direction,if controller pulse direction is opposite to driver direction,can change this parameter to adjust rotate direction of motor.

040	X axis auto zero whether effective
041	Y axis auto zero whether effective
042	Z axis auto zero whether effective
043	A axis auto zero whether effective
044	B axis auto zero whether effective
045	C axis auto zero whether effective

setting range : 0～1

unit : None

permission : Upon operating administrators

preset value : 1,1,1,0,0,0

take effect time : immediately

explain : Set any of XYZABC axis return to zero at the same time whether effective. 1: effective; 0: not effective. When use this parameter to set auto zero, Z axis return to zero first.

046	servo X axis alarming reset efficient level
047	servo Y axis alarming reset efficient level
048	servo Z axis alarming reset efficient level
049	servo A axis alarming reset efficient level
050	servo B axis alarming reset efficient level
051	servo C axis alarming reset efficient level

setting range : 0～1

unit : LOGIC VOLTAGE LEVEL

permission : Upon operating administrators

preset value : 0

take effect time : Immediately

explain : Mtach port parameter of chosen servo driver, specific parameter setting depends on servo driver' s port electric level.

052	The appointing interface axis number of X axis
053	The appointing interface axis number of Y axis
054	The appointing interface axis number of Z axis
055	The appointing interface axis number of A axis
056	The appointing interface axis number of B axis
057	The appointing interface axis number of C axis

setting range : 1～6

unit : pulse interface number

permission : Upon operating administrators

preset value :

take effect time	:	Reboot
explain	:	In the default mode, the real axis number of each coordinate defining axis is corresponded to the silk-screen number of the product shell, when there is some abnormity with some function axis, you can change the axis according to the appointing function. For example, set the P2.045 as 4, set the 4，P2.048 as 1, then the operation of any x axis in the system is the operation of the A axis coding interface in the product shell. 0：have no axis 1～4：corresponding axis one to axes four

058	spindle appointing interface axis number（restart）

setting range	:	0～6
unit	:	pulse interface number
permission	:	Upon operating administrators
preset value	:	
take effect time	:	reboot
explain	:	set it as 0 in the default mode, it stands for the main shaft is the frequency conversion controlling mode, namely it is the analog or shift controlling mode, if you want to use the servo main shaft, then it must occupy a coding interface,(the servo main shaft must be the position controlling mode); Then you can amend this parameter and appoint the function. 0 :the frequency conversion main shaft of the analog adjusting 1～4: correspond to number 1 to number 4 axis **Notice: If you appoint some pulse port as the function port of the main shaft, need to delete the function axis number corresponding to this pulse before or when restart, the system will allocate the main shaft preferentially and the original function will be invalid**

059	X pulse command format
060	Y pulse command format
061	Z pulse command format
062	A pulse command format
063	B pulse command format
064	C pulse command format

setting range : 0～1

unit : None

permission : Upon operating administrators

preset value : 1

take effect time : reboot

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

0 pulse+pulse

1 pulse+direction

065	axis X pulse logic voltage level
066	axis Y pulse logic voltage level
067	axis Z pulse logic voltage level
068	axis A pulse logic voltage level
069	axis B pulse logic voltage level
070	axis C pulse logic voltage level

setting range : 0～1

unit : LOGIC VOLTAGE LEVEL

permission : Upon operating administrators

preset value : 0

take effect time : reboot

explain : set the normal level when the pulse is working, if set a level is different from the normal level which the motor driver requires, then at each time of positive and negative movements(it is nothing to do with the pulse amount), there is an accumulative error in one direction. So you find that the machine has the accumulative error in one direction, please notice whether this parameter is not compatible

071	To enable phase Z inspection at servo axis X
072	To enable phase Z inspection at servo axis Y
073	To enable phase Z inspection at servo axis Z
074	To enable phase Z inspection at servo axis A
075	To enable phase Z inspection at servo axis B
076	To enable phase Z inspection at servo axis C
077	electric level that indicates phase Z of servo axis X is effective
078	electric level that indicates phase Z of servo axis Y is effective
079	electric level that indicates phase Z of servo axis Z is effective
080	electric level that indicates phase Z of servo axis A is effective
081	electric level that indicates phase Z of servo axis B is effective
082	electric level that indicates phase Z of servo axis C is effective

setting : 0～1

unit : LOGIC VOLTAGE LEVEL

permission : Upon operating administrators

preset value : 0

take effect : immediately

explain : After this parameter is set and used, the System will

083	X axis ROUND set value
084	Y axis ROUND set value
085	Z axis ROUND set value
086	A axis ROUND set value
087	B axis ROUND set value
088	C axis ROUND set value

setting range : 0～9999999

unit : Pulse

permission : Upon operating administrators

preset value : 0

take effect time : reboot

explain :

- The loop function only exists in the hardware edition 1.5 or above.
- This functoin can prevent the overflow error that the axis logic counting exceeds the maximum counting range（2147483648）.
- Normally the overflow will happen if set the axis as the rotary axis, so after the system get the P3.089~P3.094 parameter, if it finds user setted the current axis as the rotary axis and adopt 360° displaying mode, then the system calculate the the corresponding pulse threshold value according to the gear ratio of the current axis and endue with the round parameter of the corresponding axis, the user can see the the change of the parameter after start the rotary axis displaying function. User can amend the parameter after been changed, the final displaying numbers are effective.

The parameter can only be carried out after restart; The executing qualification is the corresponding axis must be setted as the rotary axis and set it as 360°displaying （P3.089~P3.094）

089	set X axis feature
090	set Y axis feature
091	set Z axis feature
092	set A axis feature
093	set B axis feature
094	set C axis feature

setting range : 0～1

unit : None

permission : Upon operating administrators

preset value : 1

take effect time : reboot

explain : set the features of the axis.
0：rotary axis
1：linear axis

The setting of this parameter and P3.095～100 corresponding axis will influence the setting of P3.083～088, for details please refer to the parameter instructions of P2.083～088.

095	X axis rotate display mode
096	Y axis rotate display mode
097	Z axis rotate display mode
098	A axis rotate display mode
099	B axis rotate display mode
100	C axis rotate display mode

setting range : 0～1

unit : none

permission : Upon operating administrators

preset value : 0

take effect time : immediately

explain : set the coordinate display mode of axis. This parameter is valid when P3.089～P3.094 is setted as 0.

0：0～360 degree displaying

1：－9999.999～9999.999 degree displaying

The setting of this parameter and P2.089～094 corresponding axis will influence the setting of P3.083～088, for details please refer to the parameter instructions of P3.083～088.

101	the rotary route of X axis optimizing
102	the rotary route of Y axis optimizing
103	the rotary route of Z axis optimizing
104	the rotary route of A axis optimizing
105	the rotary route of B axis optimizing
106	the rotary route of C axis optimizing

setting range : 0～1

unit : None

permission : Upon operating administrators

preset value : 1

take effect time : reboot

explain : when this parameter is valid when P3.089～P3.094 and P3.095～P3.100 is setted as 0; set whether choose automatic search the shortest route moving, if this axis is rotary axis and it does not process during the positiong course, start this function, it can reduce the moving time.

0: do not optimize the route

1: start the shortest route

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

107	direction logic of X axis encoder
108	direction logic of Y axis encoder
109	direction logic of Z axis encoder
110	direction logic of A axis encoder
111	direction logic of B axis encoder
112	direction logic of C axis encoder
113	MPG encoder direction logic

setting range : 0～1

unit : none

permission : Upon operating administrators

preset value : 0

take effect time : immediately

explain : When the logic direction the coder gets is opposite to the real moving direction of the axis, you can set this parameter and set in the same direction.
MPG encoder reuse A axis encoder
Spindle encoder reuse X axis encoder
0: positive direction
1: negative direction

1.11.4 Tool magazine parameter (P4.)

001	system amount of cutting tool
002	the speed of cutting tool change(mm/min)
003	the interval of cutter spacing(mm or degree)
004	safe height of cutting tool change（mm）
005	cutting tool changing benchmark speed X(mm)
006	cutting tool changing benchmark speed Y(mm)
007	cutting tool changing benchmark speed Z(mm)
008	Z axis put tool coord(mm)
009	Z axis fetch tool coord (mm)
010	current tool number
011	next ready tool number

setting :

unit :

permission : Upon operating administrators

preset value :

take effect : immediately

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

1.11.6 IO Configuration parameter(P5.)

001	IN00 OUT00
…	…
059	IN59 OUT59

setting range : Input mapping:0～55 input IN detect elctric level:0～1 output mapping:0～53

Reset enable:0～1

unit : None

permission : Upon operating administrators

preset value : Refer to port table

take effect time : immediately

explain :

1. input mapping:control system input (IN)IO object.
2. input IN detect elctric level:specific to all system's IO input IN00～IN55 detect effective electric level.0: low electric level valid; 1:high electric level valid.
3. reset enable:specific to all system IO output OUT00～OUT53,whether reset output IO when system alarm or push [reset] button.1:reset corresponding IO output; 0: not reset corresponding IO output.

1.11.7 Manager Parameter (P6.)

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

setting range : None

unit : None

permission : None

preset value : None

take effect time : immediately

explain :

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

004	The initialing comprehensive parameters are the factory defaults
005	The initializing axis configuration network parameters are the factory defaults

setting range : None

unit : None

permission : Superuser

preset value : None

take effect time : immediately

explain : Only in superuser mode, can the axis config and network parameter table be initialized.

006	to backup synthetical parameter SYSPARAM.DAT
007	recover synthetical parameter from SYSPARAM.DAT

setting range : None

unit : None

permission : Superuser

preset value : none

take effect time : reboot

explain :

1. You can do the parameter back-up and retrieval only under the superuser mode.
2. The parameter back-up is the SYAPARAM.DAT file appointed under the root directory of the controller electronic disc, if it has existed a cognominal file in it, then the last back-up will bestrow this file.
3. The bestrow is also carried out under the root directory of the controller electronic disc according to the SYSPARAM.DAT file. When it is bestrowed, it will judge whether it belongs to the same parameter edition according to the parameter edition number of the system back-up, if it is not the same edition, then it will not carry out the retrieval to the parameter.
4. After recover, system needs to be reboot to make recoverd parameters valid.

008	backup axis config parameter to KNFPARAM.DAT
009	recover synthetical parameter from KNFPARAM.DAT

setting range : None

unit : None

permission : Superuser

preset value : None

take effect time : reboot

explain : 5. You can do the axis config and network parameter back-up and retrieval only under the superuser mode

6. The parameter back-up is the KNFPARAM.DAT file appointed under the root directory of the controller electronic disc, if it has existed a cognominal file in it, then the last back-up will bestrow this file.

7. The bestrow is also carried out under the root directory of the controller electronic disc according to the KNFPARAM.DAT file. When it is bestrowed, it will judge whether it belongs to the same parameter edition according to the parameter edition number of the system back-up, if it is not the same edition, then it will not carry out the retrieval to the parameter.

8. After recover, system needs to be reboot to make recoverd parameters valid.

010	initialize SRAM parameter factory default
010	backup SRAM parameter to SRAMDATA.DAT
012	Recover SRAM parameter from SRAMDATA.DAT

setting range : None

unit : None

permission : Superuser

preset value : None

take effect time : reboot

explain : 9. You can do the SRAM parameter initialize, back-up and retrieval only under the superuser mode

10. The parameter back-up is the SRAMDATA.DAT file appointed under the root directory of the controller electronic disc, if it has existed a cognominal file in it, then the last back-up will bestrow this file.

11. The bestrow is also carried out under the root directory of the controller electronic disc according to the SRAMDATA.DAT file. When it is bestrowed, it will judge whether it belongs to the same parameter edition according to the parameter edition number of the system back-up, if it is not the same edition, then it will not carry out the retrieval to the parameter

12. After recover, system needs to be reboot to make recoverd parameters valid.

013	generate password file

setting range : None

unit : None

permission : None

preset value : None

take effect time : immediately

explain : If user forget password, can use this function to generate PassMeg.DAT file, give this file to ADTECH, ADTECH can restore the password.

014	Import CSV system configuration table

setting range : none

unit : None

permission : None

preset value : none

take effect time : immediately

explain : Import the CSV system configuration table of the manufacturer into the system.

015	Clear current processing number
016	Clear accumulated processing number
017	accumulated process max limit number (0 is infinite)

setting range : None

unit : None

permission : None

preset value : None

take effect time : immediately

explain : P6.015 P6.016:clear current processing number or accumulated processing number.

P6.017 accumulated process max limit number:when set value is bigger than 0,then program can't be processed when accumulated processing number exceeds the set value,after accumulated processing number,program can be executed again.

018	System language pack

setting range : 0～1

unit : None

permission : Operation admin

preset value : 0 (Chinese)

take effect time : Reboot

explain : Select system language

0 Chinese

1 English

1.12 System alarming

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

G code program runs alarming information

system condition alarming information

1.12.1 NC Program executing alarming

0000	:	system not reset or XYZ axis not return to zero
0001	:	program over
0002	:	not specify movement function
0003	:	no G code to get line code function
0004	:	cutting tool changing failure
0005	:	cutting tool invalid
0006	:	G program segment repeat error
0007	:	G program segment program number error
0008	:	G7x8x compound command code can not run normally
0009	:	specified port number error
0010	:	program abend error
0011	:	appoint the M01 code program halt
0012	:	specified program number doesn’t exist
0013	:	M98 format error
0014	:	call motion executing failure
0015	:	this segment does not need compensation
0016	:	G program segment invalid format
0017	:	M99 command call abnormity, forbidden in current occasion
0018	:	movement abnormity alarming
0019	:	illegal character
0020	:	annotation mark format error or no symmetrical annotation mark
0021	:	illegal G code
0022	:	the radius offset number of G code error or value error
0023	:	undefined G code radius offset error
0024	:	arc programming error
0025	:	appointing illegal plane exceeds G17,G18,G19
0026	:	M98 calling error, probably exceeds the maximum value
0027	:	spindle appointing hardware axis number error
0028	:	M code executing error
0029	:	spindle appointing failure
0030	:	moving repeat request
0031	:	appointing arc does not exist
0032	:	lack X command error
0033	:	lack Y command error
0034	:	lack Z command error
0035	:	lack A command error
0036	:	lack B command error
0037	:	lack C command error

0038	:	lack D command error
0039	:	lack R command error
0040	:	lack F command error
0041	:	lack T command error
0042	:	lack S command error
0043	:	lack P command error
0044	:	lack M command error
0045	:	lack G command error
0046	:	lack I command error
0047	:	lack J command error
0048	:	lack K command error
0049	:	lack Q command error
0050	:	screw pitch value repeating appointing error
0051	:	system alarming and quit abnormally
0052	:	quit by man-interrupting
0053	:	no appointing G code parameter source
0054	:	non appointing G code program number sheet storage address
0055	:	macro function call error
0056	:	macro expression abnormal write
0057	:	macro variable address error
0058	:	illegal variable value
0059	:	skip sentence error
0060	:	macro loop sentence pair error
0061	:	macro loop sentence nest error
0062	:	macro loop sentence nest call too much, exceeds max layer numbers
0063	:	not define macro variable to get address function.
0064	:	user self define information
0065	:	user self define alarm
0066	:	const quote error
0067	:	last track i_gcode feature value error
0068	:	next track i_gcode feature value error
0069	:	start compensation point arc entering in
0070	:	end compensation point arc exit
0071	:	radius compensate last track starting point overlap end point
0072	:	radius compensate next track starting point overlap end point
0073	:	radius compensation value exceeds G0203 R value
0074	:	not supportive codes occur in radius compensation
0075	:	NURBS pitch point too many
0076	:	NURBS parameter error
	:	
	:	
	:	
	:	
	:	
	:	

1.12.2 system environment alarming

0001	:	Emergency stop

hand-held box interface emergency stop button is effective
External emergency stop input valid, detect whether IO assign has conflict or interference.
See the corresponding function interface in IO config, you can see it in the input diagnoses

When system emergency stop is valid, system G code program execute , axis movements, all operations stop immediately, until emergency alarm removed.

0002	:	X axis positive direction hard limit
0003	:	X axis positive direction soft limit
0004	:	X axis negative direction hard limit
0005	:	X axis negative direction soft limit
0006	:	Y axis positive direction hard limit
0007	:	Y axis positive direction soft limit
0008	:	Y axis negative direction hard limit
0009	:	Y axis negative direction soft limit
0010	:	Z axis positive direction hard limit
0011	:	Z axis positive direction soft limit
0012	:	Z axis negative direction hard limit
0013	:	Z axis negative direction soft limit
0014	:	A axis positive direction hard limit
0015	:	A axis positive direction soft limit
0016	:	A axis negative direction hard limit
0017	:	A axis negative direction soft limit
0018	:	B axis positive direction hard limit
0019	:	B axis positive direction soft limit
0020	:	B axis negative direction hard limit
0021	:	B axis negative direction soft limit
0022	:	C axis positive direction hard limit
0023	:	C axis positive direction soft limit
0024	:	C axis negative direction hard limit
0025	:	C axis negative direction soft limit

1. the system gives the limit alarming corresponding to the hint, examine the corresponding limit induction point or parameter.
2. If it is hard limiting displacement alarm, there is no problem with the induction point visual examination, then enter the diagnoses mode under the hand-mode, check the input port condition under the diagnoses mode, if the condition is effective, then eliminate it in turn, now pull out IO line, check whether the induction disappears, if it disappears then check the lines, if it still exists, the inside optical coupler maybe is destroyed, please contact the supplier.
3. If P1.031～P1.042 ±soft limit route limit value is set,when axis machinery coord moving value exceeds the set value,reach route limit alarm will happen.
4. No matter hard limit alarm or soft limit alarm,move corresponding axis in opposite direction and then push [reset] button to remove alarm in manual mode or handwheel mode.

0026	:	tool setting instrument limit

When system auto tool set,doesn't detect tool setting point,but detect limit point of tool set instrument,tool setting instrument limit alarm will happen to end auto tool setting,to prevent damage to tool setting instrument.

0026	:	spindle alarm

Spindle inverter gives alarm signal to system when spindle rotation speed too high or other reasons,system shall end program execution immediately.

0027	:	X axis servo alarm
0028	:	Y axis servo alarm
0029	:	Z axis servo alarm
0030	:	A axis servo alarm
0031	:	B axis servo alarm
0032	:	C axis servo alarm

1. If servo alarming, but actually the servo does not give the alarming, it is probably that the setting of parameter P3.003~008 is opposite to the servo real alarming level, change the parameter
2. the corresponding function interface is IN66～IN71, you can see it in the input diagnoses

0033	:	air pressure alarm

When machine pneumatic parts detect insufficient air pressure,gives alarm signal to system.

0034	:	system clamp material alarm

When machine clamp parts detect not tight clamp of work material,gives alarm signal to system.

1.2 Annex1 setting of workpiece coordinate and tool setting

When using CNC4840 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.

1.3 Annex 2 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%

1.4 Annex3 **Description of keyboard**

The keyboard of CNC4840 is divided into two areas, edition area and operation area. The following table is the description of all keys:

S/N	Name	Purpose
1	上页 PU 下页 PD	having different functions in different display modes
2	EOB	To insert a EOB when the input buffer storage has no data, and confirm the modification of parameter in parameter interface
3	Address G R /Figure 6 SP	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] 位置 POS	Select [Position] interface
6	[Program] 程序 PROG	Select [Program] and [File management] interfaces
7	[Offset] 偏移量 OFFSET	Select [Too compensation] interface
8	[System] 参数 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

Annex Product specifications

Function	Name		Specification
Controllable axes	Controlled axis		4axis (X，Y，Z，A)
	Simultneous controllable axes number		4 axes linear interpolation 2 axes arc interpolation
Input command	Min setting unit		0.001mm
	Min move unit		0.001mm
	Max instruction value		±9999.999 mm
Feed	fast feedrate		X-axis、Y-axis、Z-axis、A-axis:9999mm/min（max）
	range	feed per minute	1～9999 mm/min
		feed per rotate	0.0001～500.0000 mm/r

	Auto acc and dec speed	Yes
	feed speed rate	10～150%
Hand	Hand continuous feeding	Yes
	Reference point for manual return	one or three axes return to return to reference point simultaneously
	single step /handwheel function	Yes
Interpolation	Location,Linear,Full cycle arc	G00,G01,G02/G03
Operation mode	MDI,automation,manual,single step,edit	Yes
Commissioning function	Trial running,single program,hand wheel	Yes
Coordinate system and pause	Pause(sec/microsecond)	G04 X/P_
	coordinate system setting	G92
	Auto coordinate system setting	Yes
safety function	software limit check	Yes
	sudden stop	Yes
Memory	program storage capacity and quantity	Total capacity: 32M bytes; 512 working areas; 4M bytes each working area, can store 10000 programs.
Program edit	program edit	Insert,modification,delete,cancel
	program number,sequence,address, Character retrieving	Yes
	decimal point programming	Yes
	Position /program Cutter compensation/alarm /diagnosis Parameter/image emulation	Yes

M，S，T function	assistant function	M2 digit
	spindle function	S2 digit (level control) S3~5 digit （analog）
	Tool function	T01~18
Compensation Function	Memory for cutter compensation	±7 digit 36 sets
	Reverse gap compensation	Yes
Others function	Start program at any position	Yes
	Designating arc radius R	Yes
	Electronic gear ratio	Yes